



Tekla Structures 2019

Create models

March 2019

©2019 Trimble Solutions Corporation



Contents

1	Get to know Tekla Structures basic working methods.....	17
1.1	Set up the workspace.....	17
	Change units and decimals.....	18
	Work with grids.....	18
	Create, modify, or delete a grid.....	21
	Add a single grid line.....	26
	Modify a single grid line.....	26
	Delete a single grid line.....	29
	Work with views.....	29
	Move the view plane.....	31
	Create model views.....	31
	Open, save, modify, or delete a view.....	41
	Switch between views.....	43
	Update and refresh views.....	43
	View properties.....	44
	Grid view properties.....	45
	Define the work area.....	46
	Fit work area to entire model.....	47
	Fit work area to selected parts.....	47
	Fit work area using two points.....	47
	Hide the work area box.....	47
	If you cannot see all objects.....	48
	Coordinate system.....	48
	Right-hand rule.....	50
	Show or hide the work plane grid.....	50
	Shift the work plane.....	51
	Base points.....	53
	Select the work plane.....	63
	Change the color settings.....	64
	Find RGB values for colors.....	64
	Change the model background color	64
	Change the color of dimensions, part labels, and bolts.....	66
	Change the rendering of the model.....	67
1.2	Zoom and rotate the model.....	75
	Zoom in and out.....	75
	Rotate the model.....	76
	Pan the model.....	77
1.3	Snap to positions.....	77
	Snapping toolbar.....	78
	Snap zone.....	79
	Snap priority.....	79
	Snap depth.....	79
	Snapping in drawings.....	79
	Snap switches and symbols.....	80
	Snap in orthogonal directions.....	82

	Snap to orthogonal points.....	82
	Snap in orthogonal direction relative to previously picked points.....	83
	Snap to lines.....	85
	Snap to a line.....	85
	Snap to extension lines.....	85
	Snap to points using exact distance or coordinates - numeric snapping.....	87
	Enter a distance or coordinates.....	87
	Snapping example: Track along a line towards a snap point.....	88
	Options for coordinates.....	90
	Change the snapping mode.....	91
	Snapping aids.....	92
	Create a temporary reference point.....	92
	Lock X, Y, or Z coordinate on a line.....	94
	Align objects using a snap grid.....	94
	Snap settings.....	94
1.4	Work with model objects in Tekla Structures.....	95
	Create model objects and modify model object properties by using property pane...	96
	Create or delete a model object.....	97
	Modify model object properties by using property pane.....	97
	Modify the common properties of different object types by using property pane...	
100	Resize and reshape model objects.....	103
	Copy properties from another object.....	109
	Copy model object properties using the property pane.....	109
	Copy object properties using the contextual toolbar.....	111
	Save and load object properties.....	111
	Save and load properties in the property pane.....	111
	Save and load properties in a dialog box.....	113
	Remove existing properties.....	114
	Undo modeling and drawing changes	114
1.5	Select objects.....	116
	Select single objects.....	116
	Select multiple objects using area selection.....	117
	Select all objects.....	117
	Select previous objects.....	118
	Select objects by identifier.....	118
	Select handles.....	120
	Modify the selection.....	122
	Selecting toolbar.....	122
	Select assemblies, cast units, and nested objects.....	127
	Select assemblies and cast units.....	127
	Select nested objects.....	127
	Select reference models, reference model objects and assemblies.....	128
	Select an entire reference model.....	128
	Select a reference model object.....	129
	Select a reference model assembly.....	129
	Tips for selecting objects.....	129
	Switch rollover highlight on or off.....	129
	Select on right-click.....	130
	If you cannot select objects.....	130
	Interrupt object selection.....	131
1.6	Copy and move objects.....	131
	Copy objects.....	133
	Copy by picking two points.....	133

	Copy linearly.....	134
	Copy by specifying a distance from origin.....	135
	Copy using drag-and-drop.....	135
	Copy objects to another object.....	136
	Copy all content to another object.....	137
	Copy to another plane.....	137
	Copy from another model.....	138
	Copy objects using linear array tool.....	138
	Copy objects using radial array tool.....	141
	Copy objects using Array of objects (29) component	143
	Move objects.....	144
	Move by picking two points.....	145
	Move linearly.....	146
	Move by specifying a distance from origin.....	147
	Move using drag-and-drop.....	147
	Move to another plane.....	149
	Move objects to another object.....	149
	Rotate objects.....	150
	Rotate around a line.....	150
	Rotate around the z axis.....	151
	Rotate drawing objects.....	153
	Rotation settings.....	154
	Mirror objects.....	154
	Mirror model objects.....	154
	Mirror drawing objects.....	155
1.7	Filter objects.....	155
	Use existing filters.....	156
	How to use a view filter.....	156
	How to use a selection filter.....	158
	Create new filters.....	159
	Create a view filter.....	159
	Create a selection filter.....	161
	Create a drawing filter.....	161
	Create a drawing view filter.....	164
	Create a drawing selection filter.....	166
	Filtering techniques.....	166
	Object properties in filtering.....	169
	Template attributes in filtering.....	185
	Wildcards	186
	Examples of filters.....	186
	Filter parts based on their name.....	186
	Filter main parts.....	187
	Filter bolts based on their size.....	188
	Filter parts based on their assembly type.....	188
	Filter sub-assemblies.....	190
	Filter reference model objects.....	190
	Filter parts within component.....	191
	Filter reinforcement in pour units based on pour object type.....	191
	Filter all contents of a pour unit.....	192
	Copy and remove filters.....	193
	Copy a filter to another model.....	193
	Remove a filter.....	194
	Select values from the model.....	194
1.8	Customize the basic user interface elements	194
	Customize the keyboard shortcuts.....	195

	Define new keyboard shortcuts.....	195
	Clear and reset shortcuts.....	197
	Export keyboard shortcuts.....	197
	Import keyboard shortcuts.....	197
	Customize the ribbon.....	198
	Add a button to the ribbon.....	199
	Move a button.....	205
	Resize a button.....	205
	Change the appearance of a button.....	206
	Create a user-defined command with Command editor.....	208
	Add a separator bar.....	210
	Remove a button.....	210
	Add, hide, and edit tabs.....	210
	Save the ribbon	211
	Check the changes.....	212
	Back up and restore ribbons.....	213
	Customize the property pane layout.....	213
	Add a property or a property group.....	215
	Change the name of a property or a property group.....	217
	Copy properties from one object type to another object type	218
	Remove a customization.....	219
	Save the changes.....	219
	User-defined attributes (UDAs) in the customized property pane.....	219
	Example: How to add IFC related user-defined attributes to the property pane layout and copy them to another object type.....	220
	Customize the Selecting, Snapping, and Snap override toolbars	225
	Customize the contextual toolbar.....	226
	Customize contextual toolbar.....	226
	Create user profiles for contextual toolbars.....	228
	Back up and share contextual toolbars.....	228
1.9	Tips for large models.....	229
1.10	Create model templates.....	231
	Create a new model template.....	231
	Modify an existing model template.....	232
	Download model templates.....	233
	Model template options.....	233
2	Create parts, reinforcement, and construction objects.....	234
2.1	Create parts and modify part properties.....	235
	Create a steel column.....	237
	Modify steel column properties.....	237
	Steel column properties.....	238
	Create a steel beam.....	239
	Modify steel beam properties.....	240
	Steel beam properties.....	240
	Create a steel polybeam.....	242
	Modify steel polybeam properties.....	242
	Steel beam properties.....	242
	Create a curved beam.....	244
	Modify curved beam properties.....	245
	Steel beam properties.....	245
	Create a twin profile.....	246
	Modify twin profile properties.....	247
	Twin profile properties.....	247

	Create an orthogonal beam.....	249
	Modify orthogonal beam properties.....	249
	Orthogonal beam properties.....	249
	Create a steel spiral beam.....	251
	Basic concepts related to spiral beams.....	251
	Create a spiral beam.....	251
	Limitations.....	253
	Create a contour plate.....	253
	Create a round contour plate.....	254
	Modify contour plate properties.....	255
	Contour plate properties.....	255
	Create a conical or a cylindrical bent plate.....	256
	Create a cylindrical bent plate.....	257
	Create a conical bent plate.....	260
	Modify the bend radius.....	263
	Modify the shape of a bent plate.....	265
	Remove curved sections.....	269
	Examples.....	270
	Create a stand-alone bent plate.....	271
	Modify the shape of a stand-alone bent plate.....	274
	Create a concrete column.....	277
	Modify concrete column properties.....	278
	Concrete column properties.....	278
	Create a concrete beam.....	279
	Modify concrete beam properties.....	280
	Concrete beam properties.....	280
	Create a concrete polybeam.....	282
	Modify concrete polybeam properties.....	283
	Concrete beam properties.....	283
	Create a concrete spiral beam.....	285
	Basic concepts related to spiral beams.....	285
	Create a spiral beam.....	286
	Limitations.....	287
	Create a concrete panel or wall.....	288
	Modify concrete panel or wall properties.....	289
	Concrete panel or wall properties.....	289
	Create a concrete slab.....	290
	Create a round concrete slab.....	291
	Modify concrete slab properties.....	292
	Concrete slab properties.....	292
	Create a pad footing.....	293
	Modify pad footing properties.....	294
	Pad footing properties.....	294
	Create a strip footing.....	295
	Modify strip footing properties.....	296
	Strip footing properties.....	296
	Create items.....	298
	Create an item or a concrete item.....	299
	Modify item or concrete item properties.....	300
	Change the shape of an item.....	301
	Item and concrete item properties.....	301
2.2	Adjust part position and show part information.....	303
	Show part handles and part reference lines in a model view.....	303
	Show part handles.....	303
	Show part reference lines in a model view.....	306

	Modify the position of a part.....	306
	Part position on the work plane.....	308
	Part rotation.....	309
	Part position depth.....	310
	Part vertical position.....	312
	Part horizontal position.....	313
	Part end offsets.....	315
	Select and change the profile or material of a part.....	316
	Select and change the profile of a part.....	316
	Select and change the material of a part.....	318
	Examples of user-defined attributes (UDAs) for parts.....	319
	Show part information by using part labels.....	320
	Create curved parts.....	321
	Create horizontal parts.....	322
	Create beams close to each other.....	323
	Position columns, pad footings, and orthogonal beams.....	323
	How to model identical areas.....	324
2.3	Modify parts.....	325
	Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts.....	325
	Define default adaptivity settings.....	325
	Modify the adaptivity of an individual model object.....	326
	Split parts.....	326
	Split a straight or curved part or polybeam.....	326
	Split a plate or slab using a polygon.....	326
	Combine parts.....	327
	Attach parts to each other.....	327
	Attach a part to another part.....	328
	Detach an attached part.....	328
	Explode attached parts.....	329
	Warp a part.....	329
	Warp a beam or a column using deformation angles.....	329
	Warp a concrete slab by moving chamfers.....	330
	Warp a Floor Bay (66) slab.....	330
	Camber a part.....	331
2.4	Add details to parts.....	332
	Create bolts.....	332
	Create a bolt group.....	332
	Create a single bolt.....	333
	Create bolts using the Auto bolt component.....	333
	Create a bolt group by exploding a component.....	336
	Change or add bolted parts.....	337
	Bolt group shape.....	337
	Bolt properties.....	338
	Create studs.....	342
	Create holes.....	343
	Create round holes.....	343
	Create oversized holes.....	344
	Create slotted holes.....	345
	Create welds.....	346
	Create a weld between parts.....	346
	Create a weld to a part.....	347
	Create a polygon weld.....	348
	Weld properties.....	349
	List of weld types.....	354

	Welds in components.....	356
	Weld preparation.....	357
	Set the visibility and appearance of welds.....	358
	Change a weld to a polygon weld.....	359
	Split a polygon weld.....	360
	Create user-defined cross sections for welds.....	360
	Create fittings.....	361
	Create cuts.....	362
	Cut parts with a line.....	362
	Cut parts with a polygon.....	363
	Cut parts with another part.....	364
	Hide cut lines in a model view.....	365
	Tips on how to cut efficiently.....	366
	Polygon cut properties.....	366
	Part cut properties.....	367
	Create part chamfers.....	368
	Chamfer part corners.....	368
	Chamfer part edges.....	369
	Corner chamfer properties.....	370
	Edge chamfer properties.....	372
	Add surface treatment to parts.....	373
	Add surface treatment to an entire part face.....	374
	Add surface treatment to a selected area on a part face.....	374
	Add surface treatment to all faces of a part.....	375
	Add surface treatment to cut faces.....	375
	Surface treatment on chamfered parts.....	375
	Surface treatment on parts with openings and recesses.....	376
	Modify surface treatment properties.....	376
	Surface treatment properties.....	377
	Define new surface treatment subtypes.....	378
	Tiled surface treatment.....	379
	Create an unpainted area using the No paint area component.....	383
	Add surfaces to part faces and pour object faces.....	385
	Add a surface to a face.....	386
	Modify surface properties.....	386
2.5	Create assemblies.....	386
	Create an assembly.....	387
	Create a sub-assembly.....	387
	Use bolts to create assemblies.....	388
	Bolt sub-assemblies to an existing assembly.....	388
	Use welds to create assemblies.....	389
	Weld sub-assemblies to an existing assembly.....	389
	Add objects to assemblies.....	390
	Assembly hierarchy.....	390
	Add parts to an assembly.....	391
	Create a nested assembly.....	392
	Join assemblies.....	392
	Change the assembly main part.....	392
	Change the main assembly.....	393
	Remove objects from an assembly.....	393
	Check and highlight objects in an assembly.....	393
	Explode an assembly.....	394
	Assembly examples.....	394
2.6	Create cast units.....	396
	Define the cast unit type of a part.....	396

	Create a cast unit.....	397
	Add objects to a cast unit.....	397
	Change the cast unit main part.....	398
	Remove objects from a cast unit.....	398
	Check and highlight objects in a cast unit.....	398
	Explode a cast unit.....	399
	Casting direction.....	399
	Define the casting direction of a part.....	400
	Show the top-in-form face.....	401
2.7	Manage pours.....	402
	Enable pour management.....	402
	Disable pour management temporarily.....	403
	View cast-in-place concrete structures.....	404
	Set the appearance of cast-in-place concrete structures.....	404
	Part view vs. pour view.....	406
	Define the pour phase of a part.....	407
	Pour objects.....	408
	Change the color and transparency of pour objects.....	409
	Modify the properties of a pour object.....	410
	Pour units.....	410
	Calculate pour units.....	411
	Check and inquire objects in a pour unit.....	411
	Add objects to a pour unit.....	412
	Remove objects from a pour unit.....	413
	Reset pour unit relations.....	413
	Modify the properties of a pour unit.....	414
	How Tekla Structures automatically adds objects to pour units.....	414
	Pour breaks.....	415
	Pour break adaptivity.....	417
	Set the visibility of pour breaks.....	417
	Create a pour break.....	418
	Select a pour break.....	420
	Copy a pour break.....	420
	Move a pour break.....	420
	Modify a pour break.....	421
	Remove a pour break.....	422
	Troubleshoot pours.....	422
	Example: Create concrete geometry and work with pours.....	425
2.8	Create reinforcement.....	427
	Create a rebar set.....	428
	Basic concepts related to rebar sets.....	428
	Create longitudinal rebars.....	430
	Create crossing rebars.....	432
	Create planar rebars.....	435
	Create rebars by point input.....	437
	Rebar set properties.....	438
	Limitations.....	438
	Create a rebar set using Rebar shape placing tool.....	438
	Examples: Rebar sets in curved structures.....	445
	Create a single reinforcing bar.....	451
	Create a reinforcing bar group.....	452
	Create a reinforcing bar group using Rebar shape catalog.....	454
	Create a curved reinforcing bar group.....	461
	Create a circular reinforcing bar group.....	463
	Create a tapered or spiral reinforcing bar group.....	465

	Create a reinforcement mesh.....	467
	Create a rectangular reinforcement mesh.....	468
	Create a polygonal reinforcement mesh.....	469
	Create a bent reinforcement mesh.....	471
	Create a customized reinforcement mesh.....	473
	Create a reinforcement strand pattern.....	473
	Debond reinforcement strands.....	475
	Create a reinforcement splice.....	476
2.9	Modify reinforcement.....	477
	Modify a rebar set.....	478
	Modify the properties of a rebar set.....	478
	Change the layer order of a rebar set.....	479
	Modify a rebar set using guidelines.....	479
	Modify a rebar set using leg faces.....	480
	Modify a rebar set locally using modifiers.....	486
	How to cut rebar sets.....	493
	Distribute bars in a rebar set.....	494
	Modify a single reinforcing bar, bar group, or mesh.....	496
	Distribute bars in a reinforcing bar group.....	500
	Delete bars from a reinforcing bar group.....	502
	Ungroup a reinforcement.....	503
	Group reinforcement.....	504
	Combine two reinforcing bars or reinforcing bar groups into one.....	505
	Split a reinforcing bar group	506
	Use handles to modify a reinforcement.....	507
	Add hooks to reinforcing bars.....	508
	Define the reinforcement cover thickness.....	510
	Use adaptivity to modify a reinforcement.....	512
	Attach a reinforcement to a concrete part.....	514
	Check the validity of reinforcement geometry.....	514
	Split and splice a reinforcement.....	515
	Assign running numbers to reinforcement.....	517
	Classify reinforcement to layers.....	517
	How to calculate the reinforcing bar length.....	518
	How to calculate the reinforcing bar leg length.....	521
	Reinforcement shape recognition.....	523
	Rebar shape manager in reinforcement shape recognition.....	524
	Hard-coded bending type identifiers in reinforcement shape recognition.....	535
	Reinforcement in templates.....	559
2.10	Create construction objects and points.....	561
	Create a construction line.....	562
	Create a construction plane.....	563
	Create a construction circle.....	563
	Create a construction arc.....	564
	Create a construction polycurve.....	565
	Copy a construction object with offset.....	566
	Modify a construction object.....	567
	Create points.....	570
	Create points on a line.....	571
	Create points on a plane.....	571
	Create points parallel to two points.....	572
	Create points along the extension line of two points.....	573
	Create projected points on a line.....	574
	Create points along an arc using center and arc points.....	574
	Create points along an arc using three arc points.....	575

	Create points tangent to a circle.....	575
	Create points at any position.....	576
	Create bolt points.....	576
	Create points at the intersection of two lines.....	577
	Create points at the intersection of a plane and a line.....	577
	Create points at the intersection of a part and a line.....	577
	Create points at the intersection of a circle and a line.....	577
	Create points at the intersection of two part axes.....	578
	Import points.....	578
	Point properties.....	579
3	Change how model objects are displayed	581
3.1	Show and hide model objects.....	581
	Set the visibility and appearance of model objects.....	582
	Show parts with exact lines.....	583
	Show parts with high accuracy.....	583
	Change the rendering of parts and components.....	584
	Hide model objects.....	588
	Show only selected model objects.....	588
	Temporarily display assembly and component objects.....	589
	Display detailing of a part.....	590
	Show parts in a selected view angle.....	591
3.2	Create object groups.....	591
	Create an object group.....	592
	Copy an object group to another model.....	592
	Delete an object group.....	593
3.3	Change the color and transparency of model objects.....	593
	Change the color of a model object.....	594
	Change the color of an object group.....	595
	Define your own colors for object groups.....	595
	Define color and transparency settings.....	597
	Copy color and transparency settings to another model.....	598
	Delete color and transparency settings.....	598
4	Check the model.....	599
4.1	Inquire object properties.....	599
	Object property report templates.....	601
	Custom inquiry.....	602
	Use the Custom Inquiry tool.....	602
	Define what information is shown by Custom Inquiry tool.....	602
	Modify the default attributes in InquiryTool.config file.....	604
4.2	Measure objects.....	605
	Measure distances.....	605
	Measure angles.....	606
	Measure arcs.....	606
	Measure bolt spacing.....	607
4.3	Compare parts or assemblies.....	607
4.4	Create a clip plane.....	608
4.5	Fly through the model.....	609
4.6	Detect clashes.....	610
	Find clashes in a model.....	611

	Manage clash check results.....	612
	Symbols used in clash checking.....	612
	About clash types.....	613
	Manage the list of clashes.....	616
	Search for clashes.....	617
	Change the status of clashes.....	617
	Change the priority of clashes.....	617
	Group and ungroup clashes.....	618
	View the details of a clash.....	618
	Add comments to a clash.....	619
	Modify a clash comment.....	619
	Remove a clash comment.....	619
	View the history of a clash.....	620
	Print a list of clashes.....	620
	Preview a list of clashes before printing.....	620
	Set the paper size, margins and page orientation.....	621
	Open and save clash check sessions.....	622
	Define a clash check clearance area for bolts.....	623
4.7	View solid errors.....	623
4.8	Diagnose and repair the model.....	624
4.9	Find distant objects.....	626
5	Number the model.....	627
5.1	What is numbering and how to plan it.....	627
	Numbering series.....	628
	Plan your numbering series.....	629
	Assign a numbering series to a part.....	630
	Assign a numbering series to an assembly.....	630
	Overlapping numbering series.....	631
	Identical parts.....	632
	Identical reinforcement.....	632
	Define what affects numbering.....	633
	User-defined attributes in numbering.....	634
	Family numbers.....	634
	Assign family numbers.....	635
	Change the family number of an object.....	636
5.2	Adjust the numbering settings.....	636
5.3	Number parts.....	637
	Number a series of parts.....	638
	Number assemblies and cast units.....	638
	Number reinforcement.....	639
	Number welds.....	640
	Save preliminary numbers.....	640
5.4	Change existing numbers.....	640
5.5	Clear existing numbers.....	641
5.6	Check the numbering.....	642
5.7	View the numbering history.....	644
5.8	Repair numbering errors.....	645
5.9	Renumber the model.....	646
5.10	Control numbers.....	646

	Assign control numbers to parts.....	647
	Control number order.....	648
	Display control numbers in the model.....	649
	Remove control numbers.....	650
	Lock or unlock control numbers.....	651
	Example: Use control numbers to indicate the erection order	651
5.11	Number parts by design group.....	653
5.12	Numbering examples.....	656
	Example: Number identical beams.....	656
	Example: Use family numbers.....	657
	Example: Number selected part types.....	658
	Example: Number parts in selected phases.....	659
5.13	Tips for numbering.....	661
	Numbering settings during a project.....	661
	Create a standard-part model.....	662
6	Applications.....	664
6.1	Working with applications.....	666
6.2	Import a .tsep extension to the Applications & components catalog....	670
6.3	Publish a group in the Applications & components catalog.....	671
7	Components.....	674
7.1	Component properties.....	675
7.2	Add a component to a model.....	677
7.3	View a component in a model.....	679
7.4	Component tips.....	680
7.5	How to use the Applications & components catalog.....	681
	Groups in the catalog.....	681
	Search for a component in the catalog.....	682
	Change the view in the catalog.....	683
	Show selected components in the catalog.....	683
	View and modify component information in the catalog.....	684
	Add a thumbnail image for a component in the catalog.....	684
	Publish a component in the catalog.....	685
	Create and modify groups in the catalog.....	685
	Change the order of groups in the catalog.....	686
	Hide groups and components in the catalog.....	687
	Show the catalog message log.....	687
	Catalog definitions.....	688
7.6	Converting a conceptual or a detailed component.....	688
7.7	Automate connection creation	689
	AutoConnection.....	689
	Define AutoConnection settings and rules.....	690
	Create a connection using AutoConnection.....	693
	AutoDefaults.....	694
	Define AutoDefaults settings and rules.....	695
	Modify a connection using AutoDefaults.....	699
	AutoConnection and AutoDefaults rules.....	699
	Combining and iterating properties for AutoDefaults.....	702
	AutoDefaults example: Using iteration with connection check.....	704

	Using reaction forces and UDLs in AutoDefaults and AutoConnection.....	706
7.8	Advanced component settings	707
	Define connection properties in the joints.def file.....	708
	How to use the joints.def file.....	708
	Example: How Tekla Structures uses the joints.def file.....	710
	General defaults in the joints.def file.....	711
	Bolt diameter and number of bolts in the joints.def file.....	712
	Bolt and part properties in the joints.def file.....	714
	Excel spreadsheets in connection design.....	720
	Files used in Excel spreadsheet connection design.....	721
	Example of an Excel spreadsheet in connection design.....	722
	Example of visualizing the Excel connection design process.....	726
	Showing connection status in Excel connection design.....	730
	General tab.....	730
	Design and Design type tabs.....	731
	Analysis tab.....	734
8	Custom components.....	736
8.1	Custom parts.....	738
8.2	Custom connections.....	740
8.3	Custom details.....	742
8.4	Custom seams.....	744
8.5	Create a custom component.....	747
	Explode a component.....	750
	Create a nested custom component.....	750
	Create a thumbnail image of a custom component.....	753
	Example: Create a custom end plate component.....	754
	Example: Create a nested connection with stiffeners.....	757
8.6	Modify a custom component.....	761
	Protect a custom component with a password.....	762
8.7	Add variables to a custom component	763
	Bind component objects to a plane.....	764
	Bind objects automatically.....	764
	Bind objects manually.....	766
	Test a binding.....	769
	Delete a binding.....	769
	Example: Bind an end plate to a plane.....	770
	Bind component objects using magnetic construction planes or lines.....	772
	Bind handles using a magnetic construction plane.....	772
	Bind handles using a magnetic construction line.....	774
	Add a distance between component objects.....	774
	Set object properties using parametric variables.....	777
	Copy properties and property references from another object.....	780
	Create a variable formula.....	782
	Functions in variable formulas.....	784
	How to avoid cyclic dependencies in formulas.....	796
	Examples of parametric variables and variable formulas.....	797
	Example: Set the end plate material.....	798
	Example: Create new component objects.....	799
	Example: Replace sub-components.....	801
	Example: Modify a sub-component by using a component attribute file.....	802
	Example: Define the stiffener position using construction planes.....	803

	Example: Determine the bolt size and bolt standard.....	806
	Example: Calculate the bolt group distance.....	807
	Example: Calculate the number of bolt rows.....	809
	Example: Link variables to user-defined attributes.....	811
	Example: Calculate the number of handrail posts using a template attribute.....	813
	Example: Link an Excel spreadsheet to a custom component.....	815
	Examples: Rebar set modifiers in custom components.....	816
8.8	Save a custom component.....	823
8.9	Modify the dialog box of a custom component.....	824
	Custom component input files.....	827
	Modify custom component input files manually.....	827
	Add new tabs.....	827
	Add text boxes.....	828
	Add images.....	829
	Change the order of boxes.....	829
	Change the location of boxes.....	830
	Lock or unlock the custom component input file.....	831
	Example: Create a toggle group.....	831
	Example: Modify the dialog box of a stiffener detail.....	835
	Example: Create a custom stiffener detail with variables.....	836
	Example: Add a list with images.....	847
	Example: Arrange text boxes and labels.....	852
	Example: Dim unavailable options.....	854
	Dialog editor settings.....	857
8.10	Add a custom component to a model.....	859
8.11	Add or move a custom part in the model.....	860
8.12	Import and export custom components.....	862
	Export custom components.....	862
	Import custom components.....	863
8.13	Custom component settings.....	864
	Custom component properties.....	864
	Type/Notes tab properties.....	864
	Position tab properties.....	865
	Advanced tab properties.....	866
	Default properties of a custom component.....	867
	Default properties of custom parts.....	868
	Default properties of custom connections, details, and seams.....	870
	Plane types.....	872
	Examples of component planes.....	873
	Variable properties	875
8.14	Hints and tips for using custom components.....	881
	Tips for creating custom components.....	881
	Tips for sharing custom components.....	882
	Tips for updating custom components to a new version.....	883
9	Predefined parametric profiles available in Tekla Structures.....	884
9.1	I profiles.....	884
9.2	I beams (steel).....	885
9.3	L profiles.....	885
9.4	Z profiles.....	886

9.5	U profiles	887
9.6	C profiles	887
9.7	T profiles	888
9.8	Welded box profiles	888
9.9	Welded beam profiles	888
9.10	Box profiles	891
9.11	WQ profiles	892
9.12	Rectangular sections	892
9.13	Circular sections	893
9.14	Rectangular hollow sections	893
9.15	Circular hollow sections	894
9.16	Cold rolled profiles	894
9.17	Folded plates	897
9.18	Hat profiles	904
9.19	I beams (concrete)	905
9.20	Ledger beams (concrete)	905
9.21	T profiles (concrete)	906
9.22	Irregular beams (concrete)	908
9.23	Panels	911
9.24	Variable cross sections	914
9.25	Others	916
10	Modeling settings	918
10.1	View and representation settings	918
	Display settings.....	918
	Color settings for object groups.....	921
	Transparency settings for object groups.....	922
10.2	Part position settings	923
10.3	Numbering settings	923
	General numbering settings.....	923
	Weld numbering settings.....	925
	Control number settings.....	926
10.4	Reinforcement settings	927
	Reinforcing bar and bar group properties.....	927
	Reinforcement mesh properties.....	930
	Custom reinforcement mesh properties.....	931
	Rebar set properties.....	933
	Secondary guideline properties.....	936
	Leg face properties.....	938
	Property modifier properties.....	938
	End detail modifier properties.....	941
	Splitter properties.....	943
	Reinforcement strand properties.....	946
11	Disclaimer	949

1

Get to know Tekla Structures basic working methods

Before starting to create models and drawings, get to know some basic working methods that you will need to work efficiently both in the Tekla Structures models and in the drawings.

We recommend you to familiarize yourself how to

- [work with grids \(page 18\)](#) and views
- define the work area and [coordinates \(page 48\)](#) that affect your work
- zoom and rotate the model
- [create \(page 96\)](#), select and move objects
- position objects by using [snapping \(page 77\)](#)
- [filter objects \(page 155\)](#) both in the modeling mode and in the drawing mode.

See also

[Create parts and modify part properties \(page 235\)](#)

1.1 Set up the workspace

Before starting to model, check that your Tekla Structures workspace is set up correctly.

1. [Define the units and decimals you will use. \(page 18\)](#)
2. [Modify the grid to suit your needs. \(page 18\)](#) Create a modular grid if needed.
3. [Create some views \(page 29\)](#) to examine the model from different angles and elevations.
4. [Resize the work area to suit your project. \(page 46\)](#)

5. [Get familiar with the coordinate system \(page 48\)](#). If you are modeling sloped structures, [shift the work plane accordingly. \(page 51\)](#)

Change units and decimals

You can define which units and how many decimals Tekla Structures uses. The settings are model-specific. Note that these settings do not have any effect on drawings or reports, or on the **Inquire** and **Measure** tools.

1. On the **File** menu, click **Settings** --> **Options**, and go to the **Units and decimals** settings.
2. Modify the units and decimals to suit your needs.

The number located to the right of each option indicates the number of decimals. The number of decimals affects the input and storage accuracy. Always use a sufficient number of decimals.

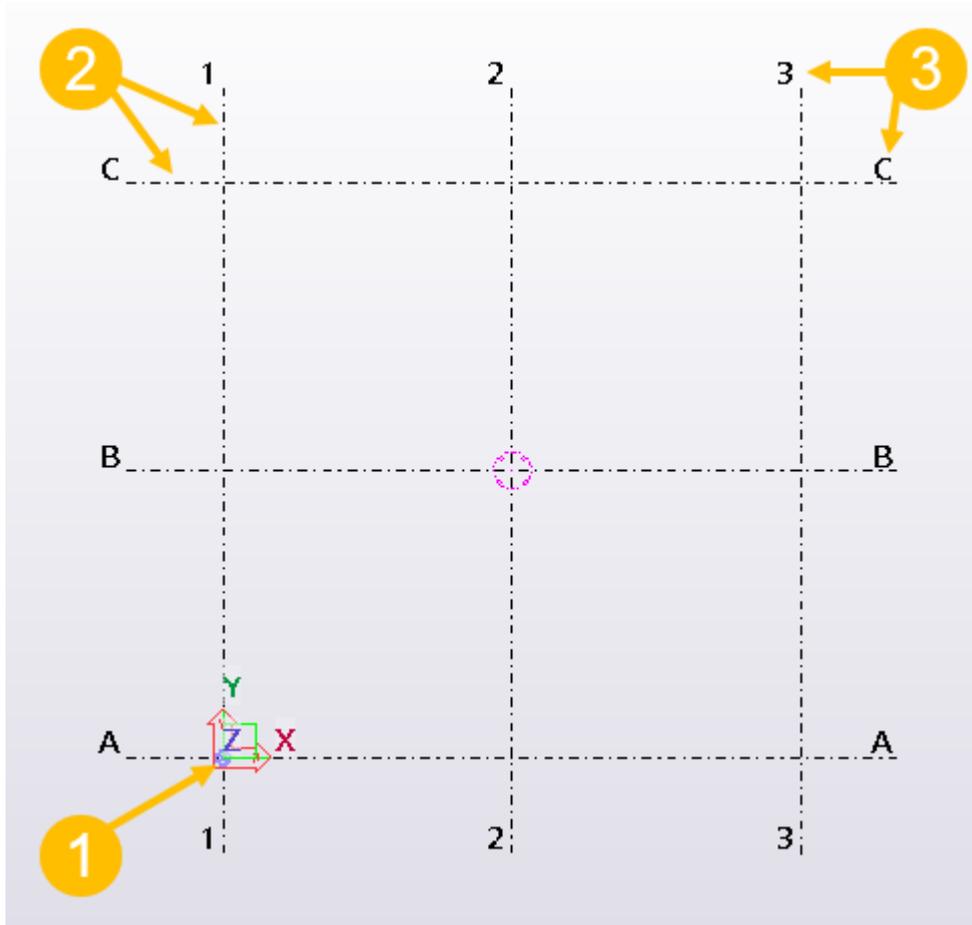
- The settings on the **Modeling** tab affect the data that is used when you are modeling, for example copying, moving, creating grids, creating points, and so on.
 - The settings on the **Catalogs** tab affect the data stored in the profile and material catalogs.
 - The settings on the **Analysis results** tab affect the output data.
3. Click **OK** to save the changes.

Work with grids

A *grid* represents a three-dimensional complex of horizontal and vertical planes. The grid is shown on the view plane using dash-and-dot lines. You can create both rectangular and radial grids. Use grids as an aid in locating objects in a model. You can make rectangular grids and *grid lines* act magnetically so that the objects on the grid lines of a rectangular grid follow if you move the grid line.

- [Create, modify, or delete a grid \(page 21\)](#)
- [Add a single grid line \(page 26\)](#)
- [Modify a single grid line \(page 26\)](#)

Grid terminology



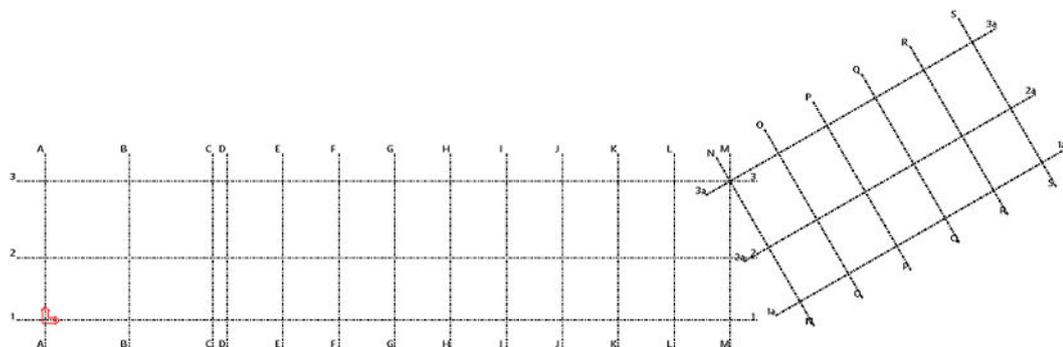
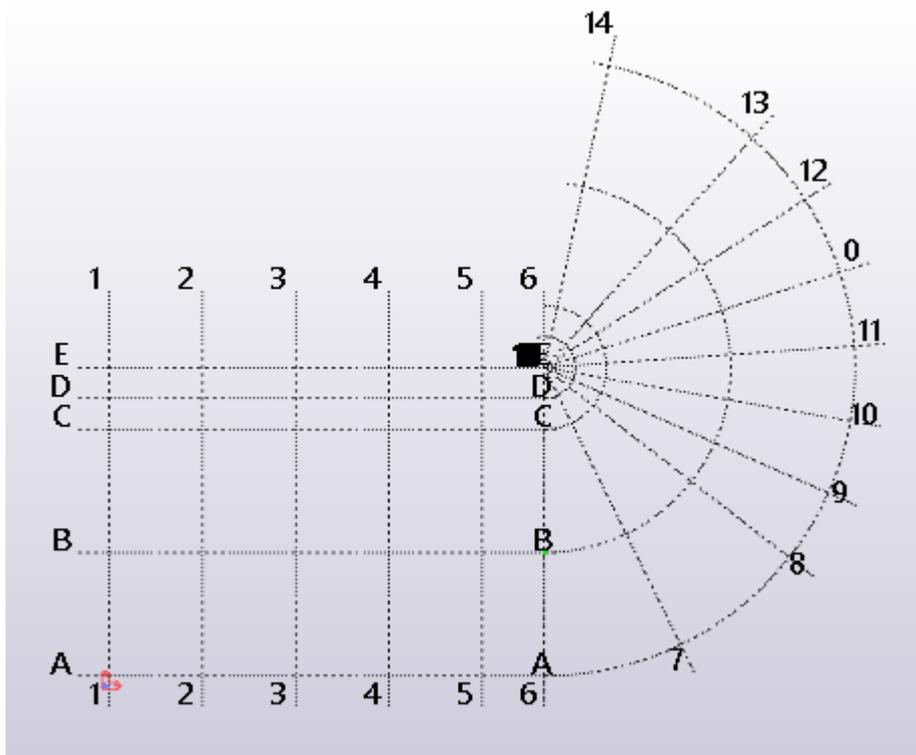
(1) Grid origin is the point where the zero points of each coordinate axis intersect

(2) Grid line extensions define how far the grid lines extend in each direction

(3) Grid labels are the names of the grid lines shown in views

Several grids in one model

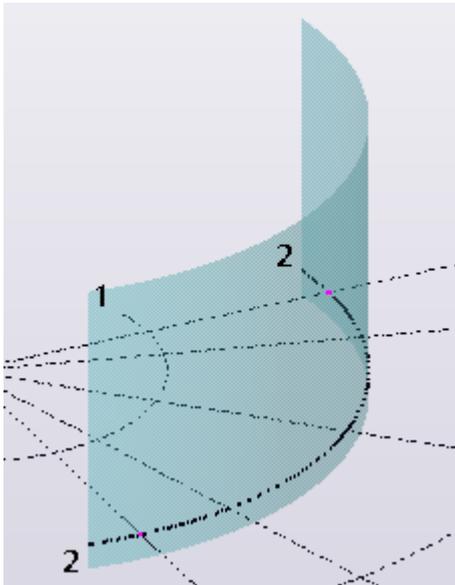
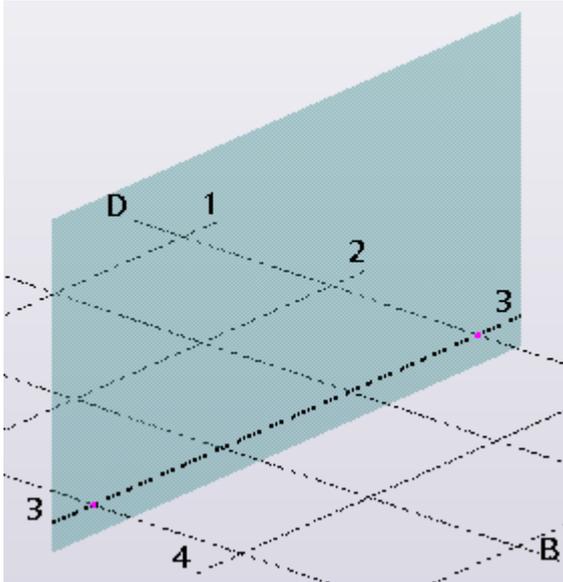
You can have more than one grid in a model. You can create a large-scale grid for the entire structure, and smaller grids for some detailed sections. Create as many grids as needed, so that you can easily place objects in your model.



Single grid lines

You can create single grid lines and attach them to an existing grid.

Single grid lines have [handles \(page 303\)](#). If the  **Select grid line selection switch (page 122)** is active and you select a grid line, the handles appear in magenta. If you move the handles to make a skewed grid, you can do this only on the local [XY plane \(page 29\)](#) of the grid.

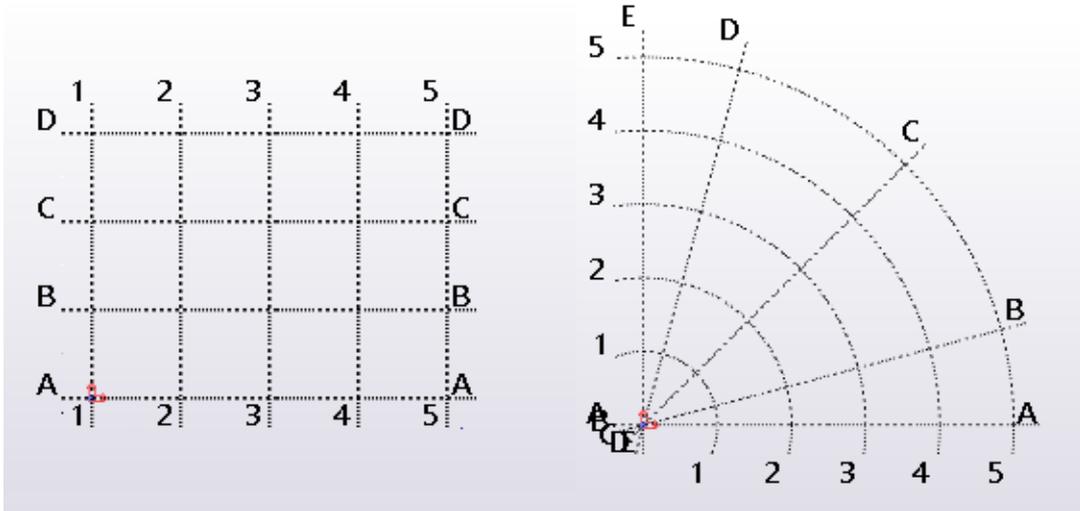


Create, modify, or delete a grid

When you create a new model, Tekla Structures automatically creates a rectangular grid and a view according to the saved standard properties. If needed, you can later modify the grid properties, or create new rectangular and radial grids. You can modify an existing grid, or delete it. Use the grid properties in the property pane to view and modify the grids.

Create a rectangular grid or a radial grid

You can create both rectangular and radial grids and modify their properties in the property pane.



1. Select whether to create a rectangular or a radial grid.
 - To create a rectangular grid: on the **Edit** tab, click **Grid** --> **Create rectangular grid** .
 - To create a radial grid: on the **Edit** tab, click **Grid** --> **Create radial grid** .

Tekla Structures shows a preview of the grid. You can modify the grid properties in the property pane before inserting the grid. The grid preview changes according to the changes in the property pane.

2. Pick a point in the model to indicate the origin of the grid, or click the middle mouse button to accept the grid property values in the property pane.
 - If you pick an origin point to the grid, the grid is created using the properties in the property pane, and the origin is the location you picked.
 - If you click the middle mouse button, the grid is created using the properties in the property pane, and to the origin defined in the property pane.

The coordinates of the origin are shown in the property pane in the **Origin** section as the **X**, **Y**, and **Z** values.

NOTE When working with very large grids, always having the grid labels visible might slow down Tekla Structures. To hide the grid labels when you zoom in, use the advanced option `XS_ADJUST_GRID_LABELS`.

Create a radial grid (alternative method)

This is an alternative method to create a radial grid. You can create a radial grid using a component called **Radial Grid**. Note that the curved grid lines created by the **Radial Grid** component are not truly curved but straight.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Start typing `radial grid` in the search box.
3. Double-click **Radial Grid** to open the properties dialog box.
4. Modify the grid properties.

In the coordinate properties

- **X** defines the location of the curved grid lines and the distance between the grid lines.
The first value is the radius of the innermost curved grid line.
- **Y (degrees)** defines the location of the straight grid lines and the distance between the grid lines in degrees.
The first value defines how the grid is rotated. The grid is rotated counter-clockwise from the x axis in the current work plane.

5. Click **OK**.
6. Pick a point to indicate the origin of the grid.
The grid is created automatically.

Modify a grid

Double-click an existing grid to modify it.

1. Ensure that the  **Select grid selection switch** (page 122) is active.
2. Double-click a grid line.
Depending on the type of the grid, the **Rectangular grid** or the **Radial grid** properties open in the property pane.
3. Change the grid properties.
4. Click **Modify** to apply the changes.

Delete a grid

When you delete an entire grid, ensure that you do not have any other objects selected. Otherwise Tekla Structures only deletes the objects, not the grid.

1. Ensure that **only** the  **Select grid selection switch** (page 122) is active.
2. Select the grid.

3. Press **Delete**.
4. Confirm that you want to delete the grid.

Grid properties

Use the **Rectangular grid** or the **Radial grid** properties in the property pane to view and modify the properties of a grid. To open the properties, double-click the grid. The file name extension of a rectangular grid property file is *.grd. The file name extension of a radial grid property file is *.rgrd.

If you have customized the property pane layout, the list of properties may be different.

Setting	Description
Coordinates	<p>In the Rectangular grid properties:</p> <p>Coordinates of the grid in the x, y, and z directions.</p> <p>X: grid lines vertical to the work plane</p> <p>Y: grid lines horizontal to the work plane</p> <p>Z: elevations in the structure</p> <p>You can enter a maximum of 1024 characters. Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.</p> <p>The x and y coordinates are relative, which means that the entries for X and Y are always relative to the previous entry. The Z coordinates are absolute, which means that entries for z are absolute distances from the work plane origin.</p> <p>You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:</p> <pre>0 4000 4000</pre> <pre>0 2*4000</pre> <p>In the Radial grid properties:</p> <p>Radial and angular values of the grid lines. Elevations is the value of the grid in z direction.</p> <p>Radial: curved grid lines. If you enter only one value for radius, it must be >0.</p> <p>Angular: straight grid lines. You can enter negative angle values, if needed.</p> <p>Elevations: elevations in the structure</p>

Setting	Description
	<p>Use a zero at the start to represent a grid on the 0,0 coordinate and spaces as separators for coordinates.</p> <p>The Radial and Angular coordinates are relative, which means that the entries are always relative to the previous entry. The Elevations are absolute, which means that entries are absolute distances from the work plane origin.</p> <p>You can either define the coordinates individually, or you can define several grid lines with equal spacing. Both of the following coordinate entries create three grid lines with the spacing of 4000:</p> <pre>0 4000 4000</pre> <pre>0 2*4000</pre>
Labels	<p>In the Rectangular grid properties:</p> <p>Names of the grid lines shown in views.</p> <p>Names in the X box are associated with the grid lines parallel to the y axis and vice versa. The Z box is for the names of levels parallel to the work plane. If you wish, you can leave the label boxes empty.</p> <p>In the Radial grid properties:</p> <p>Names of the grid lines shown in views.</p> <p>Names in the Radial box are associated with the curved grid lines.</p> <p>Names in the Angular box are associated with the straight grid lines.</p> <p>The Elevations box is for the names of levels parallel to the work plane.</p> <p>If you wish, you can leave the label boxes empty.</p>
Line extensions	<p>In the Rectangular grid properties:</p> <p>Define how far the grid lines extend in the directions Left/Below and Right/Above.</p> <p>In the Radial grid properties:</p> <p>Define how far the grid lines extend in the directions Start and End.</p>
Origin	<p>Coordinates of the grid origin in the x, y, and z directions. These values offset the grid from the work plane origin, not from the global model origin.</p>

Setting	Description
Magnetic grid plane	In the Rectangular grid properties: Select whether objects are bind to grid lines (page 26) . If you bind the objects to grid lines, the objects follow if you move the grid line.
Grid color	Use the color picker to pick the grid color.
Label font size and color	Define the label font size. Use the color picker to pick the label color.
Custom properties	Click the More button to open the user-defined attributes (UDAs) of the grid.

Add a single grid line

You can add new grid lines either between existing grid lines or between two freely chosen points that you define in the model.

Add a grid line between existing grid lines

You can add new grid lines between existing grid lines.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid selection switch (page 122)** is active.
3. Select an existing grid to attach the grid line to.
4. Click the  symbol between two existing grid lines or outside the grid.
Tekla Structures creates the grid line and gives it a label using the labels of the adjacent grid lines. For example, a new grid line between the grid lines 1 and 2 receives the label 12*.

Add a grid line between two points

You can add new grid lines between two picked points.

1. On the **Edit** tab, click **Grid --> Add grid line**.
2. Select an existing grid to attach the grid line to.
3. Pick the start point of the grid line.
4. Pick the end point of the grid line.

Tekla Structures creates the grid line.

Modify a single grid line

You can modify single grid line properties. You can also move grid lines or change the grid line labels.

Modify grid line properties

You can edit the properties of a single grid line in the property pane.

1. Ensure that the  **Select grid line selection switch** (page 122) is active.
2. If the property pane is not open, double-click the grid line to open the **Grid line** properties.
3. [Change](#) (page 96) the properties as needed.
4. Click **Modify** to apply the changes.

Move a grid line

Use direct modification to move single grid lines.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid selection switch** (page 122) is active.
3. Select the grid.
4. Select the grid line you want to move.
5. Drag the grid line to a new location.

You can also use the keyboard to enter a numeric location.

To start with the negative sign (-), use the numeric keypad. To enter an absolute coordinate, first enter \$, then the value. Press **Enter** to confirm.

Change a grid line label

Use the contextual toolbar to change the label of a single grid line.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid line selection switch** (page 122) is active.
3. Select a grid line.
4. On the contextual toolbar, enter a new label.



Stretch, shrink, or incline a grid line

Use direct modification to stretch, shrink, or incline single grid lines in rectangular grids.

Note that this works only for the single grid lines that you have added in rectangular grids by using the **Add grid line** command.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid selection switch (page 122)** is active.
3. Select the rectangular grid.
4. Select the grid line.
5. Drag a grid line handle  to a new location.

Turn grid line stretching off

If you move the outermost grid lines in rectangular grids using the line handles, Tekla Structures stretches or shrinks the perpendicular, crossing grid lines accordingly by default. You can switch this off temporarily.

1. Ensure that the  **Direct modification** switch is active.
2. Ensure that the  **Select grid selection switch (page 122)** is active.
3. Select the grid line.
4. On the contextual toolbar, click the  **Turn grid line stretching off** button.

Grid line properties

Use the **Grid line** properties in the property pane to view and modify the properties of a single grid line. To open the properties, double-click a single grid line. The file name extension of a grid line property file is `.gridp`.

The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

Setting	Description
Label	Name of the grid line.
Depth in view plane	Height of the grid plane perpendicular to the view plane.
Line extension Left/Below	Define how far the grid lines extend in the directions Left/Below and Right/Above .
Line extension Right/Above	

Setting	Description
Magnetic	Select whether objects are bind to straight the grid lines. If you bind the objects to grid lines, the objects follow if you move the grid line.
Visible in drawing	Select whether grid lines are made visible in drawings.
Grid line automatic dimensioning	Select whether single grid lines are used in grid dimensioning.
More	Click the More button to open the user-defined attributes (UDAs) of the grid line.

Delete a single grid line

You can delete grid lines in two different ways. The easiest way is by using direct modification.

Delete a grid line using direct modification

Use direct modification to quickly delete single grid lines.

1. Ensure that the  **Direct modification** switch is active.
2. Select the grid line you want to delete.
3. Press **Delete**.

Delete a grid line (alternative method)

This is the alternative way of deleting single grid lines.

1. Ensure that the  **Select grid line** [selection switch \(page 122\)](#) is active.
2. Select the grid line you want to delete.
3. Ensure that you do not have any other objects selected.
If you also have other objects selected, Tekla Structures only deletes the objects, not the grid line.
4. Right-click and select **Delete** from the pop-up menu.
5. Confirm that you want to delete the grid line.

Work with views

A *view* is a representation of a model from a specific location. Each view is represented in its own window within Tekla Structures. Selecting a part in a view highlights the part in all open views.

- [Create model views \(page 31\)](#)
- [Open a view \(page 41\)](#)
- [Switch between views \(page 43\)](#)
- [Change the color settings \(page 64\)](#)

View plane

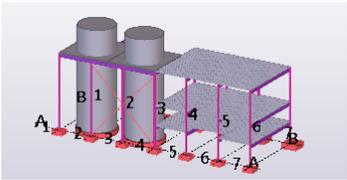
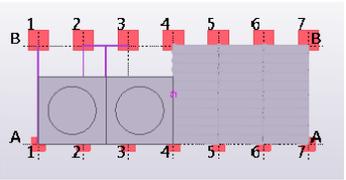
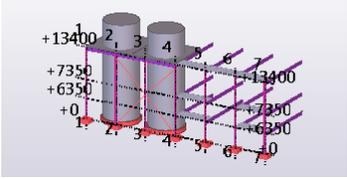
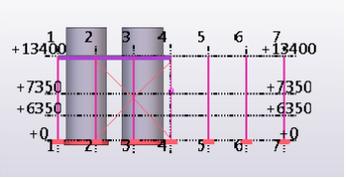
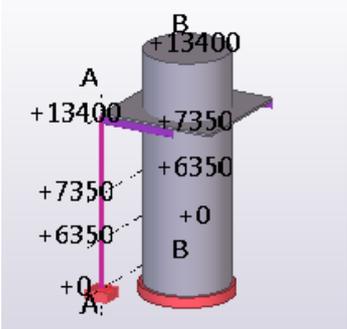
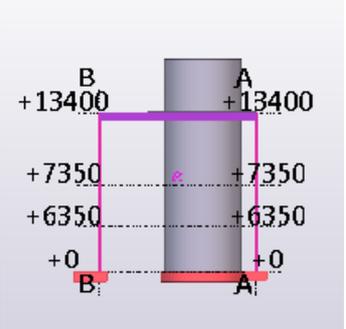
Each view has a view plane on which the [grids \(page 18\)](#) are visible and points are represented as blue crosses. Points that are located outside the view plane are red. You can [move the view plane \(page 31\)](#) like any other object.

Basic views

Basic views are those parallel to the global basic planes (xy, xz, and zy). In basic views, two axes always define the view plane and the axes appear in the plane name. The third axis is perpendicular to the view plane. It does not appear in the plane name. In the basic plane view, the model is shown from the direction of the third axis.

When you [create basic views \(page 32\)](#), you must define the view plane's distance (the view plane coordinate) from the global origin in the direction of the third axis.

Examples of basic views:

Plane	3D view	Plane view
XY		
XZ		
ZY		

Other views

For other view types, you either define the view plane and coordinate by picking points, or the points are defined automatically, depending on the creation method.

Should I model in a 3D or plane view?

Views can be 3D or plane views. 3D, plane, and also elevation views provide different type of information, which is useful for different tasks.

One common technique is to open several views:

- A 3D view to see a real-life version of the model
- A plane view, where you can add and connect parts
- An elevation view to check the level

If you are working with several screens, maximize your work area by putting the different views on different screens.

You can easily switch between the 3D and plane view by using the keyboard shortcut **Ctrl+P**.

Move the view plane

You can move the view plane like any other object. When you move it, Tekla Structures only uses the vector that is perpendicular to the view plane.

1. Click the view.
2. Right-click and select **Move --> Linear**.
3. Pick the start point of the translation vector, or enter its coordinates.
4. Pick the end point of the translation vector, or enter its coordinates.
5. Click **Move** to move the view plane.

Create model views

You can create model views in several ways in Tekla Structures.

For example, you can create views

- [of the entire model \(3D view\) \(page 32\)](#)
- [of selected parts \(page 37\)](#)
- [of selected components \(page 38\)](#)
- [along grid lines \(axis views\) \(page 33\)](#)
- [to the surface of an object \(page 38\)](#)

Each view has properties which define its appearance. You can change the appearance of a view after you have created it. To [view and modify the](#)

[properties of each view \(page 44\)](#), double-click anywhere on the model background to open the **View Properties** dialog box.

If you create views along grid lines, you can [view and modify the properties \(page 45\)](#) using the **Creation of Views Along Grid Lines** dialog box.

Create a basic view of the model

You can create a basic view along two coordinate axes. Use this view for the overall viewing of the model.



1. On the **View** tab, click **New view --> Basic view**.
2. Select a view plane from the **Plane** list.
The view plane is defined by two axes.
3. In the **Coordinate** box, enter the view level.
This value defines the distance of the view plane from the global origin, perpendicular to the view plane.
4. Click **Create**.

Create a view using two points

You can create a view using two points you pick: the origin and a point in the direction of the x axis.



1. On the **View** tab, click **New view --> Using two points**.
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.
The y axis is perpendicular to the view plane on which you picked the first point.

Create a view using three points

You can create a view using three points you pick: the origin, a point in the direction of the x axis, and the third point in the direction of the y axis.



1. On the **View** tab, click **New view --> Using three points**.
2. Pick a point to indicate the origin of the view plane.
3. Pick a second point to indicate the direction of the x axis.
4. Pick a third point to indicate the direction of the y axis.

Create a view of the work plane

You can create a view of the work plane using the current view properties.



- On the **View** tab, click **New view --> On work plane** .

Create grid views

You can create views along the grid lines you select.

Before you start, create a view that contains a grid, and check the grid properties. Note that with radial grid you can create the grid views only on the straight grid lines, not on the arched grid lines.

If the grid properties are incorrect in some way, Tekla Structures may cut the views at the wrong elevations or they may be named incorrectly. If you change the grid labels or the elevation or grids later on, the views will not be automatically renamed.

1. Select the grid.



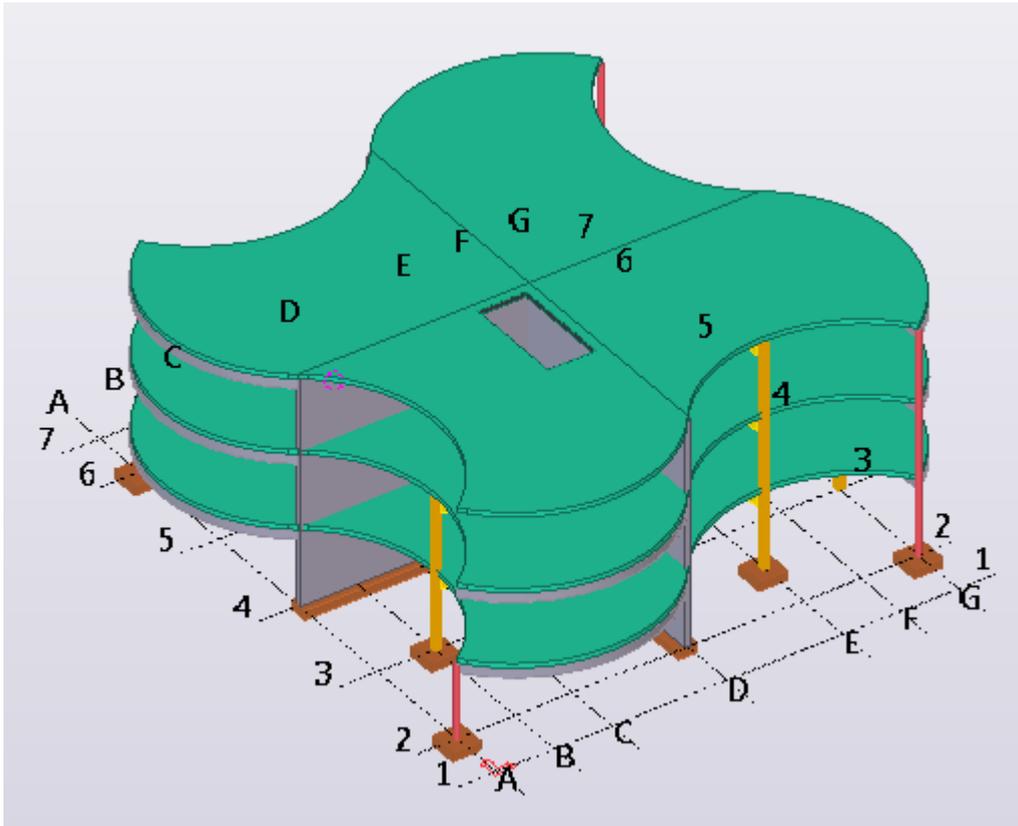
2. On the **View** tab, click **New view --> Along grid lines** .
3. Modify the grid view properties if needed.
 - a. In the **Number of views** list, select how many views you want to create.
 - b. In the **View name prefix** box, enter a prefix.
 - c. In the **View properties** list, define which view properties (applied or saved) you want to use.
4. Click **Create**.

The **Views** dialog box opens.
5. Click the arrow buttons to move views from the **Named views** list to the **Visible views** list.

The views will not be visible until you move them to the **Visible views** list.

Example

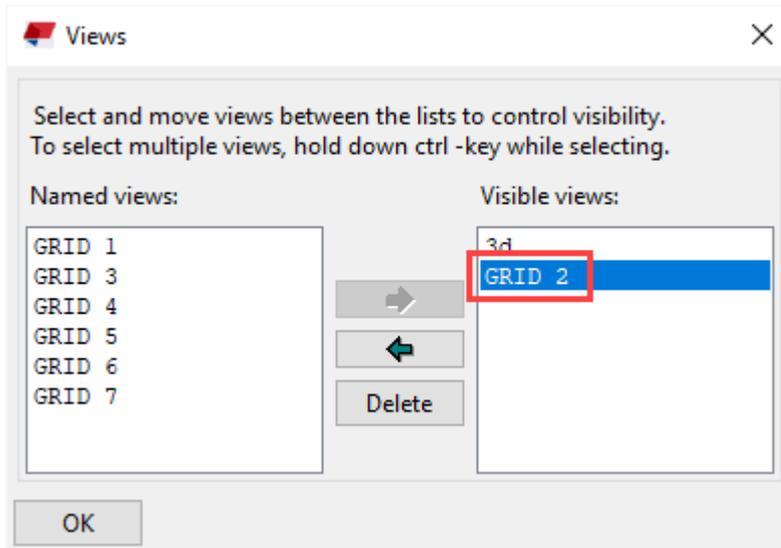
This example shows how to create vertical views of the grid lines 1–7 on the following model:



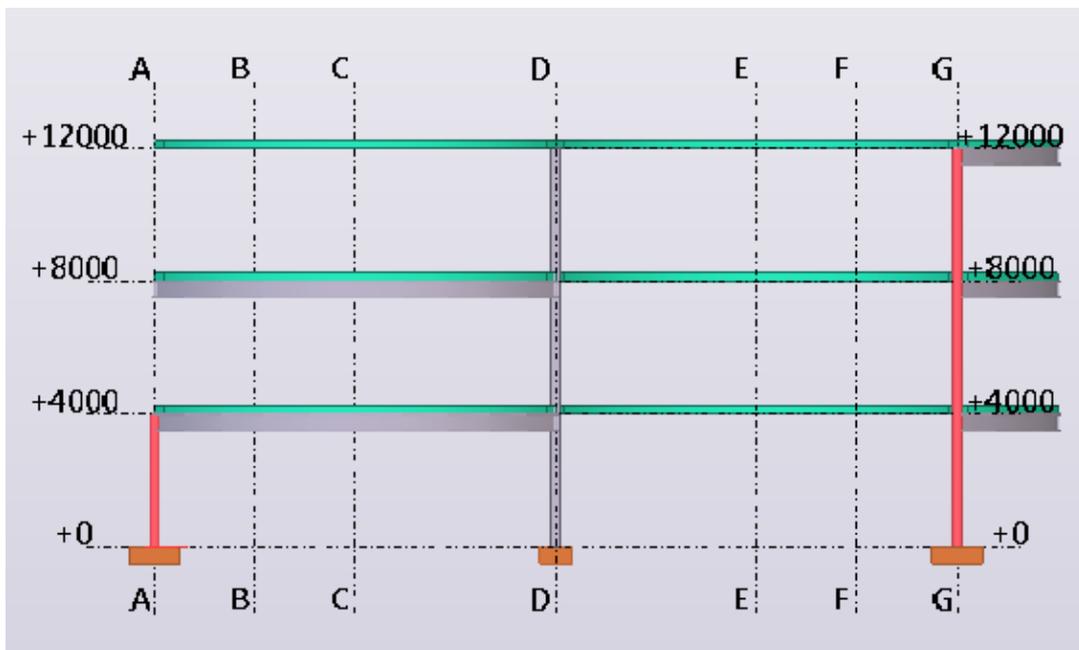
In the **Creation of Views Along Grid Lines** dialog box, select **All** for the view plane XZ and **None** for the view planes XY and ZY. Use the default settings for the view name prefix and the view properties.

Rectangular grid				
View plane	Number of views	View name prefix	View properties	
XY	None	PLAN	Grid-Plan	Show...
ZY	None	GRID	Grid-Elevation	Show...
XZ	All	GRID	Grid-Elevation	Show...

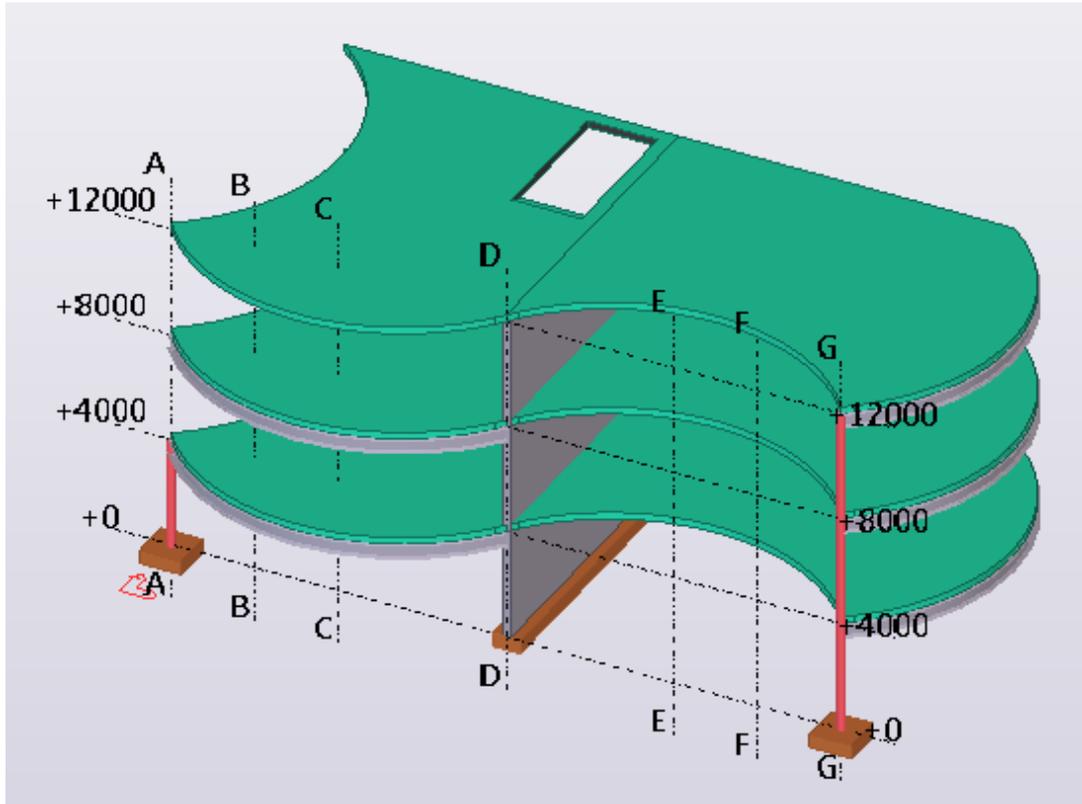
After creating the grid views, move the view named **Grid 2** to the **Visible views** list:



The grid view is displayed as a plane view in a new window:



Rotate the view to see it in 3D:



Create a view on a plane

You can create a view on the work plane or almost any plane of an existing part.



1. On the **View** tab, click **New view --> On plane** .

When you hover the mouse cursor over model objects, Tekla Structures highlights the available planes in light blue.

2. Select the desired plane.

TIP Alternatively, to create a view on the front, top, back, or bottom plane of a part, you can also use the following commands available on the **View** tab:

- **On part front plane**
- **On part top plane**
- **On part back plane**
- **On part bottom plane**

To use these commands, first select a command and then the part.

Create a 3D view of a part

When you need to see a specific part clearly, create a 3D view of the part. The part is placed in the center of the view.



1. On the **View** tab, click **New view --> 3D view of part** .
2. Select the part.

Tekla Structures creates the view using the properties defined in the `part_basic_view` property file. The view plane y axis is the global z axis of the model. The x axis is the projection of the part's local x axis onto the global xy plane.

Tekla Structures fits the work area to include the selected part.

If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default part views

You can create four views of a part: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.



1. On the **View** tab, click **New view --> Default part views** .
2. Select the part.

Tekla Structures creates the four default views all at once, using the properties defined in `part_front_view`, `part_top_view`, `part_end_view`, and `part_persp_view` property files.

Create an undeformed part view

You can create a view that shows a deformed part in undeformed form. This works only for beams and columns.



1. On the **View** tab, click **New view --> Undeformed part view** .
2. Select the part.

For example, select a warped beam. Tekla Structures displays the beam in a separate view in undeformed form.

Create a 3D view of a component

When you need to see a specific component clearly, create a 3D view of the component. The component is placed in the center of the view.



1. On the **View** tab, click **New view --> 3D view of component** .
2. Select the component.

Tekla Structures creates the view, using the properties defined in the `component_basic_view` property file. The view plane y axis is the global z axis of the model. The x axis is the projection of the first secondary part local x axis onto the global xy plane. Work area depth is 1 m in all directions.

The component you select automatically defines the work area.

If you want Tekla Structures to rotate the view once whenever you create a new 3D view, select the Basic view auto rotation setting in the **File** menu.

Create default component views

You can create four views of a component: front, top, end, and 3D view. Tekla Structures creates these views all at once with the same command. The front, top, and end views are plane views by default.



1. On the **View** tab, click **New view --> Default views of component** .
2. Select the component.

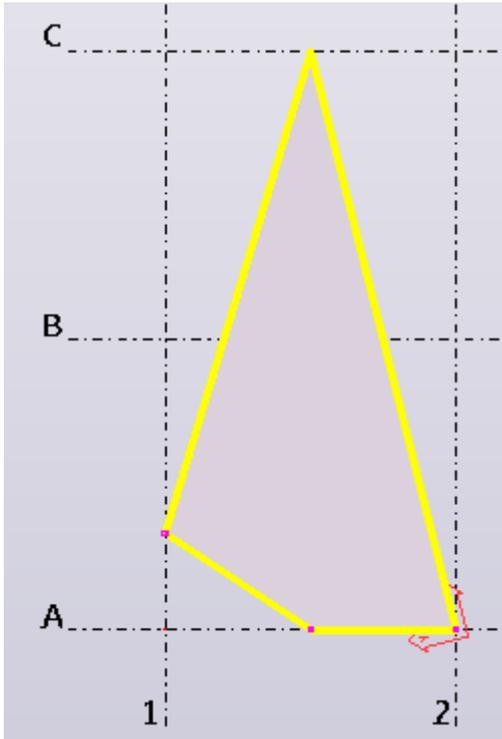
Tekla Structures creates the four default views all at once, using the properties defined in the `component_front_view`, `component_top_view`, `component_end_view`, and `component_persp_view` property files.

Create a surface view

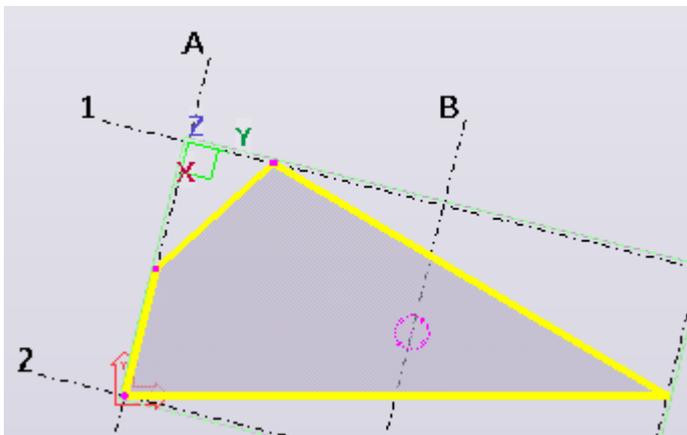
Use the **Create surface view** macro to create an automatically aligned surface view. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Create surface view** to start the macro.
4. Select the surface of the part.



Tekla Structures creates a new temporary view and moves the work plane typically along the longest edge of the part face. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.



5. Press **Esc** to stop the macro.
6. To return the work plane back to the origin:
 - a. Repeat steps 1-2 to open the **Applications** list.
 - b. Double-click the **Work plane global** macro.

The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Create a surface view along selected edge

Use the **Create surface view wEdge** macro to create a surface view and align the work plane along the edge you select. This can be useful when modeling bolt groups, stiffener plates, and hole penetrations on complex geometry.

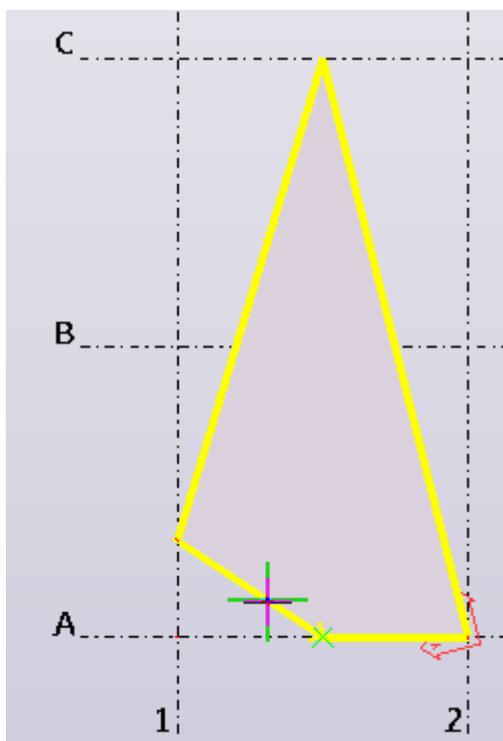
To be able to select the surface of a part, ensure that you are using a model view that shows part faces. On the **View** tab, click **Rendering**, and select either the **Parts grayscale** or the **Parts rendered** option.

1. Ensure that the  **Snap to geometry lines / points** [snap switch \(page 79\)](#) is active.

This allows you to pick along an edge to define the direction.

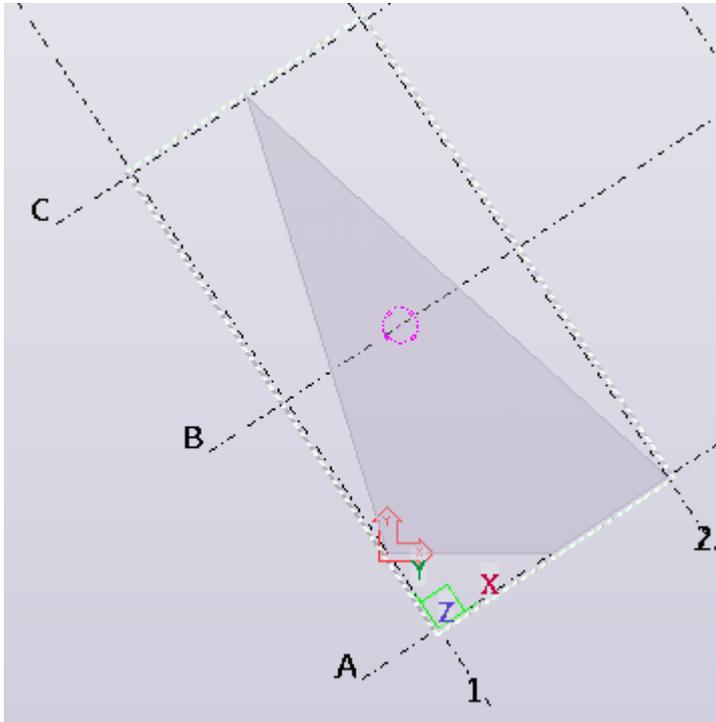
2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Click the arrow next to **Applications** to open the applications list.
4. Double-click **Create surface view wEdge** to start the macro.
5. Select the surface of the part.

When you hover the mouse pointer over the part edges, a yellow arrow symbol is displayed to indicate the possible edges you can align the view to. The head of the arrow represents the positive direction of the x axis. The view will be rotated in this direction to form the flat horizontal edge of the view. The origin of the view and work plane will be at the start of the arrow snap line.



- Pick the desired edge.

Tekla Structures creates a new temporary view, and the selected edge forms the x axis of the view. You can model in the surface view and see your modeling work being done in your original 3D view at the same time.



- Press **Esc** to stop the macro.
- To return the work plane back to the origin:
 - Repeat steps 2–3 to open the **Applications** list.
 - Double-click the **Work plane global** macro.

The work plane returns back to the origin and is aligned with the global x,y, and z planes of the model.

Open, save, modify, or delete a view

When you create views, you can have up to nine views on the screen at the same time. If you need views for later use, you can name and save them. You can modify the properties of an existing view by using the **View properties** dialog box.

Open a view

You can have up to nine views on the screen at the same time. If you are unable to open a view, check how many views you already have open - you may need to close some of them first.



1. On the **View** tab, click **View list** to open the **Views** dialog box. Tekla Structures lists all invisible named views on the left, and all visible views on the right.
2. Select a view from the **Named views** list and click the right arrow to move it to the **Visible views** list.
You can also double-click a view to open it. If the view does not appear, check how many views you already have open.
3. To open multiple views, use the **Shift** and **Ctrl** keys when you select views from the list.

Save a view

If you need to re-open views later on, give each view a unique name. When you exit the model, Tekla Structures only saves the named views. Temporary views disappear when you close them.

Before you start, [create one or more views \(page 31\)](#) in the model.

1. Double-click the view to open the **View Properties** dialog box.
2. Enter a unique name in the **Name** box.

Temporary views have a default name in parentheses. Do not use parentheses when naming a view, or the view will not be saved for later use.

NOTE In multi-user mode, it is very important to give views unique names. If several users have different views with the same name, the view settings of one user may accidentally override the settings of another user.

3. Click **Modify**.

Tekla Structures will automatically save all named views when you close the model.

Modify a view

You can modify a view simply by double-clicking it.

1. Double-click the view to open the **View Properties** dialog box.
2. Modify the [view properties \(page 44\)](#).
3. Click **Modify**.

Delete a view

You can permanently delete named views.



1. On the **View** tab, click **View list** to open the **Views** dialog box. Tekla Structures lists all invisible named [views \(page 29\)](#) on the left, and all visible views on the right.
2. Select the view you want to delete.
3. Click **Delete**.
Tekla Structures deletes the view permanently. If the view was visible during the deletion, it will remain visible until you close it.
4. To delete multiple views, use **Shift** or **Ctrl** when you select views from the list.

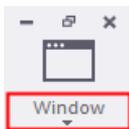
Switch between views

You can easily switch between all open views while modeling. You can also switch between the 3D and plane view, to examine the current view from different perspectives.

Switch between open views

To switch between open views, do one of the following:

- Use the keyboard shortcut **Ctrl+Tab**.
- Click **Window** and select a view from the list.



The views are listed in alphabetical order.

- Right-click a view, then select **Next window** from the pop-up menu.
The next open view becomes active.

Switch between 3D and plane view

Use the **Switch to 3D or plane** command to examine the current view from different perspectives.

- On the **View** tab, click  **Switch to 3D or plane**.
You can also press **Ctrl+P**.

Update and refresh views

Use the **Update window** and **Redraw** commands to refresh a single view or all views at once.

- **Update:** Removes temporary graphics (such as measured distances) but does not redraw the view. Faster than redrawing.
- **Redraw:** Redraws the view completely and shows all previously hidden objects.

To	Do this
Update the current view	Right-click the view and select Update window .
Update all of the views	On the View tab, click Redraw --> Erase temporary graphics . 
Redraw the current view	Right-click the view and select Redraw View .
Redraw all of the views	On the View tab, click  .

See also

[Switch between views \(page 43\)](#)

View properties

Use the **View Properties** dialog box to view and modify the model view properties.

Option	Description
Name	The name of the view.
Angle	Whether the view angle is Plane or 3D .
Projection	The projection type of views. Orthogonal: All objects are of equal size (no perspective). When you zoom, text and point size remains the same. In addition, the zoom remains on object faces. Perspective: Distant objects appear smaller than close ones, as do text and points. You can zoom, rotate, or fly through the model.

Option	Description
Rotation	How the view is rotated around the z and x axes. Rotation is view-specific. The units depend on the settings in File menu --> Settings --> Options --> Units and decimals .
Color and transparency in all views	The color and transparency setting that is used in all views (according to the status of the objects in the model).
Representation...	Opens the Object Representation dialog box for defining color and transparency settings.
View depth	The thickness of the displayed slice of model. You can define the depth separately upwards and downwards from the view plane. Only objects positioned within the view depth are visible in the model. The units depend on the settings in File menu --> Settings --> Options --> Units and decimals .
Display...	Opens the Display dialog box for defining which objects are displayed (page 918) in the view and how.
Visible object group	Which object group is displayed in the view.
Object group...	Opens the Object Group - View Filter dialog box for creating and modifying object groups.

See also

[Open, save, modify, or delete a view \(page 41\)](#)

Grid view properties

Use the **Creation of Views Along Grid Lines** dialog box to view and modify the properties of grid views.

Option	Description
View plane	The plane of the view defined by two axes similarly to the default view.
Number of views	Defines which grid lines the views will be created of. None does not create any views.

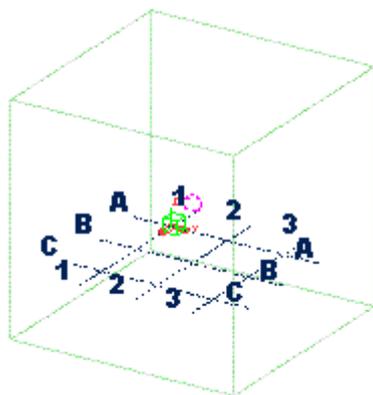
Option	Description
	<p>One (First) only creates the view closest to the grid origin.</p> <p>One (Last) only creates the view furthest from the grid origin.</p> <p>All creates all views in grid planes in the relevant direction.</p>
View name prefix	<p>The prefix to be used with the grid label in the view name. This name overrides the name in the view properties.</p> <p>View names consist of a prefix and a grid label, e.g. PLAN +3000. If the View name prefix box is left empty, no prefix is used. Tekla Structures adds a dash and a running number to the view name if view names are otherwise identical.</p>
View properties	<p>Defines which view properties (applied or saved) will be used.</p> <p>Each view plane has its own view properties. You can load the properties from the current view properties with the option <applied values> or from saved view properties. The Show button displays the current view properties.</p>

See also

[Create model views \(page 31\)](#)

Define the work area

Tekla Structures indicates the work area of a view using dashed lines. Objects outside the work area exist, but they are not visible. You can shrink and expand the work area to suit particular situations, for example to concentrate on a particular area of the model. You can temporarily hide the work area box.



Fit work area to entire model

You can resize the work area to include all model objects, either in all views or in selected views only.

1. On the **View** tab, click **Work area**  and select one of the following:
 - **To entire model in all views**
Fits the work area to include all model objects in all visible views.
 - **To entire model in selected views**
Fits the work area to include all model objects in the selected views.

Fit work area to selected parts

You can resize the work area to include only selected parts, either in all views or in selected views only.

1. Select the objects you want to include.
2. On the **View** tab, click **Work area**  and select one of the following:
 - **To selected parts in all views**
Fits the work area to include the selected model objects in all views.
 - **To selected parts in selected views**
Fits the work area to include the selected model objects in the selected views.

Fit work area using two points

You can resize the work area based on two corner points you pick on the view plane. The depth of the work area is the same as the view depth.

1. On the **View** tab, click **Work area**  and select **Using two points**.
2. Pick the first point.
3. Pick the second point.

Hide the work area box

You can temporarily hide the work area box in a view. This can be useful, for example, when creating screenshots for presentations.

1. Hold down the **Ctrl** and **Shift** keys simultaneously.
2. On the **View** tab, click **Redraw** --> **Redraw all views**.



3. To make the box visible again, click **Redraw** --> **Redraw all views** again.

TIP Alternatively, use the advanced option `XS_HIDE_WORKAREA`.

If you cannot see all objects

The visibility of objects in a view depends on a number of different settings. If you cannot see all the desired objects in a model view, check the following settings:

- work area
- view depth
- view filter
- view and representation settings
- color and transparency settings

Note that work area and view depth are like two virtual boxes. Objects that have their handles partially or totally inside both boxes are visible. Newly created objects are also visible outside the view depth but never outside the work area. When you redraw a view, only the objects inside the view depth are displayed.

See also

[Show and hide model objects \(page 581\)](#)

[View properties \(page 44\)](#)

[Define the work area \(page 46\)](#)

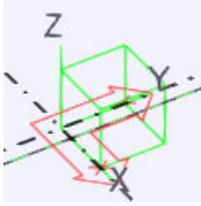
[Filter objects \(page 155\)](#)

Coordinate system

Tekla Structures uses two coordinate systems: the global and the local coordinate system. The local coordinate system is also known as the work plane.

Global coordinate system

The green cube symbol represents the global coordinate system and lies at the global point of origin ($x=0$, $y=0$, $z=0$). The global coordinate system is static and it cannot be changed.

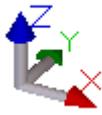


Do not place the model far away from the origin. If you create model objects far away from the origin, [snapping to points \(page 77\)](#) in the model views may become inaccurate. The further away from the origin you model, the less precise all computations become.

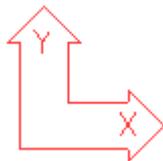
If you need to use another coordinate system for inserting reference models or exporting IFC models, you can use base points. When you use base points, you can keep the coordinates small and locate the model wherever needed.

Local coordinate system (Work plane)

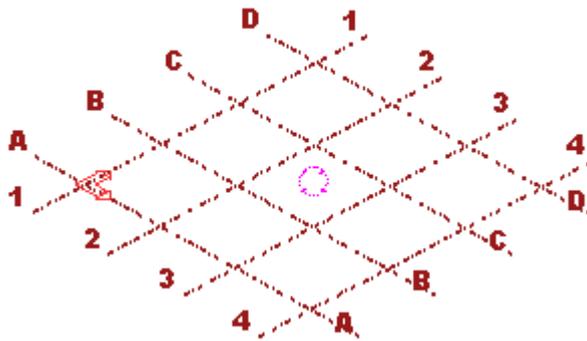
The work plane represents the local coordinate system. Most of the commands that are dependent on the coordinate system use the work plane coordinates. For example creating points, part positioning, and copying always comply with the work plane coordinate system. The coordinate symbol, which is located in the lower right corner of the model view, follows the work plane.



The work plane is model specific, so it is the same in all views. The red work plane arrow symbol shows the xy plane. The z direction follows the [right-hand rule \(page 50\)](#).



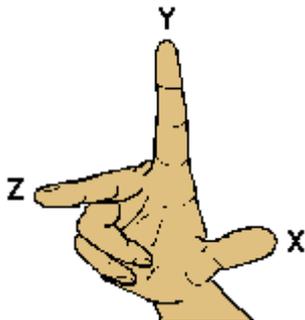
You can change the local coordinate system by shifting the work plane. The work plane also has its own red grid, which can be used for positioning parts. You can [show or hide \(page 50\)](#) this grid as needed.



To control which work plane or base point you currently have in use in the model, use the work plane handler toolbar.

Right-hand rule

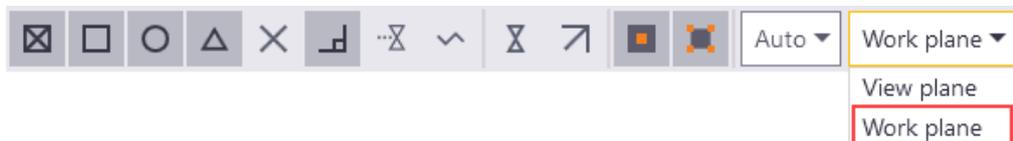
Right-hand rule indicates the direction of the coordinate axes. When you hold the thumb, index finger, and middle finger of your right hand so that they form three right angles, then the thumb symbolizes the x axis, the index finger the y axis, and the middle finger the z axis.



Show or hide the work plane grid

The work plane grid is hidden by default. Use the options on the **Snapping** toolbar to show or hide the work plane grid.

1. To show the grid, select **Work plane** from the second list.



2. To hide the grid, select **View plane** from the same list.

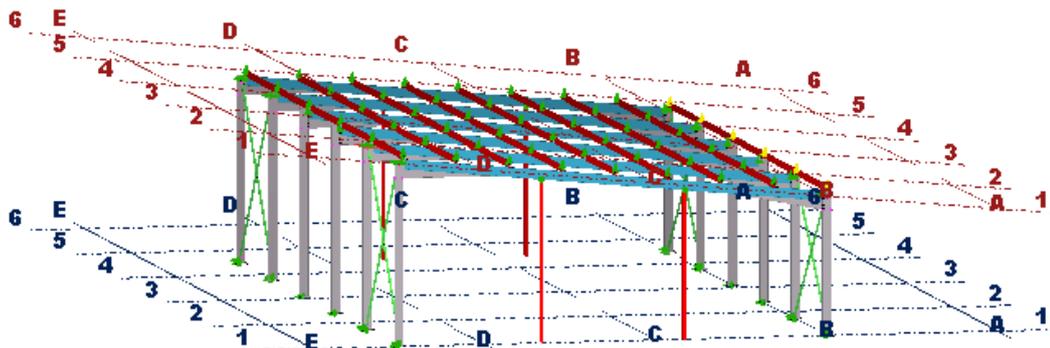
See also

[Shift the work plane \(page 51\)](#)

Shift the work plane

You can set the work plane to any position by picking points or by selecting a plane. This makes it easier to place parts accurately when modeling sloped parts.

For example, you can shift the work plane to the slope of the roof to make it easier to model horizontal bracing and purlins in a sloped roof.



Set work plane to any part plane

Use the **Work plane tool** command to set the work plane to any part plane.

1. On the **View** tab, click **Work plane** --> **Work plane tool**.

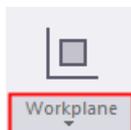


2. Pick a point.

Set work plane parallel to xyz plane

You can set the work plane parallel to the xy, xz, or zy plane.

1. On the **View** tab, click **Work plane** and select **Parallel to XY(Z) plane**.



2. In the **Plane** list box, select the plane parallel to the work plane.
3. Enter the depth coordinate.

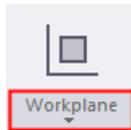
The depth coordinate defines the distance of the work plane from the global origin along a line perpendicular to the plane parallel to the third axis.

4. Click **Change**.

Set work plane using one point

You can set the work plane using one picked point. The work plane stays parallel to the current work plane, but moves it to a new position. The x and y directions are unchanged.

1. On the **View** tab, click **Work plane** and select **Using one point**.

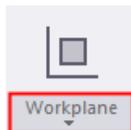


2. Pick the new position of the work plane.

Set work plane using two points

You can set the work plane using two picked points. The first point you pick is the origin, the second defines the x direction of the work plane. The y direction remains the same as the previous work plane.

1. On the **View** tab, click **Work plane** and select **Using two points**.



2. Pick the origin of the work plane.
3. Pick a point in the work plane, in the positive x direction.

Set work plane using three points

You can set the work plane using three picked points. The first point you pick is the origin, the second defines the x direction, and the third defines the y direction of the work plane. Tekla Structures fixes the z direction according to the right-hand rule.

1. On the **View** tab, click **Work plane** and select **Using three points**.

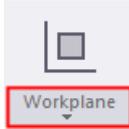


2. Pick the origin for the work plane.
3. Pick a point in the positive x direction.
4. Pick a point in the positive y direction.

Set work plane parallel to view plane

You can set the work plane to be the same as the view plane of a selected view.

1. On the **View** tab, click **Work plane** and select **Parallel to view plane**.

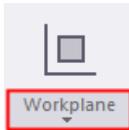


2. Select the view.

Restore the default work plane

Remember to change back to the default work plane when you have finished modeling sloped structures.

1. On the **View** tab, click **Work plane** --> **Parallel to XY(Z) plane**.



2. In the **Plane** list, select **XY**.
3. In the **Depth coordinate** box, enter **0**.
4. Click **Change**.

Base points

The base points (control points) allow you to use a coordinate system based on the civil origin or other coordinate system for interoperability and collaboration. For example, you can use base points when inserting reference models, exporting IFC models, in drawings, in **Layout manager**, and in reports and templates.

Civil origin is the datum point or fundamental benchmark point of the national land survey network.

When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point.

Note the following:

- Reference model should not have any additional lines to the origin.
- Reference models should not include objects that are very far from each other because otherwise using the model may get difficult.
- Tekla Structures native objects including reference models should not be inserted very far from the Tekla Structures model origin.

Define a base point

You can define base points in **Project properties**. In case you need to import or export a reference model, you need to know the coordinates of the

reference model that you are importing, or the coordinates that you want to use in IFC export.

1. Open a Tekla Structures.
2. Click **File --> Project properties --> Base points** to open the **Base point** dialog box.
3. Fill in the needed information:

The screenshot shows the 'Base point' dialog box with the following data:

Field	Value
Name	Trimble Building
Description	Trimble Building in Espoo, Finland
Coordinate system	ETRS-GK25
East coordinate (E)	25489283613.00
North coordinate (N)	6674830501.00
Elevation	3557.00
Latitude	60.186171
Longitude	24.806864
Location in the model (X)	6000.00
Location in the model (Y)	6000.00
Location in the model (Z)	0.00
Angle to North	26.408

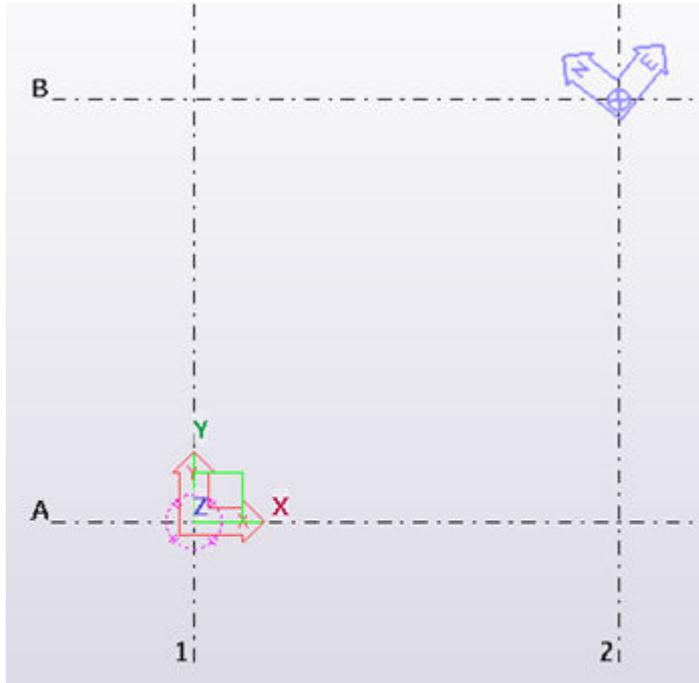
Name, Description	Enter a name and a description for the base point.
Coordinate System	Enter the name of the coordinate system you are using.
East coordinate (E)	Enter the East coordinate (E) that represents the X coordinate related to the civil origin.
North coordinate (N)	Enter the North coordinate (N) that represents the Y coordinate related to the civil origin.
Elevation	Enter the Elevation that represents the Z coordinate related to the civil origin.

Latitude,Longitude	<p>Enter the Latitude and Longitude of the base point to be used in the IFC export.</p> <p>Latitude and Longitude is additional information, which some software can use. In the IFC file, this is written to <code>IFCSITE</code> information.</p> <p>If the total number of digits in Longitude is more than 15, the value is rounded up to the nearest if it is > 99.9999999999999999.</p> <p>To convert Latitude and Longitude information between decimal format and degree/minute/second (DMS) format, see Convert Latitude/Longitude to decimal.</p>
Location in the model	<p>Pick or enter a location for the base point in the Tekla Structures model. The distance is measured from the model origin. The location needs to be near the model origin, preferably less than 1000 m from the origin.</p>
Angle to North	<p>Pick or enter the Angle to North, which is the angle between Y and North directions. The maximum number of decimals for the angle is 13.</p>
Project base point	<p>If you want to set a coordinate system as the project base point, select a base point from the list at the top and then select the Project base point check box.</p>

4. Click **Modify** to save the base point.

A blue symbol is added in the model.

If you later on make changes to the base point, the base point location in the model changes according to the location or rotation changes you make in the **Base point** dialog box when you press **Enter** or click another input field, and there is no need to click **Modify**.



Now you can insert a reference model, or export an IFC model using the defined base point.

Set a coordinate system as the project base point

One base point can be set to be as the project base point. Model origin is the default project base point value if the model does not contain any base points, or if none of the existing base points has been set to be the project base point. You can check and change the current project base point through **File -->**

Project properties --> Location by.

Note that it is not recommended to change the project base point temporarily during a project.

1. Click **File --> Project properties.**

You can see the current project base point in the **Location by** box.

2. To change the project base point, click **Edit**, and select a new project base point from the **Location by** list.
3. Click **Apply**.

TIP You can also set a base point as the project base point in the **Base point** dialog box by selecting a base point from the list at the top and then selecting the **Project base point** check box.

Insert a reference model using a base point

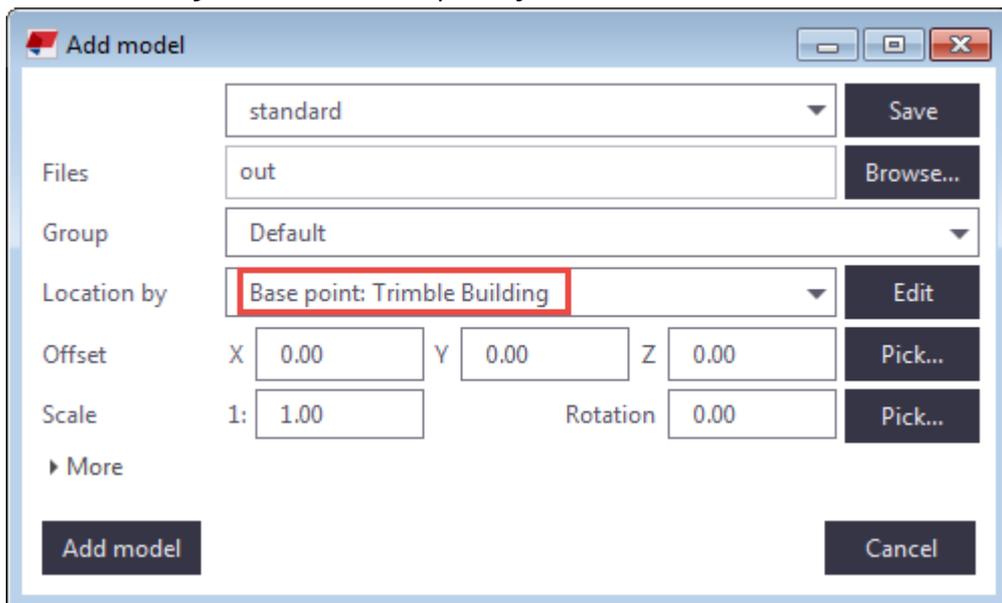
Before you can insert a reference at the base points, you need to create a base point in your model. To create the base point, you need to know the coordinates of the reference model that you are importing.

1. Open the **Reference Models** list by clicking the **Reference Models** button in the side pane .

2. In the **Reference Models** list, click the **Add model** button.
3. In the **Add model** dialog box, if you have any previously created reference model properties files, load the desired file by selecting the file from the properties file list at the top.
4. Browse for the reference model by clicking **Browse...**
5. In **Group**, select a group for the reference model, or enter a name for a new group.

If you do not enter a name for the group, the reference model is inserted in the **Default** group.

6. In **Location by**, select the base point you want to use.



7. Click the **Add model** button. Tekla Structures inserts the reference model relative to the selected base point by using the coordinate system values, elevation and angle in the base point definition in the model **Project properties**.

Export an IFC model using a base point

Before you can export an IFC file using a base point, you need to create a base point in your model.

1. Click **File --> Export --> IFC** to open the **Export to IFC** dialog box.

2. In **Location by**, select a base point that you have created.
3. Fill in other necessary IFC export information.
4. Click **Export**. The base point option exports the IFC model relative to the base point using the coordinate system values, elevation, latitude, longitude and angle in the base point definition in the model **Project properties**.

Base points in drawings

It is possible to use base-point-defined coordinate system values in drawings. If you change the project base point Z or the elevation value, the level value will change accordingly when a drawing is opened.

- Base point data can be used in drawing and view level to set the coordinate system. The base point can be used instead of datum offset.
- When the base point is set, level attributes and template attributes in marks provide values in the specific base-point-defined coordinate system.
- This setting affects level marks and attributes that end with `_BASEPOINT`.
- When the base point is set at the drawing level, the `_BASEPOINT` template attributes can be used in drawing templates.

You can set **Location by** in drawing view properties to use model origin, project base point or any base point defined coordinate system. **Location by** uses the project base point as the default value.

The datum level only affects the attributes `TOP_LEVEL` and `TOP_LEVEL_UNFORMATTED` when **Location by** is set to **Model origin** or to the project base point that is in the model origin.

To change the **Location by** value:

1. In an open drawing, double-click the drawing view frame to open the **View Properties** dialog box.
2. On the **Attributes 2** tab, set **Location by** to a new base point, or to the model origin.
3. Click **Modify**.

Example of base point usage in a drawing

In the following example, do the following:

1. Create a slab with a thickness of 200 mm with the slab's top at the level 0 in the model.

2. Create a new base point "Control point 1" with elevation 20000 mm.

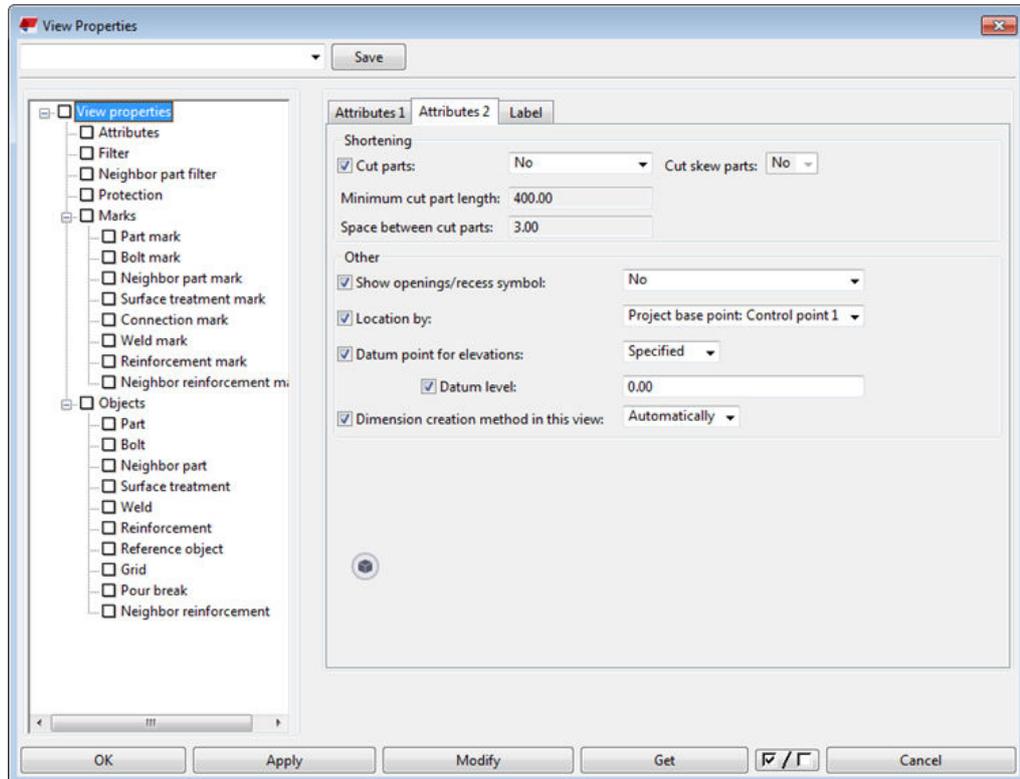
The screenshot shows the 'Base point' dialog box with the following fields and values:

Field	Value
Name	Control point 1
Description	
Coordinate system	
East coordinate (E)	0.00 mm
North coordinate (N)	0.00 mm
Elevation	20000.00 mm
Latitude	0.00
Longitude	0.00
Location in the model (X)	0.00 mm
Location in the model (Y)	0.00 mm
Location in the model (Z)	0.00 mm
Angle to North	0.00

Buttons: Modify, Project base point (checkbox), Zoom to, Pick, Close.

3. Create a GA drawing in plan view.
4. Open the GA drawing, double-click the view frame to open the **View properties** dialog box.

- On the **Attributes 2** tab, you set **Location by** to the new base point (Project base point) "Base point 2" and click **Modify**.

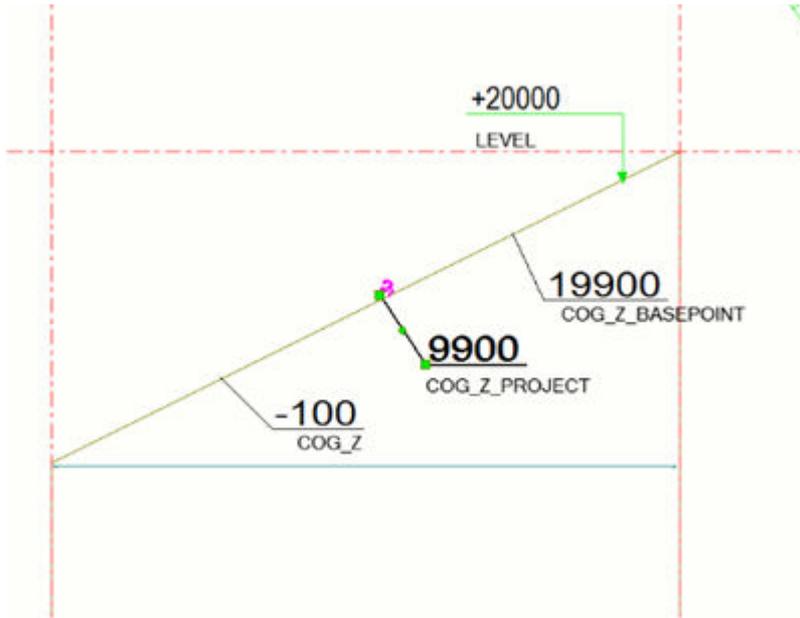


- Add a level mark using the following template attributes:

- COG_Z
- COG_Z_PROJECT
- COG_Z_BASEPOINT

- Reopen the drawing.

Note that changing the value does not update the template attribute automatically but after reopening the drawing.



Base points in Layout manager

You can use base points in **Layout manager** when defining the location of layout points.

- You can use base points as location coordinates when exporting and importing layout points.
- When you add, modify or delete base points, you need to reopen or refresh **Layout manager** to make the changed base point data available in **Layout manager**.

Base point in reports and templates

You can inquire project base point and current base point value in reports and templates.

The following table lists the template attributes where you can use `_PROJECT` and `_BASEPOINT` at the end, for example, `ASSEMBLY_BOTTOM_LEVEL_PROJECT` or `ASSEMBLY_BOTTOM_LEVEL_BASEPOINT`. Note that `_BASEPOINT` uses the current base point in the same way as the workplane uses the current workplane. If there is no current base point defined, `_BASEPOINT` provides values relative to model origin (Global).

Content type	Attributes
ASSEMBLY, CAST_UNIT and PART	ASSEMBLY_BOTTOM_LEVEL ASSEMBLY_BOTTOM_LEVEL_UNFORMATTED ASSEMBLY_TOP_LEVEL ASSEMBLY_TOP_LEVEL_UNFORMATTED

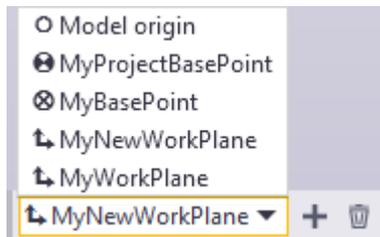
Content type	Attributes
	BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y BOUNDING_BOX_MAX_Z BOUNDING_BOX_X BOUNDING_BOX_Y BOUNDING_BOX_Z COG_X COG_Y COG_Z START_X START_Y START_Z END_X END_Y END_Z TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION ASSEMBLY.LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
REFERENCE MODEL, REFERENCE OBJECT and REFERENCE_ASSEMBLY	BOUNDING_BOX_MIN_X BOUNDING_BOX_MIN_Y BOUNDING_BOX_MIN_Z BOUNDING_BOX_MAX_X BOUNDING_BOX_MAX_Y BOUNDING_BOX_MAX_Z LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION

Content type	Attributes
POUR OBJECT	BOTTOM_LEVEL BOTTOM_LEVEL_UNFORMATTED TOP_LEVEL TOP_LEVEL_UNFORMATTED LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION
CONNECTION	ORIGIN_X ORIGIN_Y ORIGIN_Z
HIERARCHIC OBJECT	LOCATION_BREAKDOWN_STRUCTURE.LBS_FLOOR_ELEVATION

Select the work plane

If you have defined base points or saved work planes, you can use the **Work plane handler** toolbar to select which work plane you currently have in use in the model.

By default, the **Work plane handler** toolbar is located at the bottom of the screen.



You can select among the following work planes:

- Model origin (if project base point is set somewhere else)
- Project base point
- All [base points \(page 53\)](#) you have defined
- All [work planes \(page 51\)](#) you have set and saved

The symbol for model origin is different if the project base point is set to model origin, or if the project base point is set somewhere else.

Add a work plane to the toolbar

1. Set a work plane in the model.
2. On the **Work plane handler** toolbar, enter a name for the work plane in the **Select work plane** box.

3. Click the  button to add the new work plane to the list.
If needed, you can rename your work plane by double-clicking it and entering a new name.

4. To remove a work plane from the list, click the  button.

By default, the **Work plane handler** toolbar is located at the bottom of the screen. If you are unable to find the toolbar, click **File** --> **Settings**, and in the **Toolbars** list ensure that the **Work plane handler toolbar** is selected.

See also

[Coordinate system \(page 48\)](#)

Change the color settings

You can define which color you want to use for dimensions, labels, and background in the model. For example, if you set the background color to black, you may need to adjust the other color settings as well to ensure that the text and dimensions will be visible.

Change the color settings in the **Advanced Options** dialog box by using RGB values on a scale of 0.0 to 1.0. Separate the values with spaces. For example, the color code for yellow is 1.0 1.0 0.0.

TIP Alternatively, if you wish to change the color settings at one go without using the advanced options, you can use the [Background Color Tool](#), which is available in Tekla Warehouse.

Find RGB values for colors

To find the right RGB values for colors, use, for example, the following tools:

- [Background Color Selector](#) tool, which is available in Tekla Warehouse
- [Color picker for Tekla Structures](#) tool, which is available in Tekla User Assistance

Change the model background color

Set the background color using a combination of four different advanced options. You can control the color of each corner of the background separately.

1. On the **File** menu, click **Settings** --> **Advanced Options**, and go to the **Model View** category.
2. Set the background color using the following advanced options:
 - XS_BACKGROUND_COLOR1

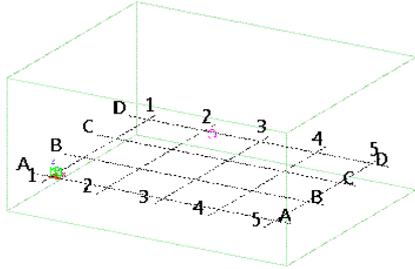
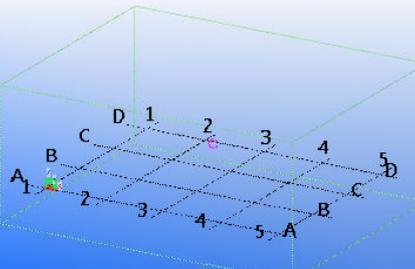
- XS_BACKGROUND_COLOR2
- XS_BACKGROUND_COLOR3
- XS_BACKGROUND_COLOR4

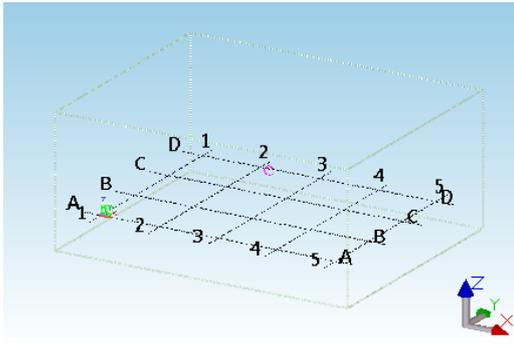
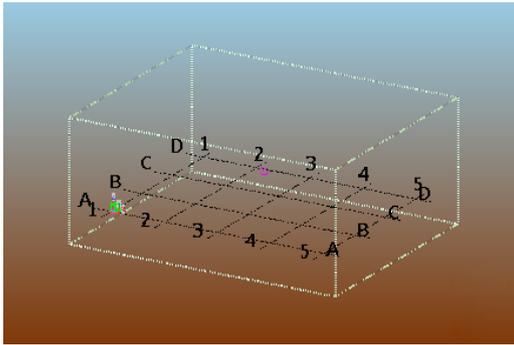
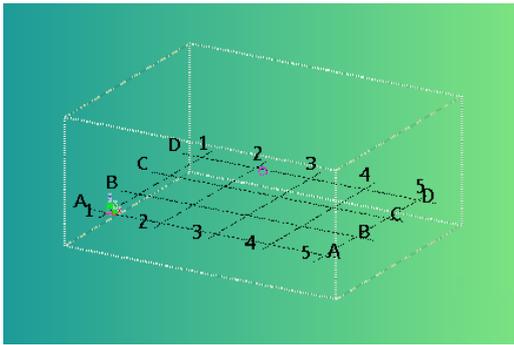
To use a single-colored background, set the same color code for all four corners of the background. To use the default background color, leave the boxes empty.

3. Click **OK** to save the changes.
4. Close and reopen the view to see the changes.

Examples

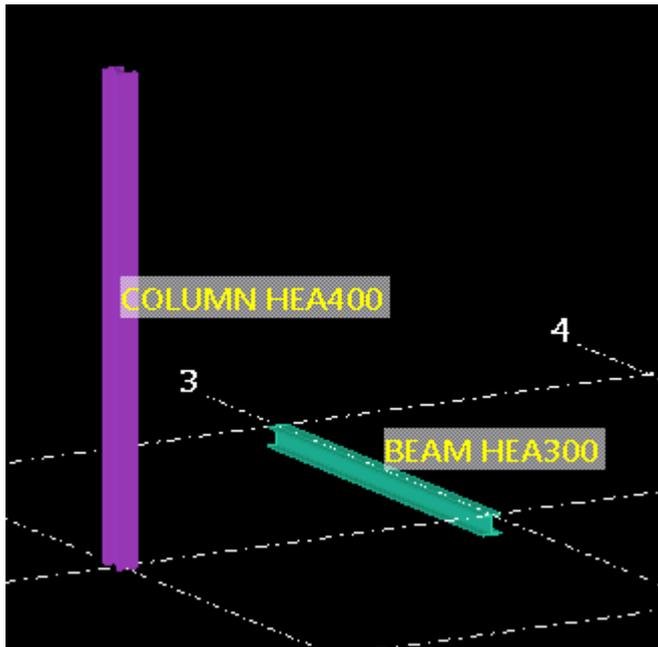
Below are some examples of possible background colors that you can define. The first RGB value refers to the advanced option XS_BACKGROUND_COLOR1, the second value to the advanced option XS_BACKGROUND_COLOR2, and so on.

RGB values	Result
1.0 1.0 1.0 1.0 1.0 1.0 1.0 1.0 1.0 1.0 1.0 1.0	
0.98 0.98 0.99 0.99 0.99 0.99 0.00 0.37 0.99 0.21 0.46 0.88	

RGB values	Result
0.6 0.8 0.9 0.6 0.8 0.9 1.0 1.0 1.0 1.0 1.0 1.0	
0.6 0.8 0.9 0.6 0.8 0.9 0.5 0.2 0.0 0.5 0.2 0.0	
0.1 0.6 0.6 0.5 0.9 0.5 0.1 0.6 0.6 0.5 0.9 0.5	

Change the color of dimensions, part labels, and bolts

You can define which color you want to use for dimensions, part labels, and bolts that are using the representation option **Fast** in the model.



1. On the **File** menu, click **Settings --> Advanced Options** .
2. Search for the color setting you want to change.

Color setting	Advanced option
Dimension lines	XS_VIEW_DIM_LINE_COLOR
Dimension text	XS_VIEW_DIM_TEXT_COLOR
Part labels	XS_VIEW_PART_LABEL_COLOR
Work plane grid	XS_GRID_COLOR_FOR_WORK_PLANE
Bolts using the representation option Fast	XS_VIEW_FAST_BOLT_COLOR

TIP To quickly find all color related advanced options, type `color` in the **Search** box and press **Enter**. Make sure the **In all categories** check box is selected.

3. Define the color using RGB color codes.
4. Click **OK** to save the changes. You may be required to restart Tekla Structures.
5. Close and reopen the view to see the changes.

Change the rendering of the model

You can set Tekla Structures to use the DirectX rendering engine instead of the default OpenGL-based rendering engine. Compared to the OpenGL rendering, the DirectX rendering improves the rendering quality and adds a subtle

shading effect to Tekla Structures objects, making the 3D visualizations clearer and more defined.

The DirectX graphics performance is better on the recommended NVIDIA Geforce GTX graphics cards than with graphics cards that have a lower end or no graphics processing unit (GPU). For more information on the recommended graphics cards, see [Tekla Structures 2019 Hardware Recommendations](#).

You can switch the DirectX rendering on or off in **File --> Settings --> Switches**. The rendering setting is view specific, meaning that you can use the DirectX rendering or the OpenGL rendering in different views. If you switch between the rendering, you need to reopen the view to activate the rendering.

NOTE If you use Tekla Structures via remote connections, the DirectX rendering may not work as expected: parts that you have created may not show in the model, or the model is working slowly. If you experience such problems, switch the DirectX rendering off.

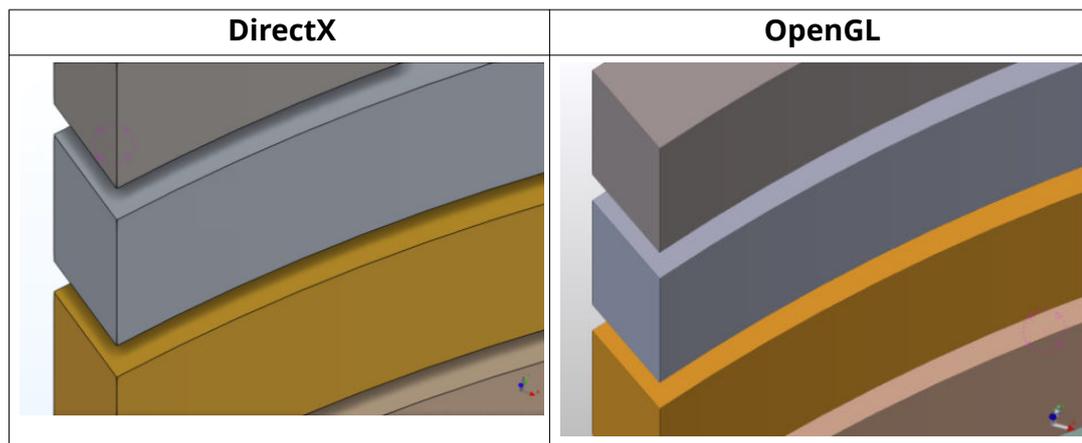
Use the following advanced options to fine-tune the DirectX rendering:

- XS_SHOW_SHADOW_FOR_ORTHO_IN_DX
- XS_SHOW_SHADOW_FOR_PERSPECTIVE_IN_DX
- XS_USE_ANTI_ALIASING_IN_DX
- XS_HATCH_OVERLAPPING_FACES_IN_DX

DirectX rendering examples

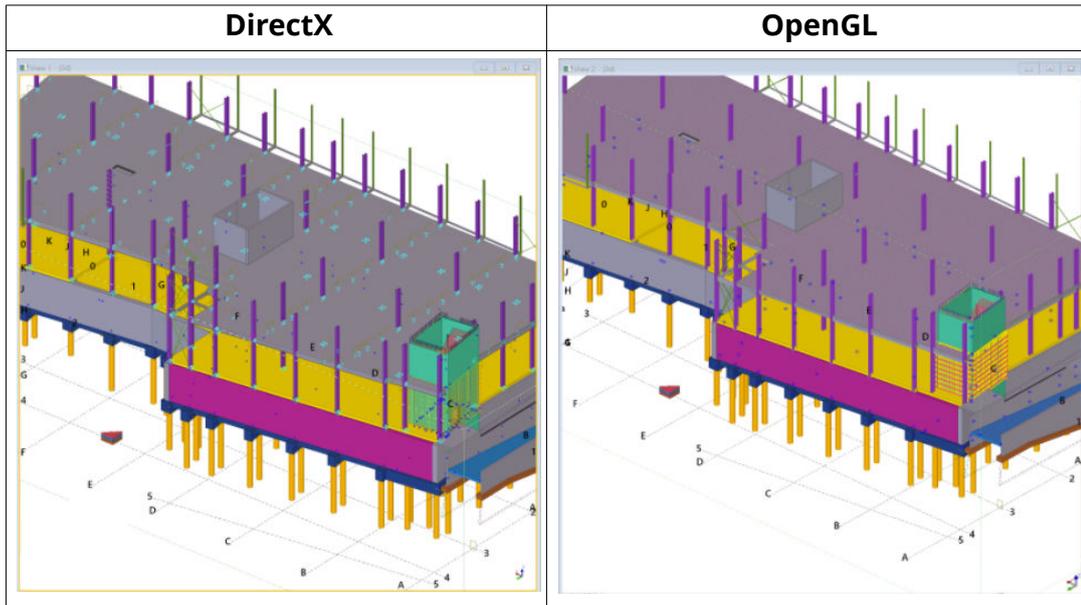
Visualization of distance

The visualization of the distances in the DirectX rendering uses subtle shadows and ambient occlusion. This gives a better understanding of the structure and distances.



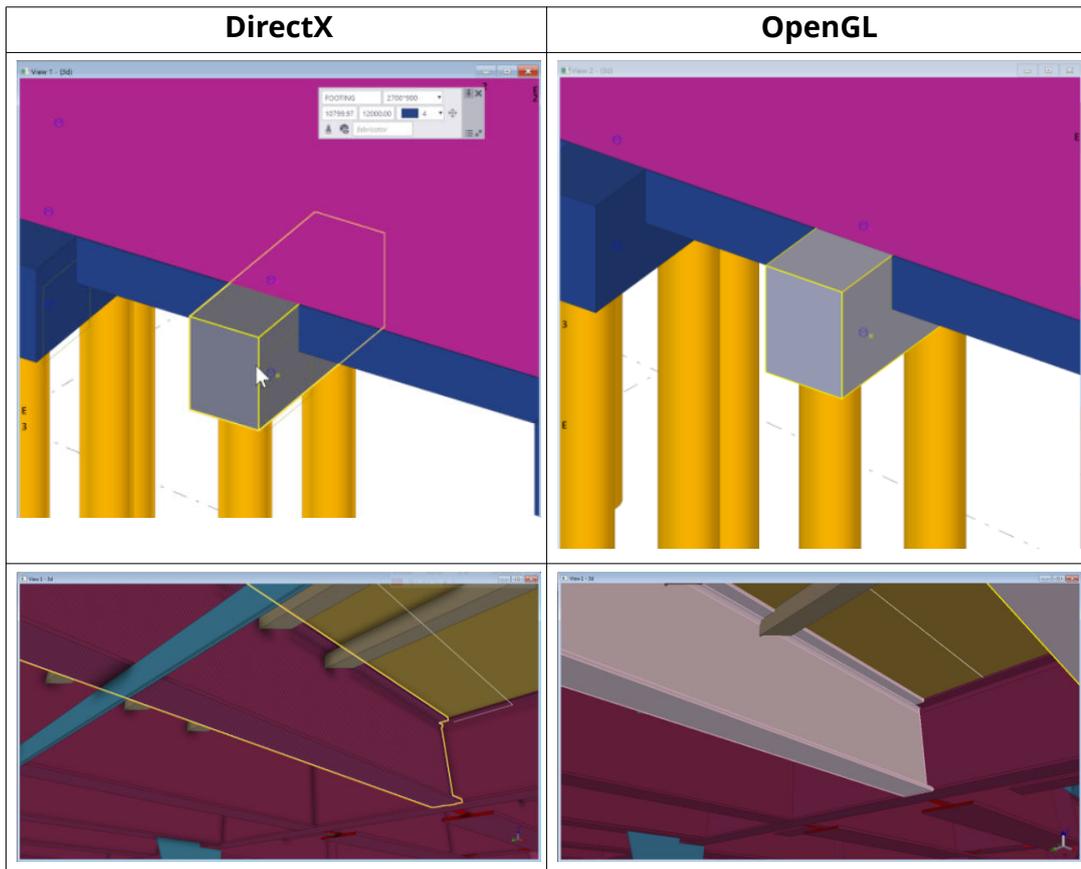
Depth accuracy

The accuracy in depth buffer is improved in the DirectX rendering so that when the model is zoomed, parts are not shown through other parts' faces as often as previously.



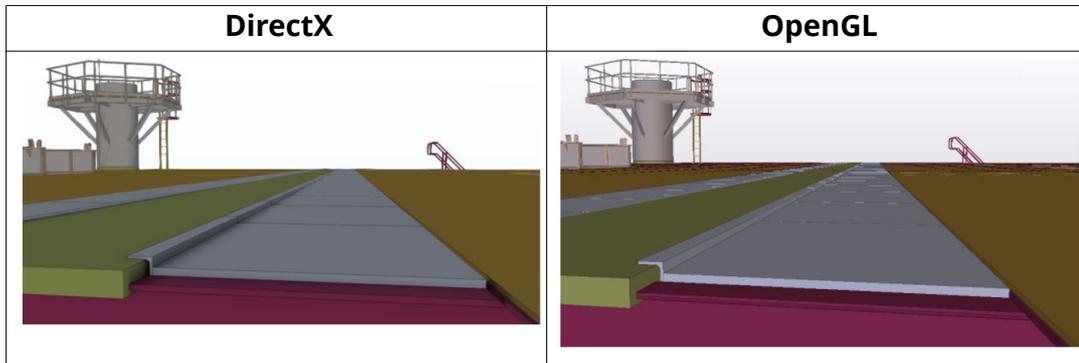
Dynamic states

In dynamic states, such as in selection and in preselection highlight, the selection is clearer and the highlight is less intrusive when the DirectX rendering is used.



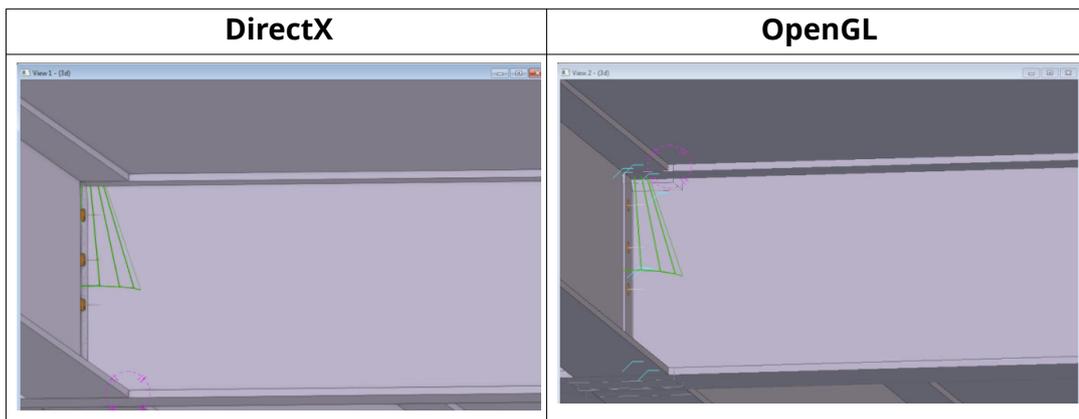
Antialiasing for higher fidelity

In the DirectX rendering the image quality is by default better than in the OpenGL rendering, with less flickering.



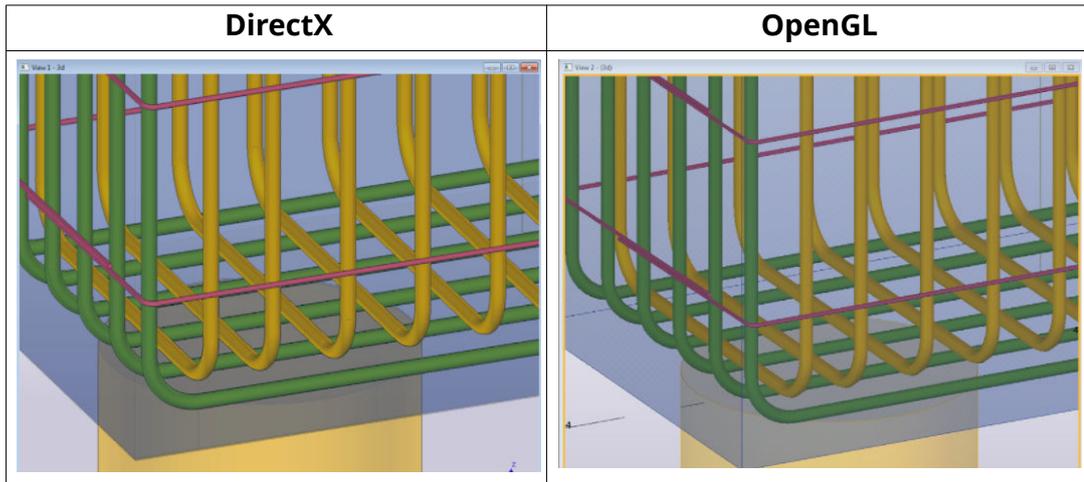
High fidelity edge line

In the DirectX rendering, there are no flickering zigzag lines but continuous, smooth edges.



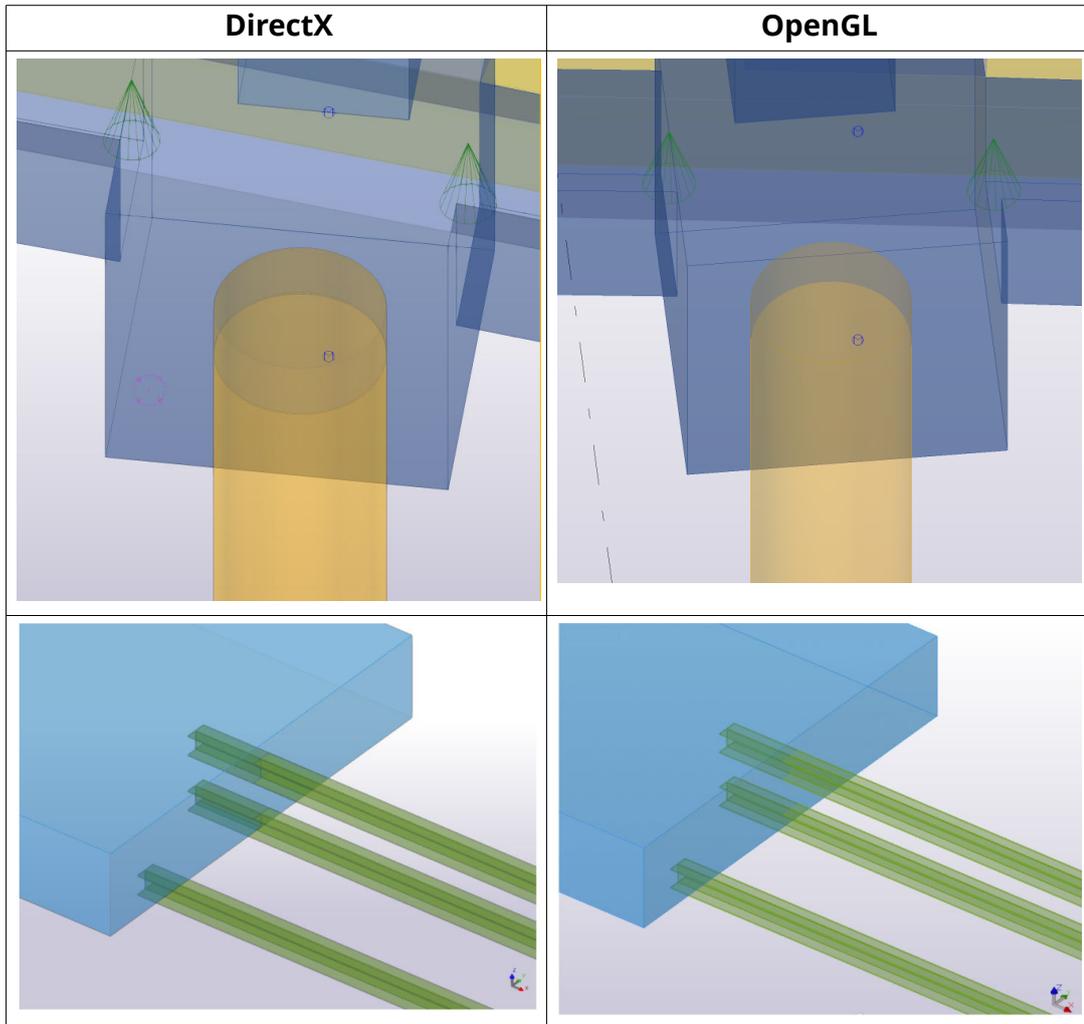
Accurate reinforcing bars

Reinforcing bars have edge lines in the DirectX rendering. When you zoom in, the reinforcing bars are shown as round.

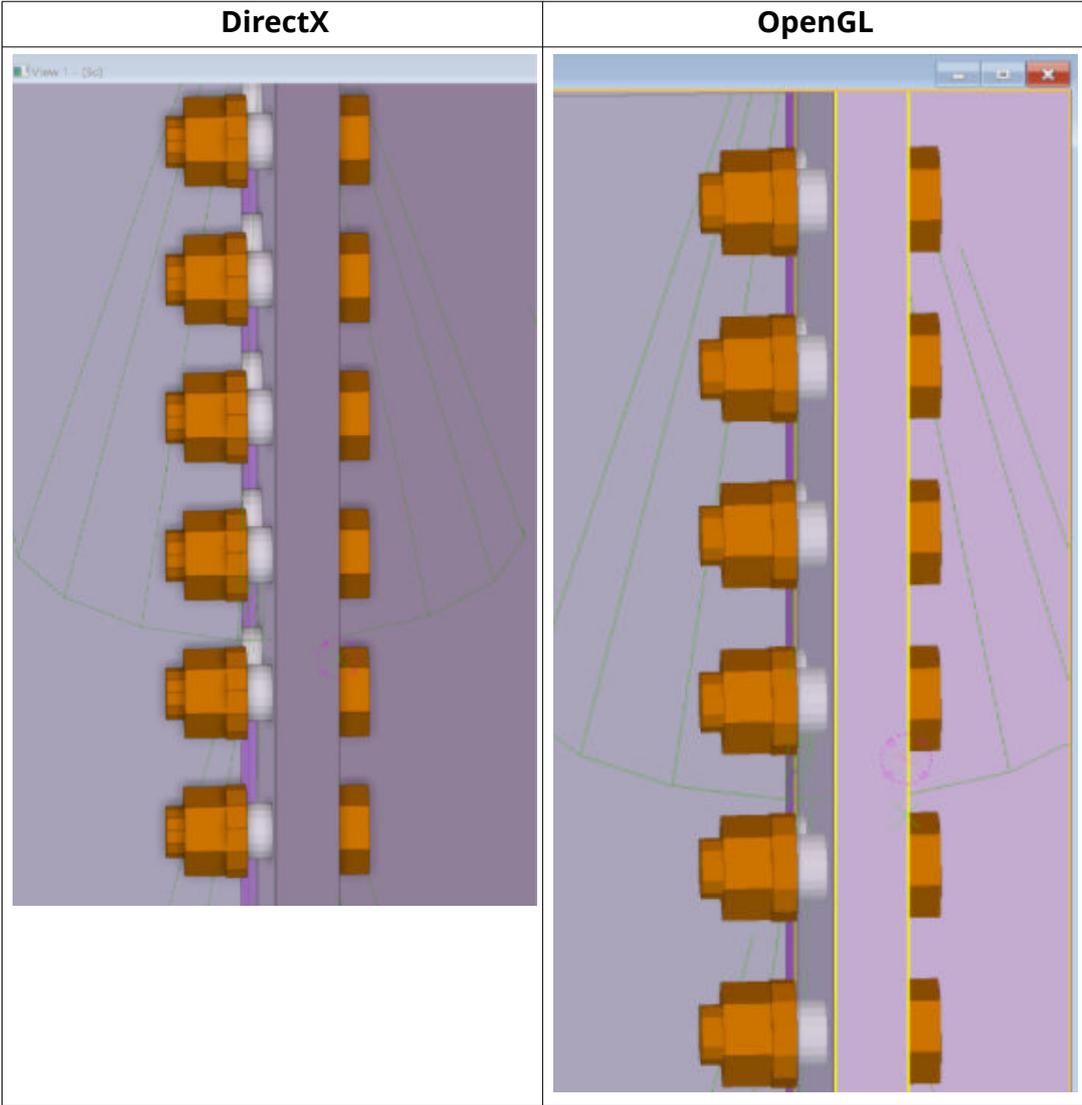


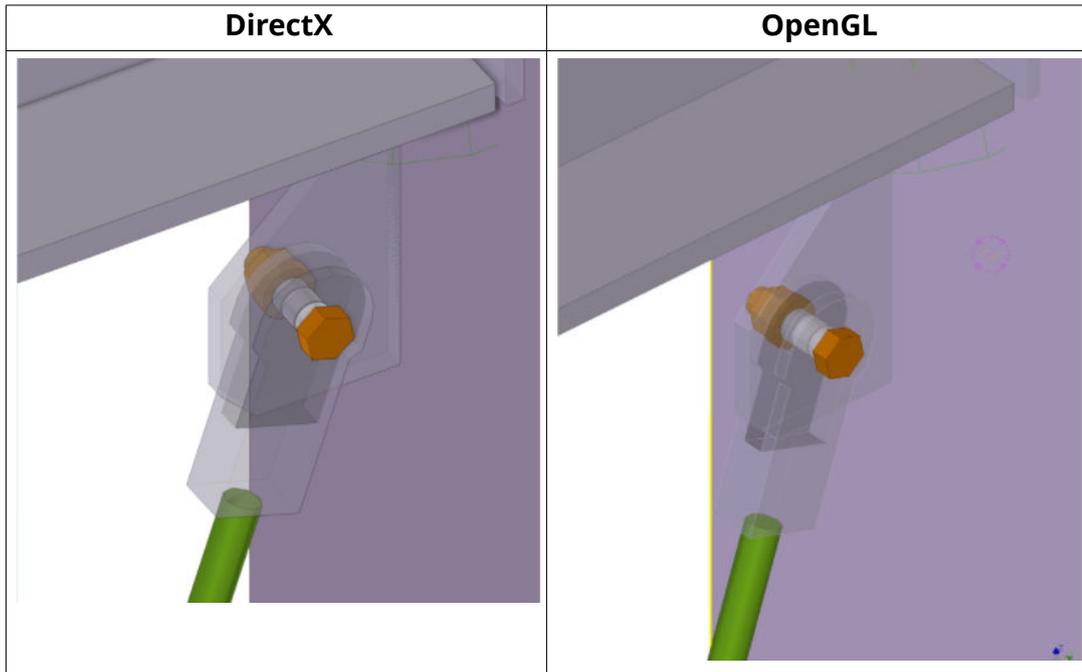
Automatic edge lines for intersecting material in transparent view

In the DirectX rendering, you can see where there are intersecting materials in the model.



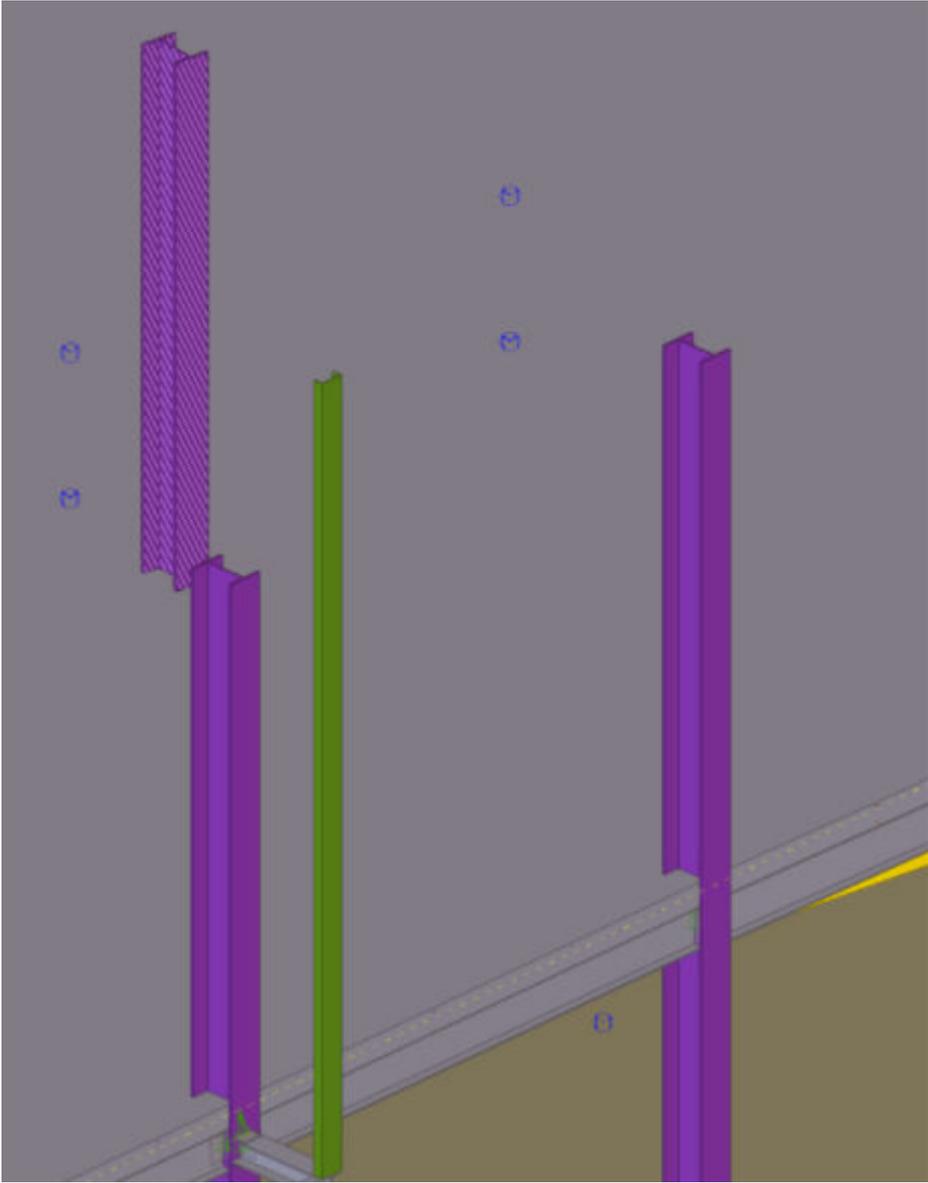
Accuracy and clarity on details

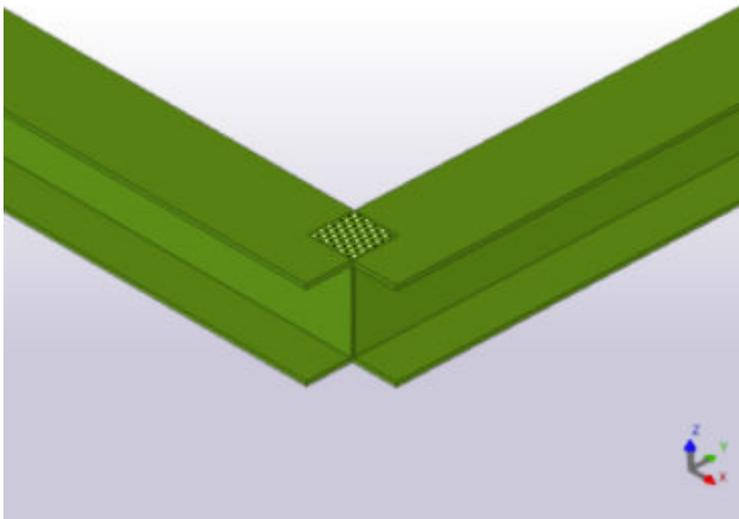




Automatic hatching for overlapping surfaces on same plane

You can detect duplicate objects or any overlapping parts in the DirectX rendering.





1.2 Zoom and rotate the model

The commands on the **View** tab allow you to focus on a particular area, or pull out for a wider view. You can use a mouse, command, keyboard shortcut, or a combination of these.

Zoom in and out

You can use a variety of tools to zoom in and out in the model. By default, the mouse pointer position determines the center point of zooming.

To	Do this
Zoom in	Scroll forward with the mouse wheel. Alternatively, press Page Up .
Zoom out	Scroll backward with the mouse wheel. Alternatively, press Page Down .
Zoom to selected objects	<ol style="list-style-type: none"> 1. Select the objects. 2. On the View tab, click  Zoom -- > Zoom selected.
Zoom with menu commands	On the View tab, click  Zoom and select one of the zoom commands.
Keep the center point of zooming in the middle of the view	On the File menu, click Settings and select Centered zooms .

To	Do this
Define the zoom ratio	Use these advanced options: XS_ZOOM_STEP_RATIO XS_ZOOM_STEP_RATIO_IN_MOUSEWHEEL_MODE XS_ZOOM_STEP_RATIO_IN_SCROLL_MODE

Rotate the model

You can use either the middle or left mouse button, or keyboard, to rotate the model in a view.

To	Do this
Rotate using the middle mouse button	<ol style="list-style-type: none"> 1. On the View tab, click  Navigate --> Set view point. You can also press V. 2. To set the view point, pick a position in the view. The following symbol appears in the model:  3. Hold down the Ctrl key, and click and drag the model with the middle mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using the left mouse button	<ol style="list-style-type: none"> 1. On the View tab, click  Navigate --> Rotate with mouse. You can also press Ctrl+R. 2. To set the view point, pick a position in the view. The following symbol appears in the model: 

To	Do this
	3. Click and drag the model with the left mouse button. Tekla Structures rotates the model around the view point you defined in step 2.
Rotate using keyboard	Use the keyboard shortcuts Ctrl+arrow keys and Shift+arrow keys . Ctrl+arrow keys rotates the model in 15 degree increments. Shift+arrow keys rotates the model in 5 degree increments.

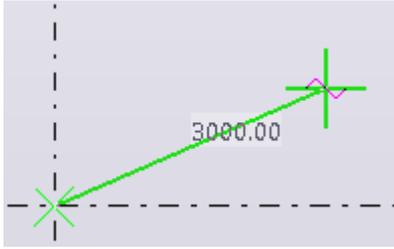
Pan the model

You can use either the middle or left mouse button to pan the model in a view.

To	Do this
Move the model using the middle mouse button	1. On the File menu, click Settings and check that the Middle button pan check box is selected. 2. Hold down the middle mouse button and drag the model.
Move the model using the left mouse button	1. To activate dynamic panning, go to the View tab and click  Navigate --> Pan . You can also press P . The mouse pointer changes to a hand:  2. Hold down the left mouse button and drag the model. 3. To stop panning, press Esc .

1.3 Snap to positions

Most commands ask you to pick points to place objects in the model or drawing. This is called *snapping*. When you are creating a new object, Tekla Structures displays snap symbols for the available snap points and displays a green line between the snap point and the last point picked.



Use the [snap switches \(page 79\)](#) on the Snapping toolbar to control which positions you can snap to.

For example, you can snap to

- different points, such as end points and midpoints
- centers
- intersections
- lines and edges
- dimensions and mark lines, drawing layout items and drawing frames

If you want to use exact distances or coordinates when snapping to positions, use numeric snapping.

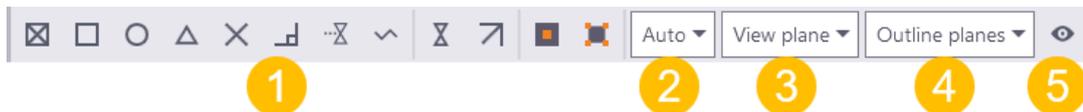
With the combination of different snapping tools, you can, for example, snap to the closest orthogonal point on the plane, both in the model and in the drawings. Additionally, you can follow a line and pick a point at a specified distance along the line, or create a temporary reference point to use as a local origin, both in the model and in the drawings.

Tekla Structures displays snap dimensions in the model, which means you can easily create objects of a desired length. Use the advanced option `XS_DISPLAY_DIMENSIONS_WHEN_CREATING_OBJECTS` to switch the snap dimensions on or off.

TIP Use the snapping keyboard shortcuts to speed up your work.

Snapping toolbar

Use the **Snapping** toolbar to activate snap switches and to access additional snapping options.



(1) Use the [snap switches \(page 79\)](#) to control which positions you can pick when placing objects. Snap switches define exact locations on objects, for example, end points, midpoints, and intersections.

(2) Use the first list to define the snap depth. For more information, see the separate instructions further along on this page.

(3) Use the second list to switch between the view plane and [work plane \(page 50\)](#).

(4) Use the third list to set the plane type. The plane type defines what planes you can select in the model.

(5) You can [hide \(page 225\)](#) selected switches from the toolbar.

By default, the **Snapping** toolbar is located at the bottom of the screen. If you are unable to find the toolbar, click **File** --> **Settings** , and in the **Toolbars** list ensure that the **Snapping** toolbar is selected.

Snap zone

Each object has a snap zone. It defines how close you need to pick to hit a position. When you pick within the snap zone of an object, Tekla Structures automatically snaps to the closest pickable point on that object.

You can set the snap zone using the advanced option XS_PIXEL_TOLERANCE.

Snap priority

If you pick and hit several positions simultaneously, Tekla Structures automatically snaps to the point with the highest snap priority. To control which positions you can pick, use snap switches. Snap switches define the snap priority of positions.

Snap depth

The first list on the **Snapping** toolbar defines the depth of each position you pick. You have the following options:

- **Plane:** You can snap to positions either on the [view plane \(page 29\)](#) or the [work plane \(page 48\)](#), depending on what you have selected in the second list on the **Snapping** toolbar.
- **Auto:** In perspective views, this option works like the **3D** option. In non-perspective views, it works like the **Plane** option.
- **3D:** You can snap to positions in the entire 3D space.

Snapping in drawings

See Snapping in drawings.

Snap switches and symbols

Use the snap switches to control which positions you can pick in the model or drawing. By using snap switches, you can position objects precisely without having to know the coordinates. You can use snap switches whenever Tekla Structures prompts you to pick a point.

Click the snap switches on the **Snapping** toolbar to switch them on or off. If there is more than one point available to snap to, press the **Tab** key to cycle forward through the snap points, and **Shift+Tab** to cycle backwards through them. Click the left mouse button to select the appropriate point.

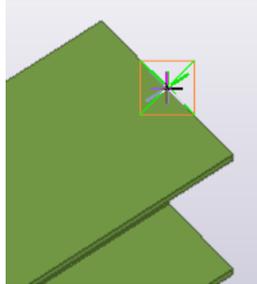
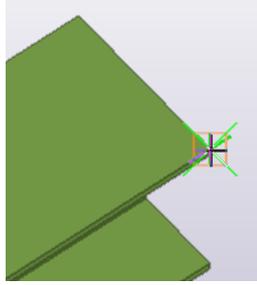
Alternatively, you can control the snap switches with **Quick Launch**. Start typing the name of the snap switch, for example, `snap`, and click the name of the snap switch on the search results list to activate the switch.

Main snap switches

The two main snap switches define whether you can snap to reference points or any other points on objects, for example part corners. These switches have the highest [snap priority \(page 79\)](#). If both these switches are off, you cannot snap to any positions, even if all the other switches are on.

The snap symbols have two colors:

- Orange for part or object snap
- Green for component snap

Switch	Snap positions	Description	Symbol
	Reference lines and points	You can snap to object reference points (points that have handles).	Large 
	Geometry lines and points	You can snap to any point on an object. In drawings, you can use this switch to snap to snapshot overlays.	Small 

Other snap switches

The table below lists the remaining snap switches and their symbols in the model and drawing.

Make sure that you do not have too many snap switches on when snapping, as it may easily lead to inaccuracies and errors in snapping. Be particularly careful

when you use the  **Snap to any position** snap switch.

Switch	Snap positions	Description	Symbol
	Points	Snaps to points and grid line intersections.	
	End points	Snaps to end points of lines, polyline segments, and arcs.	
	Centers	Snaps to centers of circles and arcs. In a drawing, when you want to snap to the center point of a circle that has been created by using a polygon cut in the model, set the advanced option <code>XS_ADD_SNAPPING_SYMBOL_TO_CIRCLES</code> to <code>TRUE</code> .	
	Midpoints	Snaps to midpoints of lines, polyline segments, and arcs.	
	Intersections	Snaps to intersections of lines, polyline segments, arcs, and circles.	
	Perpendicular	Snaps to points on objects that form a perpendicular alignment with another object.	
	Line extensions	Snaps to the line extensions of nearby objects, and reference and geometry lines of drawing objects.	
	Any position	Snaps to any position.	
	Nearest point	Snaps to the nearest points on objects, e.g. any point on part edges or lines.	

Switch	Snap positions	Description	Symbol
	Lines	Snaps to grid lines, reference lines, and the edges of existing objects.	
	Dimensions and mark lines, drawing layout items and drawing frames	Snaps to annotation geometries, drawing layout items and drawing frames. Available only in drawings.	

Override the current snap switch

You can temporarily override the current snap switch settings.

- Run a command that asks you to pick a point.
For example, start creating a beam.
- To override the current snap switches, do one of the following:
 - Right-click to bring up a list of snap options, and then select one of the options.
 - Click **File** --> **Settings** , and in the list of toolbars select **Snap override toolbar**.

A new toolbar appears. Click a button to override the current snap switch.



See also

[Snap to positions \(page 77\)](#)

[Snap settings \(page 94\)](#)

Snap in orthogonal directions

You can snap to orthogonal points in models and in drawings using the **Ortho** tool. If you create objects that require you to pick multiple points, you can snap in orthogonal directions relative to the two previously picked points.

Snap to orthogonal points

Use the **Ortho** tool to snap to the closest orthogonal point on the plane (0, 45, 90, 135, 180 degrees, and so on). The mouse pointer automatically snaps to positions at even distances in the given direction. This can be useful, for

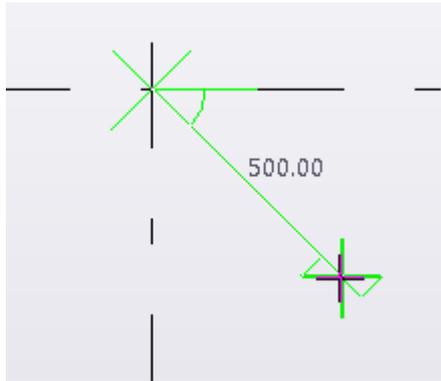
example, if you need to place marks in a consistent manner in exact locations in a drawing.

1. On the **File** menu, click **Settings** and select the **Ortho** check box.

Alternatively, you can press **O**.

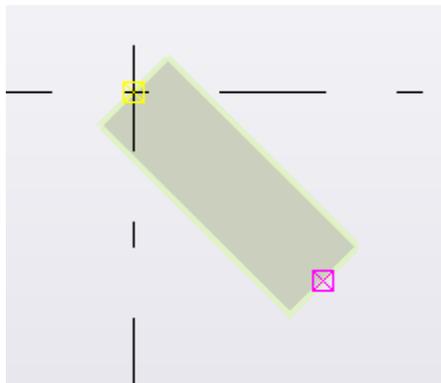
2. Run a command that requires you to pick points.

For example, start creating a beam. Tekla Structures displays an angle symbol to indicate the direction of snapping. The snapping precision depends on the current zoom level.



3. Click the left mouse button to confirm the snap position.

Tekla Structures creates the object. For example:



Snap in orthogonal direction relative to previously picked points

When you create objects that require you to pick more than two points, for example when you create a polybeam or a contour plate, you can snap in orthogonal directions relative to the two previously picked points. This can be useful, for example, if you need to create a rectangular slab that is on the view plane but not along the x and y axes.

1. Run a command that requires you to pick multiple points.

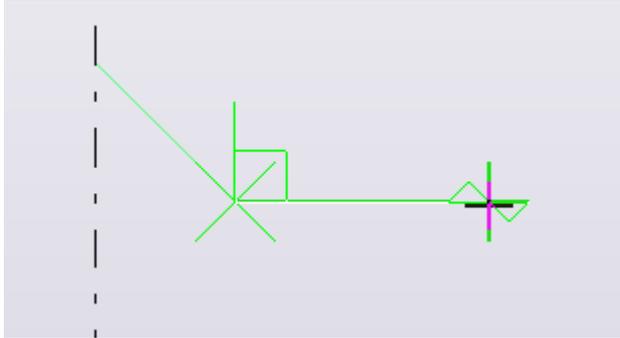
For example, start creating a polybeam or a rectangular slab.

2. Pick the first two points.

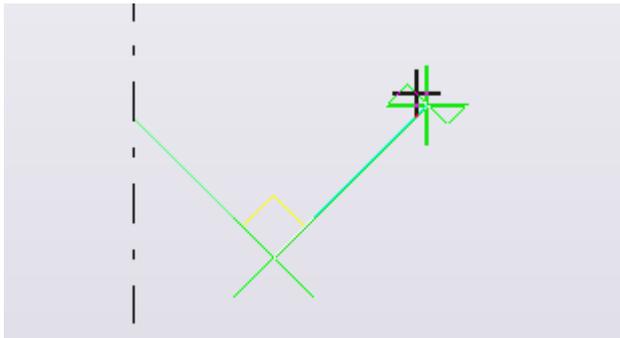
Tekla Structures displays an angle symbol to indicate the direction of snapping.

3. Move the mouse pointer in the model to see the angle symbol.

When the snap is orthogonal to the work plane, the color of the angle symbol is green:



When the snap is orthogonal to the previous points, the color of the angle symbol changes to yellow:



4. Pick the rest of the points.

Tekla Structures creates the object. For example:



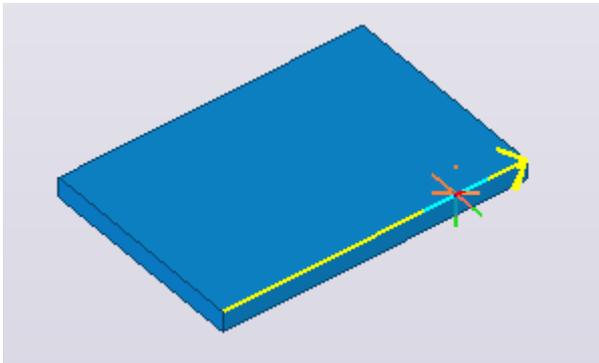
Snap to lines

You can snap to lines when you model objects that should be lined up with an existing object or a grid line, or you can snap to the extension lines of nearby objects.

Snap to a line

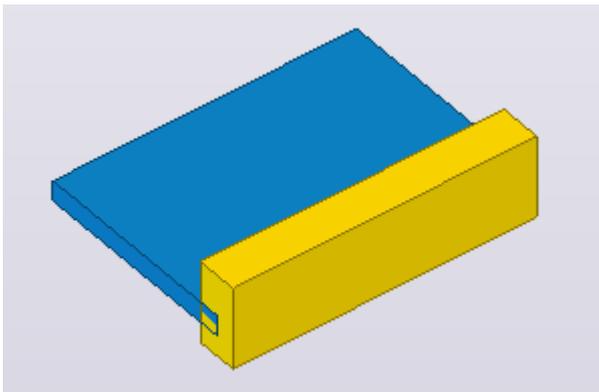
1. Ensure that the  **Snap to line** snap switch (page 79) is active.
2. Run a command that requires you to pick two or more points.

For example, start creating a beam. When you move the mouse pointer over a nearby object, Tekla Structures automatically picks both ends of the line. The yellow arrow symbol indicates the direction of the points.



3. To switch direction, move the mouse pointer closer to the opposite end of the line.
4. Click the left mouse button to confirm the snap position.

Tekla Structures creates the object. For example:



Snap to extension lines

You can snap to the extension lines of nearby objects. This can be useful, for example, when you want to align objects with one another.

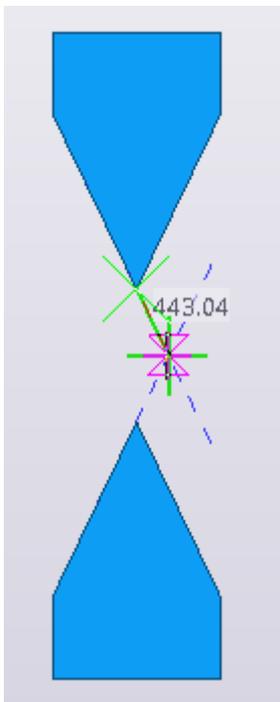
1. Ensure that the correct [snap switches \(page 79\)](#) are active:

- Switch on  **Snap to extension lines.**
- Switch on either  **Snap to intersection points** or  **Snap to nearest points (points on line)** if you are snapping to the intersection of an extension line and a grid line.
- Switch off  **Snap to end points** if you are working in 3D.

2. Run a command that requires you to pick points.

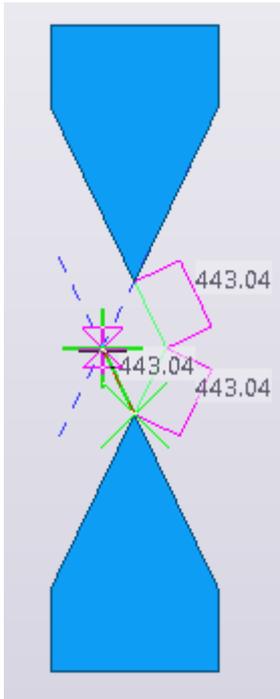
For example, start creating a plate or slab.

3. Move the mouse pointer close to nearby objects to see the extension lines.

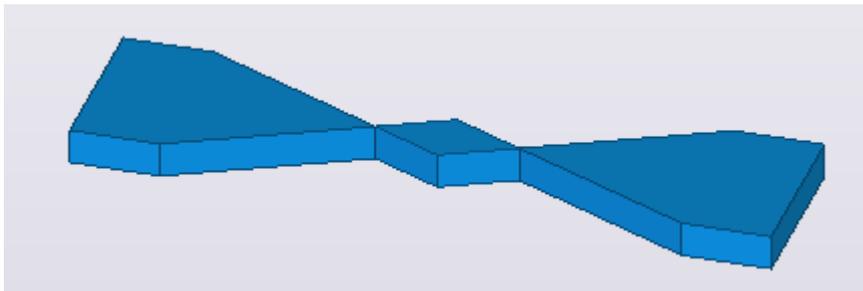


When a line is found, you can move the pointer further away while keeping the snap.

4. Pick the rest of the points.



Tekla Structures creates the object:



Snap to points using exact distance or coordinates - numeric snapping

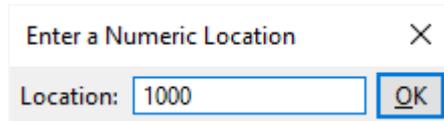
You can enter exact distances and coordinates when snapping to a position. This is called *numeric snapping*.

Enter a distance or coordinates

Use the **Enter a Numeric Location** dialog box to specify the distance or coordinates to a position you want to snap to.

1. Run a command that requires you to pick points.
For example, start creating a beam.
2. Enter a distance or coordinates by using the keyboard.

For example, type 1000 as the distance from the last point picked. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.

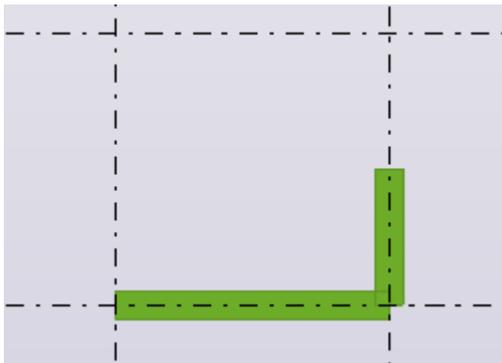


3. After entering the distance or coordinates, click **OK** or press **Enter** to snap to the position.

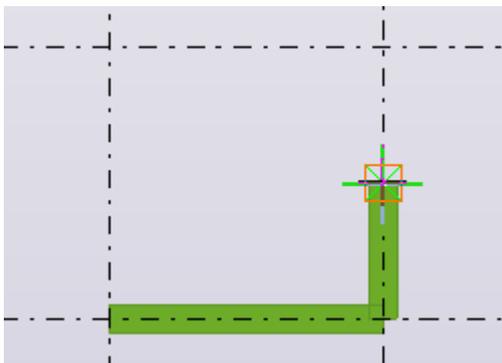
Snapping example: Track along a line towards a snap point

Tracking means that you follow a line and pick a point at a specified distance along the line. You usually use tracking in combination with numeric coordinates and other snapping tools, such as snap switches and orthogonal snapping. This example shows how to pick a point at a specified distance along a line. Use the **Enter a Numeric Location** dialog box to specify the distance from the last point picked.

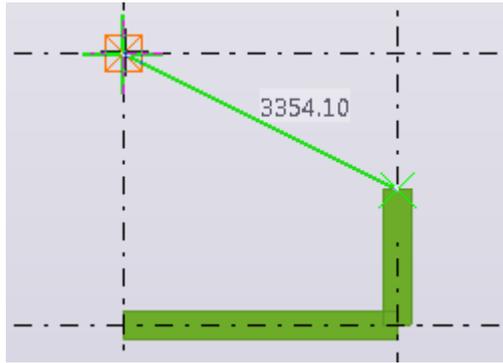
1. Create two beams and place them as shown below:



2. Activate the beam command, to create one more beam.
3. Pick the first point.

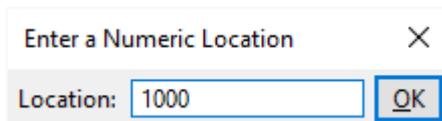


4. Move the mouse pointer over the grid line midpoint, so that it locks onto the snap point, but **do not** click the mouse button.



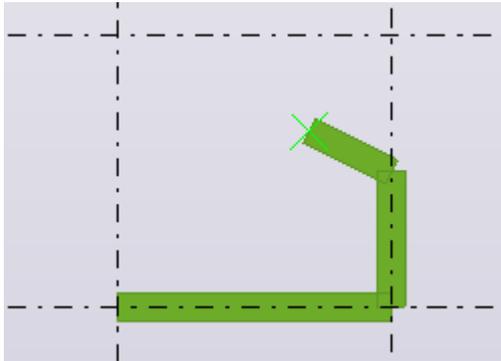
5. Type 1000.

When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.



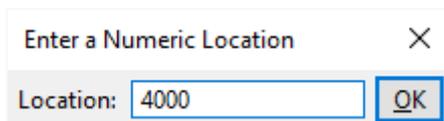
6. Click **OK** to confirm the distance.

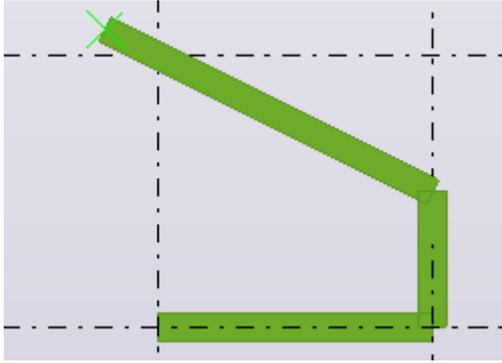
Tekla Structures creates a beam, which is 1000 units long and positioned between the points you defined:



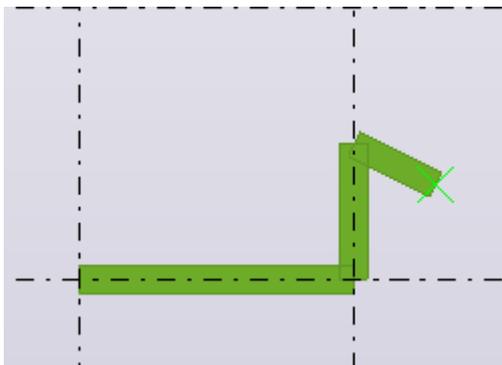
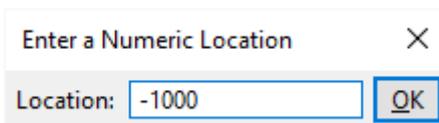
You can also:

- Track beyond the snap point, for example 4000 units from the first point:





- Track in the opposite direction by entering a negative value, for example -1000:



For an example of how to use numeric snapping in drawings, see Place a sketch object at a specified distance.

Options for coordinates

The table below explains the types of information you can enter in the **Enter a Numeric Location** dialog box.

Note that Tekla Structures has three *snapping modes*: relative, absolute, and global. You can temporarily override the default snapping mode by using a special character in front of the coordinates in the **Enter a Numeric Location** dialog box.

You can enter	Description	Special character
One coordinate	A distance to an indicated direction.	

You can enter	Description	Special character
Two coordinates Three coordinates	If you omit the last coordinate (z) or angle, Tekla Structures assumes that the value is 0. In drawings, Tekla Structures ignores the third coordinate.	
Cartesian coordinates	The x, y, and z coordinates of a position separated by commas. For example, 100, -50, -200.	, (comma)
Polar coordinates	A distance, an angle on the xy plane, and an angle from the xy plane separated by angle brackets. For example, 1000<90<45. Angles increase in the counterclockwise direction.	<
Relative coordinates	The coordinates relative to the last position picked. For example, @1000, 500 or @500<30.	@
Absolute coordinates	The coordinates based on the origin of the work plane. For example, \$0, 0, 1000	\$
Global coordinates	The coordinates relative to the global origin and the global x and y directions. For example, !6000, 12000, 0. This is handy, for example, when you have set the work plane to a part plane and want to snap to a position defined in the global coordinate system without changing the work plane to global.	!

Change the snapping mode

Tekla Structures has three snapping modes: relative, absolute, and global. Use the advanced option `XS_KEYIN_DEFAULT_MODE` to indicate the default snapping mode.

1. On the **File** menu, click **Settings** --> **Advanced Options** and go to the **Modeling Properties** category.
2. Set the advanced option `XS_KEYIN_DEFAULT_MODE` to `RELATIVE`, `ABSOLUTE`, or `GLOBAL`.

- In the relative snapping mode, the coordinates you enter in the **Enter a Numeric Location** dialog box as such without any prefix are relative to the last position picked.
 - In the absolute snapping mode, the coordinates are based on the origin of the work plane.
 - In the global snapping mode, the coordinates are based on the global origin and the global x and y directions.
3. Click **OK** to save the changes.
 4. If you want to temporarily override the default snapping mode, enter a special character in front of the coordinates when entering a numeric location.

By default, the special characters are:

- @ for relative coordinates
- \$ for absolute coordinates
- ! for global coordinates

NOTE If you want to change the special character for any of the three snapping modes, use the advanced options `XS_KEYIN_RELATIVE_PREFIX`, `XS_KEYIN_ABSOLUTE_PREFIX`, and `XS_KEYIN_GLOBAL_PREFIX`.

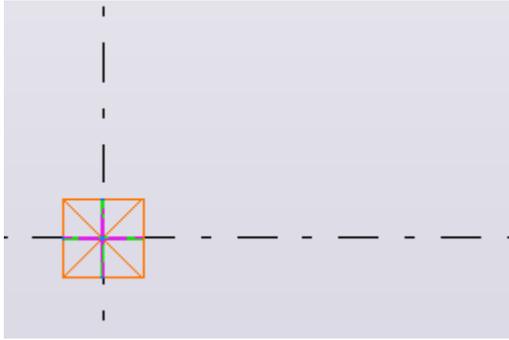
Snapping aids

In addition to the snap switches and the numeric snapping, you can use some additional methods that help you to pick the correct positions when snapping.

Create a temporary reference point

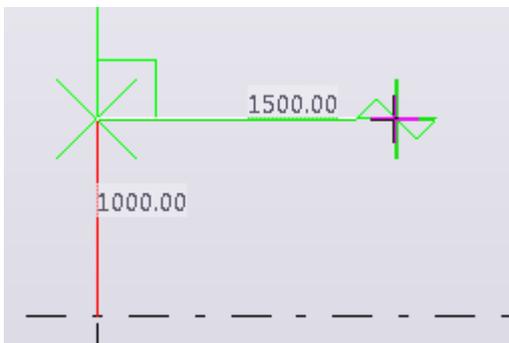
You can create a temporary reference point to be used as a local origin when snapping in models and drawings. Temporary reference points can be used in combination with other snapping tools, such as [snap switches \(page 79\)](#) and [orthogonal snapping \(page 82\)](#).

1. Run a command that requires you to pick points.
For example, start creating a beam.
2. Pick the start point.



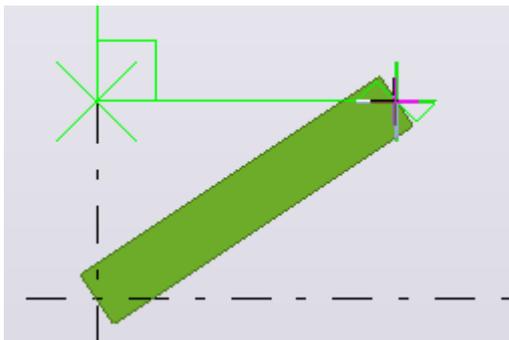
3. Hold down the **Ctrl** key and pick a position.

A green cross indicates that this position is now a temporary reference point.



4. Repeat step 3 to create as many reference points as needed.
5. Release the **Ctrl** key and pick the end point.

Tekla Structures creates the object between the start point and the end point. For example:



Lock X, Y, or Z coordinate on a line

You can lock the x, y, and z coordinates on a line. This is useful when you need to determine a point to pick and the needed point does not exist on the line. When a coordinate is locked, you can only snap to points in that direction.

1. Run a command that requires you to pick positions.
For example, start creating a beam.
2. Lock a coordinate:
 - To lock the x coordinate, press **X**.
 - To lock the y coordinate, press **Y**.
 - To lock the z coordinate, press **Z**.Now you can only snap to points in the chosen direction.
3. To unlock the coordinate, press the same letter (**X**, **Y**, or **Z**) again.

Align objects using a snap grid

A snap grid makes it easier to align objects in a model, because it allows you to snap to positions only at [set intervals \(page 94\)](#). Use a snap grid when you

pick points using the  **Snap to any position** [snap switch \(page 79\)](#).

1. On the **File** menu, click **Settings** --> **Snap settings**.
2. Define the grid spacing intervals in the **Spacing** boxes.
For example, if the spacing of the x coordinate is 500, you are able to snap to positions at intervals of 500 units in the x direction.
3. If needed, define offsets for the snap grid origin in the **Origin** boxes.
4. To activate the snap grid, select the **Active (when free snap is on)** check box.
5. Click **OK**.

Now when you pick points using the  **Snap to any position** snap switch, you can only snap to positions at set intervals. The snap grid itself is invisible in the model.

Snap settings

Use the **Model snap settings** dialog box to view and modify the snap settings in the model. The **Drawing snap settings** dialog box has the same options for drawings. These settings are user specific.

Option	Description
Symbol	Show or hide the snap symbols. Select the check box to show the snap symbols, and clear the check box to hide them.
Active (when free snap is on)	Select the check box to activate the snap grid (page 94) .
Spacing	Define grid spacing intervals for the snap grid origin. For example, if the spacing of the x coordinate is 500, you are able to snap to positions at intervals of 500 units in the x direction.
Origin	Define offsets for the snap grid origin.
Angle interval	Set the angle interval for the Ortho tool. This setting is used when you snap to orthogonal points (page 82) . For example, if you set the interval to 10 , the Ortho tool will snap to angles at intervals of 10 degrees in the model or drawing.
Custom angles	Define custom angles for the Ortho tool. This setting is used when you snap to orthogonal points (page 82) . Separate the values with empty spaces. For example, if you enter 12.5 60, the Ortho tool will snap to the angles 12.5 and 60 in the model or drawing.

See also

[Snapping toolbar \(page 78\)](#)

[Snap switches and symbols \(page 79\)](#)

1.4 Work with model objects in Tekla Structures

When you model in Tekla Structures, you create and work with different types of model objects. In most cases, a model object represents a building object that will exist in the real building or structure, or that will be closely related to it. A model object can also be a modeling aid that represents information that is only relevant when you are creating the model. Model objects are either created in the model or imported to it.

Examples of model objects include:

- [Parts \(page 235\)](#) and items
- Bolts and [welds \(page 346\)](#)
- [Reinforcement \(page 427\)](#) and embeds
- [Surface treatment \(page 373\)](#) and [surfaces \(page 386\)](#)
- Cuts, [fittings \(page 361\)](#), holes, and chamfers
- [Pour breaks \(page 415\)](#)
- Loads

Model objects may also be created by components.

You can modify the properties of many model objects by using the [property pane \(page 97\)](#). To resize and reshape model objects, use the [direct modification \(page 103\)](#) handles.

The following modeling aids in can be used in the model:

- [Grids \(page 21\)](#) and [grid lines \(page 26\)](#)
- [Construction objects \(page 561\)](#) and points
- Reference models

You can combine model objects to bigger entities by creating [assemblies \(page 386\)](#), [cast units \(page 396\)](#), and [pour units \(page 410\)](#).

You can manage model objects by using [object groups \(page 591\)](#), Organizer, and other planning tools.

See also

[Copy properties from another object \(page 109\)](#)

[Save and load object properties \(page 111\)](#)

[Undo modeling and drawing changes \(page 114\)](#)

Create model objects and modify model object properties by using property pane

There are several ways to create and modify model objects in Tekla Structures.

You can create different types of [model objects \(page 95\)](#), such as parts and items, bolts, reinforcement, and cuts, by using the commands on the ribbon.

Some ribbon commands have a keyboard shortcut that speed up your modeling work. You can customize the keyboard shortcuts and assign your own shortcuts for the most used commands.

Alternatively, you can start many of the commands that create model objects by using Quick Launch, or in the *property pane*.

After you have created the model objects, you can view and modify the model object properties by using the property pane.

Create or delete a model object

1. Run a command that creates a model object, such as a part.
 - On the ribbon: click a command. For example, click  to create a steel beam.
 - By using **Quick Launch**: enter a search term. For example, type `steel beam` to find the **Create steel beam** command.
 - In the property pane: ensure that you have nothing selected in the model. Click the **Object type list** button  and select from the list the object you want to create.
2. [Pick points \(page 77\)](#) to place the model object in the model. Tekla Structures creates the model object using the current properties of the object type.
3. Follow the status bar messages to get instructions on how to proceed.
4. To create more model objects with the same properties, pick more points. The command runs until you end it or start another command.
5. If you want to delete a model object, select the object and press **Delete**.

Modify model object properties by using property pane

Tekla Structures shows the properties of model objects in the property pane, which is a side pane window. Note that you can have only one property side pane window open at a time. This means that you can view the properties of only one object type at a time.

Use the property pane to view and modify the properties of

- [parts \(page 235\)](#), such as columns and beams
- items
- bolts
- [welds \(page 346\)](#)
- [reinforcement \(page 427\)](#)
- rebar sets
- [pour objects \(page 408\)](#), pour units and [pour breaks \(page 415\)](#)

- chamfers
- [grids \(page 21\)](#)
- [polygon cuts \(page 363\)](#) and [part cuts \(page 364\)](#)
- [construction objects \(page 561\)](#) and points
- [assemblies \(page 386\)](#) and [cast units \(page 396\)](#)
- surface treatment
- surfaces

To open the model object properties:

- If the property pane is closed: double-click a model object, or click the

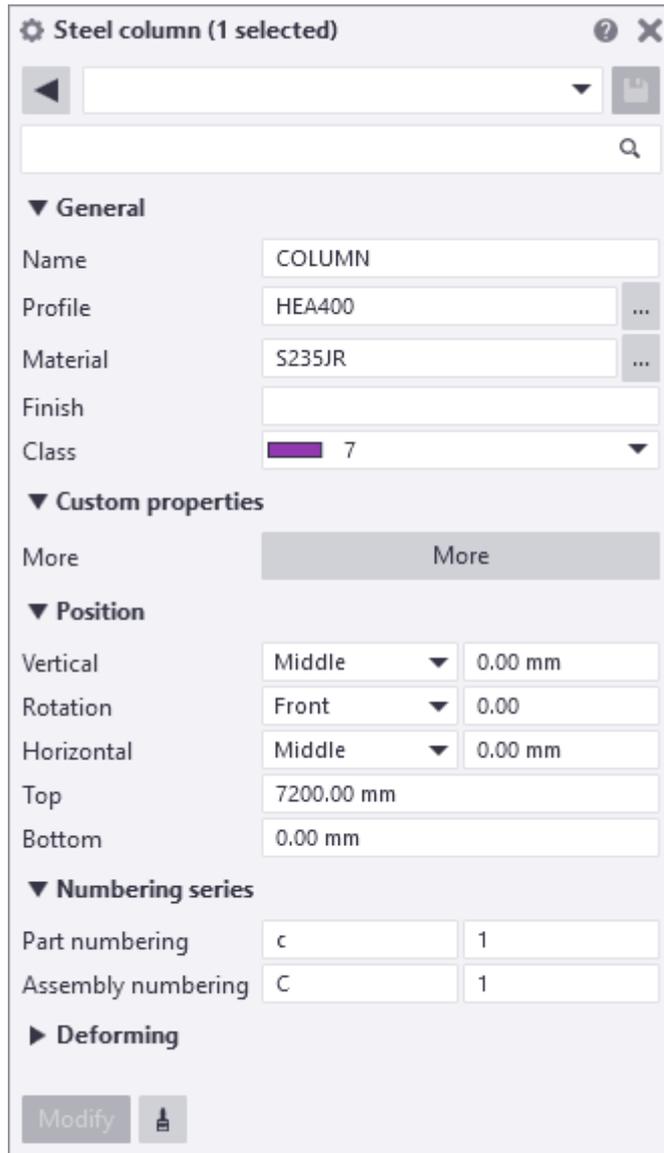
Properties button  in the side pane.

- If the property pane is open: select a model object.

Alternatively, hold down **Shift** and click a command on the ribbon to open the properties in the property pane.

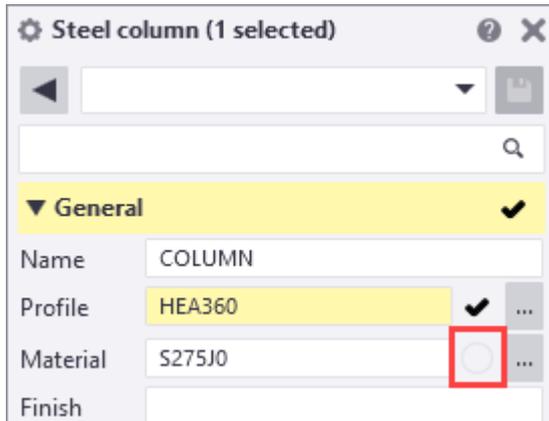
TIP You can customize the property pane. You can select separately for each object type which properties you wish to see in the property pane. For example, you can organize the settings as you wish, or remove the settings you do not need. You can also add your most needed user-defined attributes (UDAs) directly to the property pane.

1. To start modifying the properties, double-click a model object.
The property pane opens and shows the current properties of the object.
For example:

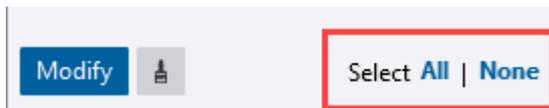


2. Modify the properties as needed.
Tekla Structures highlights the modified properties in yellow in the property pane.
3. If you want to discard some of the changes, click the check marks next to each setting to remove them.

You can clear the check marks one by one, or select a whole section and all its properties.



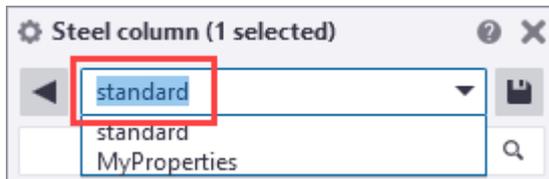
You can use the **Select All** and **Select None** switches at the bottom of the property pane to select all the changes or to clear all the changes.



4. When you are ready with the modifications, click **Modify** to apply the changes.

Tekla Structures uses the properties the next time you create an object of the same type.

If you want to create an object using the standard values instead of the values you just applied, load the standard file first.

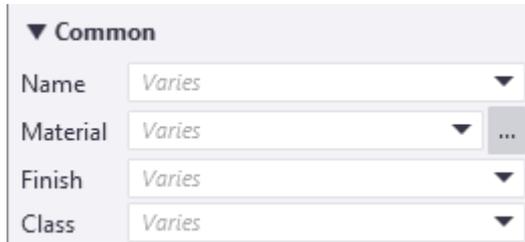


Note that if you use contextual toolbar or [direct modification \(page 103\)](#) to modify a model object, the current properties do not change and are not automatically applied when you create the next object of the same type.

Modify the common properties of different object types by using property pane

When you select several similar objects in the model, the property pane shows the properties that are common for all the selected objects. The settings which have several options have the text **Varies** in the property pane, and the values

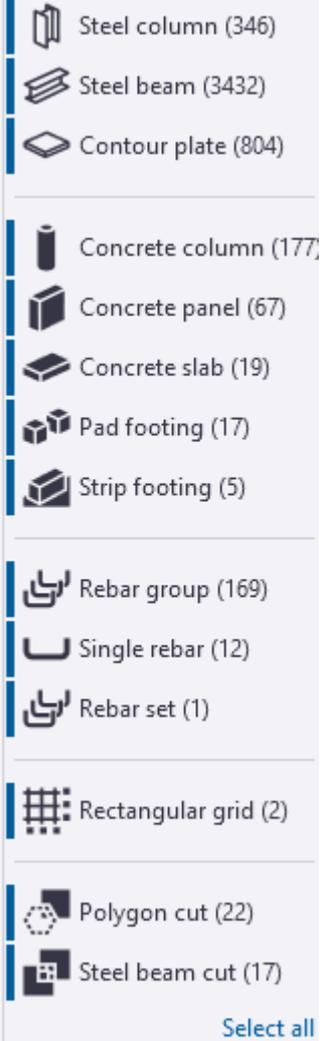
or options are shown in a list. If there are no common properties, the property pane is empty.



You can modify the common properties in the same way as any other property. Tekla Structures highlights the modified properties in yellow in the property pane, and these properties will be applied when you click **Modify**.

Use the **Object type list** in the property pane to check which objects you have selected in the model and the number of each object type.

To	Do this
Check which objects you have selected in the model	Click the Object type list button  to open the list of the selected objects.

To	Do this
	 <p>The list shows how many objects you have selected for each object type.</p> <p>The list also shows how many components you have selected. If you select Component in the list, the property pane shows the names and numbers of the components that you have selected in the model.</p>
<p>Modify the object selection in the Object type list</p>	<p>Hold down Ctrl and click the object types in the list that you want to exclude or include in the selection.</p> <p>The content of the property pane can change according to your selection.</p>
<p>Select all the objects in the Object type list</p>	<p>Click the Select all button.</p>

Resize and reshape model objects

You can resize, reshape, and move model objects by using direct modification handles. When you select an object, Tekla Structures displays the handles and dimensions that are specific for that model object.

Direct modification can be used with the following object types:

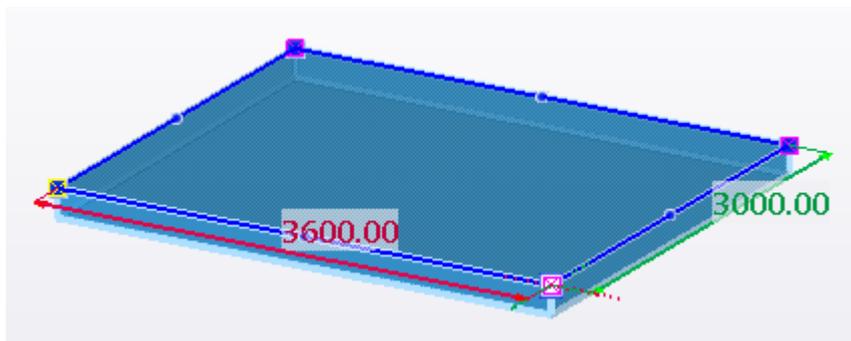
- Parts
- Construction objects
- Grids and grid lines
- Line cuts and polygon cuts
- Reinforcement
- Rebar set guidelines, modifiers, and leg faces
- Pour breaks
- Custom parts
- Loads

1. Ensure that **Direct modification** is switched on.

To switch direct modification on or off, click  or press **D**.

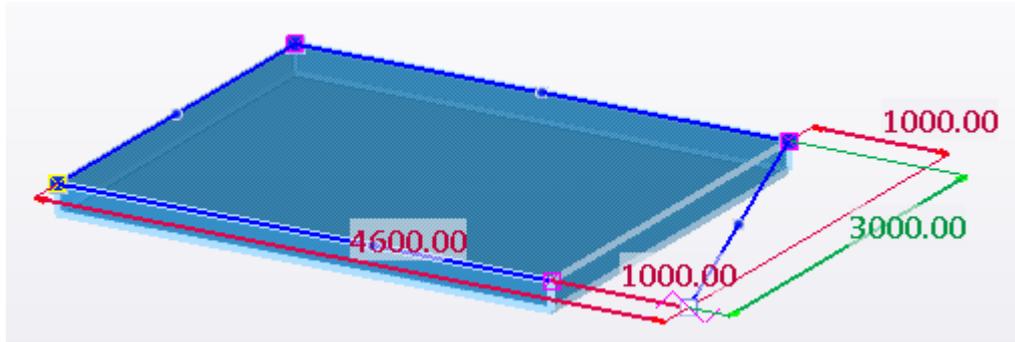
2. Click the object to select it.

Tekla Structures displays the handles that you can use to modify the object.

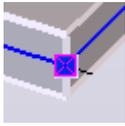
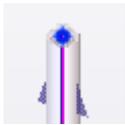
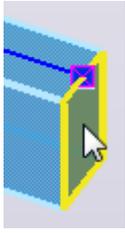
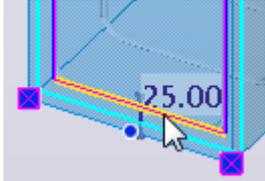


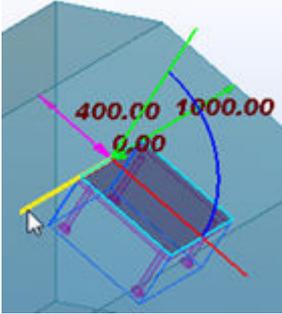
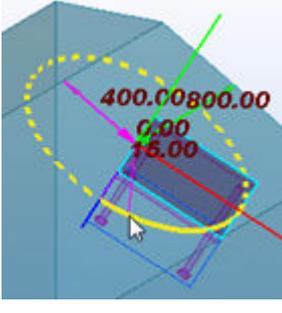
The relevant dimensions are shown when you move the mouse pointer slowly over the object's edges. The dimension colors follow the colors of the work plane coordinate axes: red in the X direction, green in the Y direction, and blue in the Z direction. Diagonal dimensions are magenta.

3. To reshape the object, drag any of the handles.



Here are some examples of direct modification handles:

Handle	Description
	Reference point handle
	Midpoint handle
	End point handle (for reinforcing bars only)
	Plane handle
	Line handle

Handle	Description
	<p>Axis handle (for items (page 298) and custom parts only)</p>
	<p>Rotation handle (for items and custom parts only)</p>

TIP You can use the [snap switches \(page 79\)](#) when dragging a handle. To temporarily disable the snap switches, hold down the **Shift** key when dragging a handle.

4. To give an exact value for a dimension, modify the dimension value.
 - a. Click a dimension to select it.

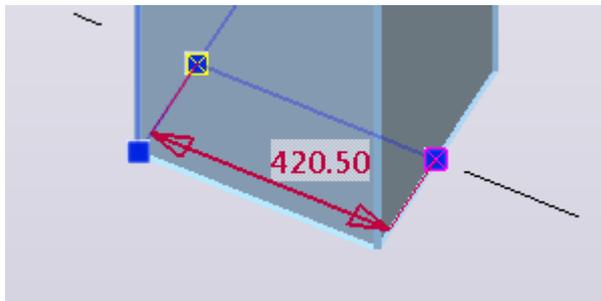


- b. Type a new value.



The yellow arrow defines the direction in which the object is extended or shortened. You can change the direction by clicking the arrows.

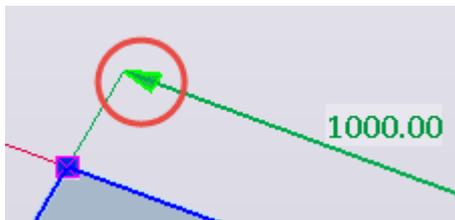
- c. Press **Enter** to confirm the new value.



5. To change a dimension only from one end, move the dimension arrowheads.

You can either drag an arrowhead to a new location, or you can type an exact distance or coordinates.

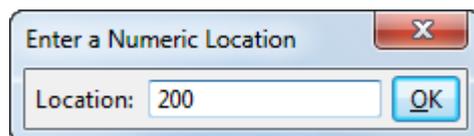
- a. Select the dimension arrowhead you want to move. For example:



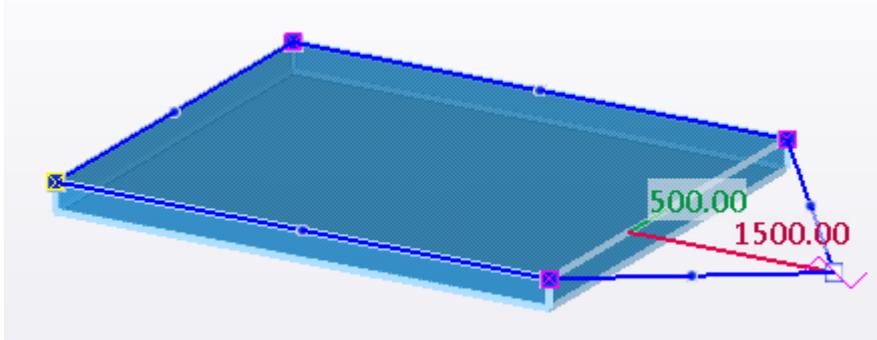
To change a dimension at both ends, select both arrowheads.

- b. Type the distance or coordinates.

When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box. Click **OK** to confirm the dimension.

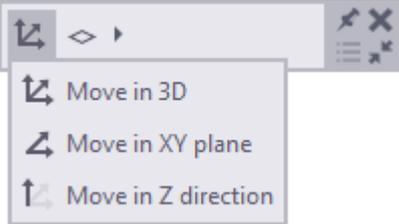
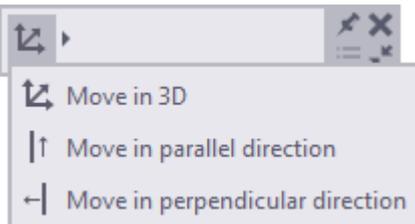
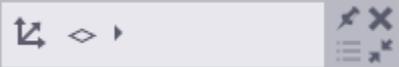


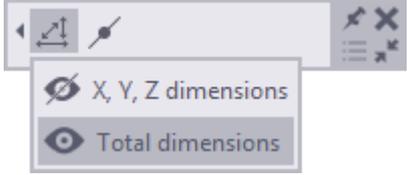
6. To add a new object corner, drag a midpoint handle . For example:



7. To display more modification options, select a handle.

A contextual toolbar appears with more options. The availability of the options depends on the object and handle you have selected.

Click this button	To do this	Location
	Move a handle to any location in the 3D space.	
	Move a handle in the XY plane only.	
	Move a handle in the Z direction only.	
	Move a handle in the parallel direction only.	
	Move a handle in the perpendicular direction only.	
	Move a handle parallel to a certain plane only. Select the plane and drag the	

Click this button	To do this	Location
	<p>handle to a new location.</p> <p>This option can be useful when working with a sloped roof, for example.</p>	
	<p>Control the visibility of direct modification dimensions. Click the eye symbol to show or hide dimensions.</p> <ul style="list-style-type: none"> • X, Y, Z dimensions: All orthogonal dimensions in the work plane directions X, Y, and Z are displayed. • Total dimensions: Only the total length is displayed. 	
	<p>Show or hide midpoint handles.</p>	
	<p>Add a new point at the end of an object.</p> <p>Only available for objects that pass through multiple points, such as polybeams, panels, strip footings, and rebar set modifiers.</p>	

NOTE Some of these options are located in an expandable section on the contextual toolbar. Click the small triangle symbol on the contextual toolbar to show or hide the options:



8. To delete a handle, select it and press **Delete**.

See also

[Modify a construction object \(page 567\)](#)

[Modify a single grid line \(page 26\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

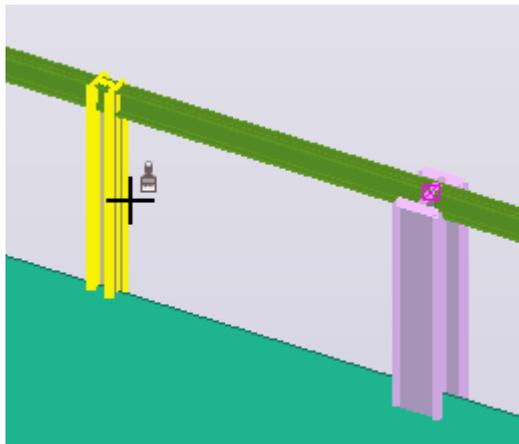
[Modify a rebar set \(page 478\)](#)

[Modify a pour break \(page 421\)](#)

[Add or move a custom part in the model \(page 860\)](#)

Copy properties from another object

Use the  **Copy properties** command in the property pane to copy model object properties from one model object to another. Alternatively, you can use the contextual toolbar if you want to quickly copy properties to a few objects only.



Copy model object properties using the property pane

You can use the property pane to copy properties between any objects as long as both of the objects have the same property available. Use this method when you need to copy properties to a large number of objects.

1. Select the object you want to copy properties from.

2. Click  **Copy properties** in the property pane.

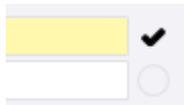
The mouse pointer changes into a paintbrush, and you can select in the property pane which properties you want to copy.

Use the **Select All** and **Select None** switches at the bottom of the property pane to select all the properties or to clear all the selections. If you use the **Select None** switch, all check marks will be cleared the next time you start the **Copy properties** command.

3. Select the objects you want to copy properties to.

You can use area selection to quickly select a large number of objects.

4. In the property pane, Tekla Structures highlights the modified properties in yellow. Select or clear the check marks to indicate which properties you want to copy.



You can clear the check marks one by one, or select a whole section and all its properties, or use the **Select All** and **Select None** switches at the bottom of the property pane to select all the changes or to clear all the changes.

5. Click **Modify** to apply the selected changes.

Tekla Structures uses the properties the next time you create an object of the same type.

When the properties have been copied, the mouse pointer returns to normal.

Note that if you have customized the property pane and added user-defined attributes (UDAs) to it, the values of the UDAs are copied when you use the **Copy properties** command in the property pane. Values of the UDAs that are visible only in the UDA dialog boxes are not copied with the object. Unique user-defined attributes (`unique_attribute`) are not copied with the object.

TIP To copy properties to multiple objects, double-click the  **Copy properties** button to keep the **Copy properties** command running. Click **Modify** after each selected object. The mouse pointer remains in the paintbrush mode until you press **Esc** or click  again.

Copy object properties using the contextual toolbar

Use this method when you want to quickly copy properties to a few objects only.

1. Select the object you want to copy properties from.

A contextual toolbar appears.

2. Click  **Copy properties** on the contextual toolbar.

The mouse pointer changes into a paintbrush.

3. Select the object you want to copy properties to.

When the properties have been copied, the mouse pointer returns to normal.

Note that user-defined attributes (UDAs) are not copied with the object, even if you have customized the contextual toolbar and added UDAs to it.

4. To copy properties to multiple objects, double-click the  **Copy properties** button.

Now you can copy properties to multiple objects. The pointer remains in the paintbrush mode until you press **Esc** or start another command.

Save and load object properties

The property pane and many dialog boxes have the possibility to save the property information in *property files*. You can load these saved properties later on when you create new objects.

For example, you can define the properties for each object type you plan to create before you start modeling, and then use these user-defined property files when you are creating new objects. Tekla Structures stores the user-defined property files, including the properties of sub-dialog boxes, in the current model's `\attributes` folder.

You can save and load object properties either in the property pane or in a dialog box, depending on the object type.

Save and load properties in the property pane

Use the property pane to save and load the properties of

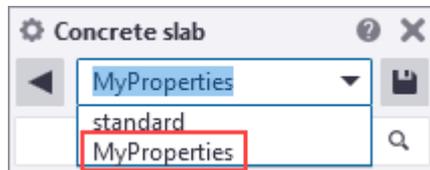
- [parts \(page 235\)](#), such as columns and beams
- items
- bolts
- [welds \(page 346\)](#)

- [reinforcement \(page 427\)](#)
 - rebar sets
 - [pour objects \(page 408\)](#), pour units and [pour breaks \(page 415\)](#)
 - chamfers
 - [grids \(page 21\)](#)
 - [polygon cuts \(page 363\)](#) and [part cuts \(page 364\)](#)
 - [construction objects \(page 561\)](#) and points
 - [assemblies \(page 386\)](#) and [cast units \(page 396\)](#)
 - surface treatment
 - surfaces
1. Click a model object to view the current properties in the property pane.
 2. In the property pane, [modify or enter the properties \(page 96\)](#) you want to save.
 3. In the box next to the  button, enter a name for the property file in which the properties are saved.

For example, `MyProperties`.

4. Click  to save the properties in the property file.

The property file is now saved in the current model folder:



5. When you want to load the saved properties, select the property file from the list.
 - If you select an object in the model and load a property file in the property pane, the properties and the user-defined attributes (UDAs) are loaded immediately. Changed values are highlighted in yellow in the property pane. Click **Modify** to apply the new values.
If there are changes only in the UDA values in the UDA dialog box and not in the values in the property pane, the **Modify** button in the property pane does not become active. Click **Modify** in the UDA dialog box to activate the UDA values.
 - If you start an object creation command in the model and load a property file in the property pane, the properties and the user-defined attributes (UDAs) are used immediately and Tekla Structures creates the object using the loaded values.

NOTE If the UDA dialog box is open when you load a property file in the property pane, the UDA dialog box updates and shows the values of the UDAs that are saved in the property file. However, if you first select an object and load a property file and only then open the UDA dialog box, the dialog box shows the UDA values of the selected object.

To check which UDA values are saved in a property file, you need to open the UDA dialog box first, or load the property file again after opening the UDA dialog box.

6. If you want to make changes to an existing property file:
 - a. Load the property file you want to change.
 - b. Modify the properties.
 - c. Click .

Tekla Structures saves the changes in the property file shown in the list, overwriting the old property file.

Tekla Structures uses the new properties the next time you create an object of the same type.

If you want to create an object using the standard values instead of the saved properties, load the standard file.

Save and load properties in a dialog box

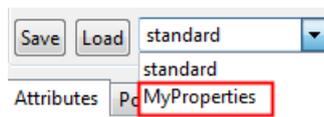
Use this method with properties that are displayed in a traditional dialog box. For example, with drawing object properties.

1. Open the properties dialog box.
2. In the dialog box, modify or enter the properties you want to save.
3. In the box next to the **Save as** button, enter a name for the property file in which the properties are saved.

For example, `MyProperties`.

4. Click **Save as** to save the properties in the property file.

The property file is now saved in the current model folder:



5. When you want to load the saved properties, select the property file from the list, and click **Load**.
6. If you want to make changes to an existing property file:

- a. Load the property file you want to change.
- b. Modify the properties.
- c. Click **Save**.

Tekla Structures saves the changes in the property file shown in the list, overwriting the old property file.

Remove existing properties

You can remove user-defined property files manually by removing them from the model's `\attributes` folder.

1. Remove the selected property file from the model's `\attributes` folder.
The property files may have different file name extensions, depending on their type. For more information, see Model folder files and file name extensions.
2. Restart Tekla Structures.

Undo modeling and drawing changes

The **Undo history** list helps you to check which commands and modifications you have done and undone, and to keep track of the changes. By selecting a command or an action in the **Undo history** dialog box, you can undo or redo several commands at one go, and thus go back and forth in the model history. **Undo history** is available both in the modeling and in the drawing mode.

How to use Undo history

To open the **Undo history** dialog box, click the  button on the Quick Access Toolbar, next to the **Undo** and **Redo** buttons. Alternatively, use **Quick Launch** to open the dialog box.

To	Do this
Undo commands	<p>Click any row on the list.</p> <p>If the undo operation takes a long time, Tekla Structures highlights the clicked row to show what was selected.</p> <p>All the modifications you have done after the selected command are undone. The modifications you have undone have a dark gray background color in the list.</p>

To	Do this
Redo previously undone commands	<p>Click any row with a dark gray background color in the list.</p> <p>All the modifications you have done before the selected command are redone.</p>
Add a bookmark	<p>Move your mouse pointer over any row. A bookmark icon  is shown. Click the bookmark icon to mark certain commands.</p> <p>Use the bookmarks to mark important commands or actions. You can later return to these commands or actions if the changes in the model are not satisfactory.</p> <p>To remove a bookmark, click the bookmark icon  again.</p>

The **Undo history** dialog box

- lists the commands you have run and modifications you have done, starting from the top of the list. The latest commands and modifications are at the bottom of the list.
- updates constantly according to the changes you make in the model or in the drawing.
- creates a hierarchy for some of the used commands. The hierarchy is created when you first run a command, then you undo some commands to a certain point in the list, and run another command.

The hierarchy is marked with an arrow in the list. You can undo or redo commands at any point in the hierarchy, making it possible to undo commands that you have previously redone.

This means that after you have undone a command, you can continue working with the model, and you still have the option to go back to the commands you have used first.

The **Undo history** list is cleared when you

- save a model
- open or close a drawing
- synchronize Organizer with the model
- read in or write out model changes using Tekla Model Sharing
- use CIS/2 or SDNF import commands.

1.5 Select objects

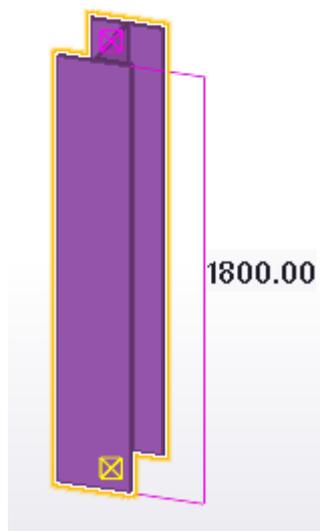
Many Tekla Structures commands require to select objects. You can make single selections and area selections. Tekla Structures highlights the selected objects. The number of selected objects and handles is displayed in the bottom right corner of the status bar. For example: 1 + 1 object(s) selected

Use the different commands and methods to select the objects. To control which type of objects can be selected, use [the selecting toolbar and the selection switches \(page 122\)](#).

Select single objects

1. Ensure that the correct [selection switches \(page 122\)](#) are active.
2. Click an object to select it.

Tekla Structures displays dimensions and dimension lines for column, beam, reinforcing bar group, and for rebar set. If you want to hide the dimensions, use the advanced options XS_DISPLAY_DIMENSIONS_WHEN_SELECTING_OBJECTS and .

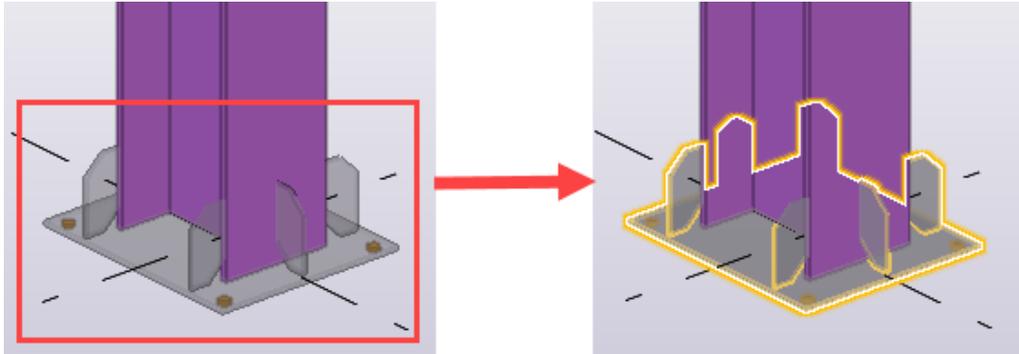


NOTE When [direct modification \(page 103\)](#)  is switched on, the object dimensions and dimension lines are always hidden. Only the direct modification dimensions are then shown. This makes it easier to know which dimensions can be edited.

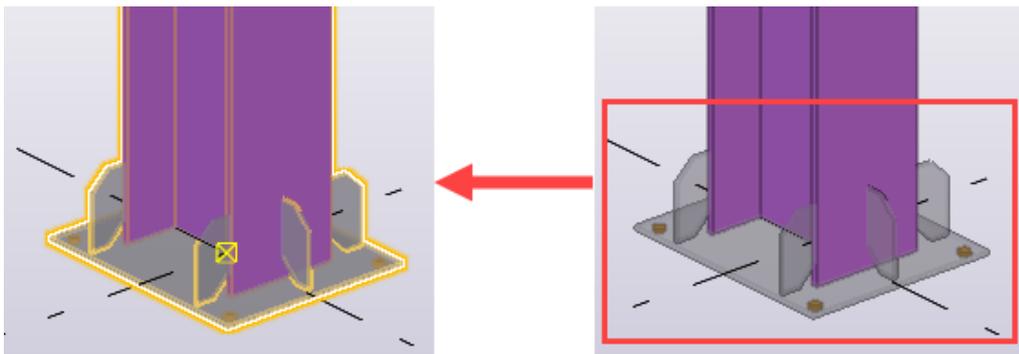
Select multiple objects using area selection

You can select multiple objects using area selection. By default, the dragging direction affects the selection of objects.

1. Ensure that the correct [selection switches](#) (page 122) are active.
2. To select all objects that are completely within a rectangular area, hold down the left mouse button and drag the mouse from **left to right**.



3. To select all objects that are at least partly within a rectangular area, hold down the left mouse button and drag the mouse from **right to left**.

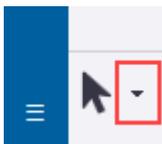


4. If you want to change how area selection works, click **File menu** --> **Settings** and select or clear the **Crossing selection** check box. By default, the option is switched off. When the option is **off**, the dragging direction affects the selection of objects. When the option is **on**, all objects that fall at least partially inside the rectangular area are selected, regardless of the dragging direction.

Select all objects

To select all objects at once, do one of the following:

- On the ribbon, click the small down arrow next to the arrow button



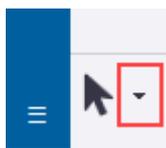
, and then click **Select all objects**.

- Press **Ctrl+A**.

Select previous objects

Sometimes you need to select again the same objects that you selected previously but have then deselected. To select previously selected objects, do one of the following:

- On the ribbon, click the small down arrow next to the arrow button



, and then click **Select previous objects**.

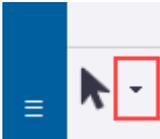
- Press **Alt+P**.

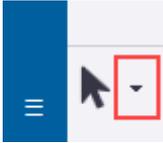
Select objects by identifier

If you know the GUID (globally unique identifier), or the ID of an object, or the IFC GUID of a reference object, you can use the **Select by identifier** command to locate the objects in a model or in a drawing.

You can often find information about the object GUID or ID, for example, in reports and log files. By using the **Select by identifier** command you can quickly find the objects in a model or in a drawing, instead of defining a view filter or a selection filter with the specific GUID or ID. You can use IFC GUIDs to find IFC reference objects. This is useful if you need to track updates and changes in IFC reference models.

Additionally, you can use the **Select by identifier** command to inquire the GUIDs of selected objects, instead of using the traditional [inquiry \(page 599\)](#).

To	Do this
Find objects based on the object GUID, ID, or IFC GUID identifier	<ol style="list-style-type: none"> <ul style="list-style-type: none"> • In the modeling mode: on the ribbon, click the small down arrow next to the arrow button , and then click Select by identifier. • In the drawing mode: in Quick Launch, type Select by identifier. <p>The Select by identifier dialog box opens.</p>

To	Do this
	<ol style="list-style-type: none"> 2. Copy the object identifier, for example from a log file, to the dialog box. You can enter multiple identifiers in the dialog box. Either enter each identifier on its own row, or separate them with semicolon ; . 3. To define the search, select the needed check boxes. <ul style="list-style-type: none"> • Reference objects: Tekla Structures selects IFC objects based on their GUID or IFC GUID. • Keep selection: Tekla Structures keeps the currently selected object and appends it with new selection. • Zoom to selected: Tekla Structures selects the object and zooms to it. 4. Click Select. Tekla Structures selects the objects based on the GUID in the model or in the drawing. If there are identifiers that are not found in the model or in the drawing, they are listed in the status bar as <code>identifier?</code>.
Find a model object in a drawing	<p>You can select an object in a model, get its identifier, and then find it in a drawing based on the identifier.</p> <ol style="list-style-type: none"> 1. In the modeling mode: on the ribbon, click the small down arrow next to the arrow button  , and then click Select by identifier. The Select by identifier dialog box opens. 2. Select an object or objects in the model. 3. Click Get. The Select by identifier dialog box lists the identifiers of the selected objects. If you want to get IFC GUIDs, ensure that the Reference objects check box is selected. 4. Keep the dialog box open. 5. Open a drawing.

To	Do this
	<p>6. In the drawing mode, click Select to find the objects in the drawing.</p> <p>You can then continue working with the found objects.</p>
Find a drawing object in a model	<p>You can select an object in a drawing, get its identifier, and then find it in a model based on the identifier.</p> <ol style="list-style-type: none"> 1. In the drawing mode: in Quick Launch, type Select by identifier. The Select by identifier dialog box opens. 2. Select an object or objects in the drawing. 3. Click Get. The Select by identifier dialog box lists the identifiers of the selected objects. 4. Keep the dialog box open. 5. Close the drawing. 6. In the modeling mode, click Select to find the objects in the model. <p>You can then continue working with the found objects.</p>

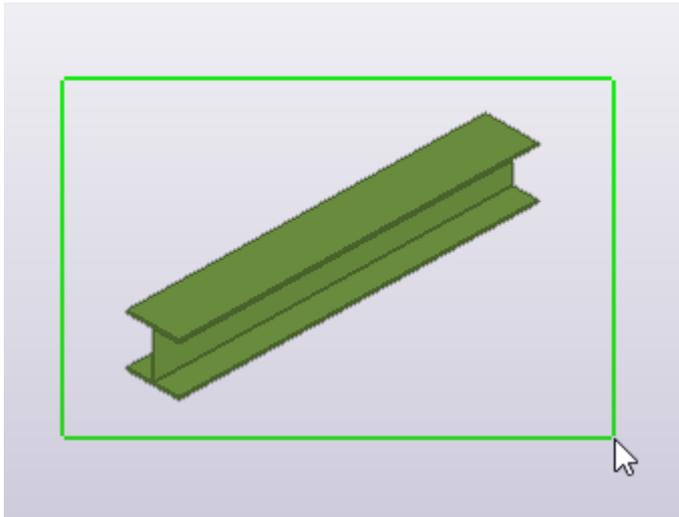
Select handles

Sometimes you need to select only the handles of a part, for example when moving the part.

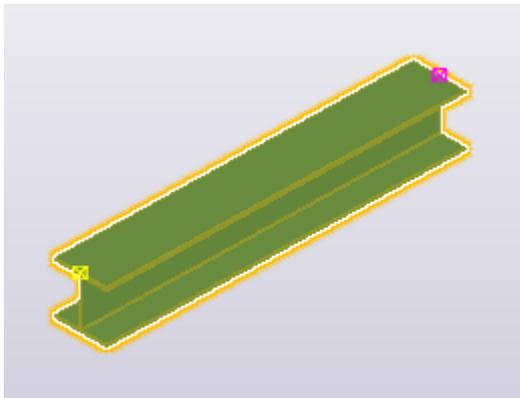
Before you start, make sure that **Crossing selection** is switched off. Ensure

that the **Direct modification** switch  is not active.

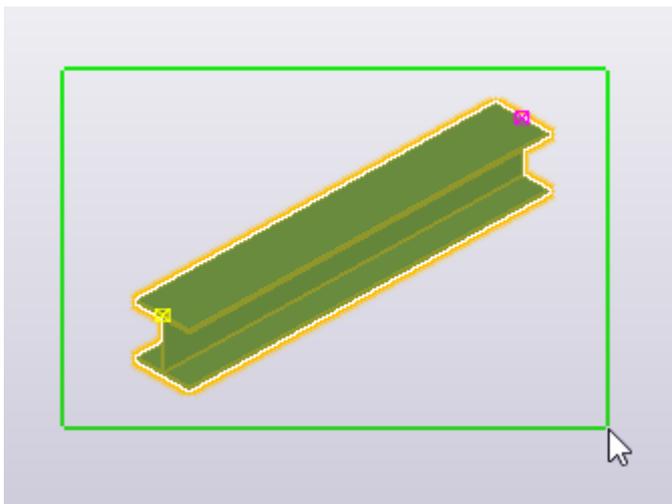
1. On the **File** menu, click **Settings** and make sure that **Crossing selection** is switched off. If the **Crossing selection** is not switched off, selecting the handles with the **Alt** key does not work.
2. Ensure that the correct [selection switches \(page 122\)](#) are active.
3. Hold down the left mouse button and drag the mouse from left to right to include the entire part.



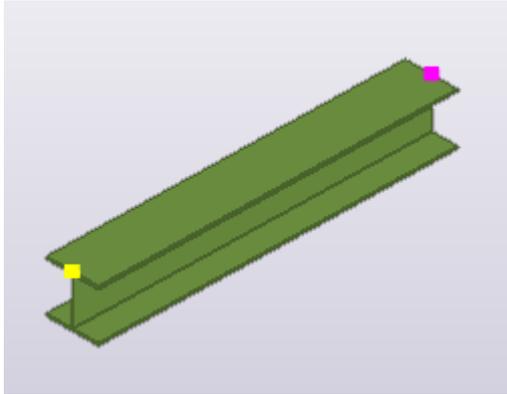
The part becomes selected:



4. Hold down the **Alt** key and drag the mouse from left to right again.



Now only the part handles are selected:



NOTE When [direct modification \(page 103\)](#)  is switched on, Tekla Structures also displays the direct modification handles for reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.

Modify the selection

You can add objects to the current selection, or remove objects from the selection.

1. To add objects to the current selection, press the **Shift** key and select more objects.
2. To switch the selection of an object on or off, press the **Ctrl** key during the selection. Tekla Structures deselects the objects that were already selected and selects those that were previously not selected.
3. To clear the selection of all objects and handles, click somewhere else. For example, click on the empty background of the current view.

Selecting toolbar

The *selection switches* on the **Selecting** toolbar are special commands that control which objects and object types you can select. For example, if you select the entire model area but only the **Select parts** switch is active, only the parts become selected.

Click the selection switches on the **Selecting** toolbar to switch them on or off.



By default, the **Selecting** toolbar is located at the bottom of the screen. If you are unable to find the toolbar, click **File --> Settings**, and in the **Toolbars** list ensure that the **Selecting** toolbar is selected.

Main selection switches

The main selection switches control whether you can select components and assemblies, or objects included in them. These switches have the highest priority.

Switch	Selectable objects	Description
	Components	When you click any object belonging to a component, Tekla Structures selects the component symbol and highlights (but does not select) all component objects.
	Component objects	Objects created automatically by a component can be selected.
	Assemblies and cast units	When you click any object in an assembly or a cast unit, Tekla Structures selects the assembly or cast unit and highlights all objects in the same assembly or cast unit.
	Objects in assemblies and cast units	You can select single objects in assemblies and cast units.

Other selection switches

The table below lists the remaining selection switches. Use these switches to control which object types you want to select.

Switch	Selectable objects	Description
	Any objects	Turns all switches on. You can select all object types, except for single bolts.
	Components	You can select component symbols.
	Parts	You can select parts, such as columns, beams, or plates.

Switch	Selectable objects	Description
	Surface treatments and surfaces	You can select surface treatments and surfaces.
	Points	You can select points.
	Construction lines and circles	You can select construction lines and circles.
	Reference models	You can select entire reference models. This selection switch may affect the speed of zooming and rotating in the model. For more information, see Tips for large models (page 229) .
	Grids	You can select entire grids by selecting one line in the grid.
	Grid lines	You can select single grid lines.
	Welds	You can select welds.
	Cuts and added material	You can select line, part, and polygon cuts, fittings, and added material.
	Views	You can select model views.
	Bolt group	You can select entire bolt groups by selecting one bolt in the group.
	Single bolts	You can select single bolts.
	Rebar sets	You can select rebar sets, and also reinforcing bar groups and single reinforcing bars.
	Rebar groups	You can select bar groups in rebar sets, and also reinforcing bar groups and single reinforcing bars.

Switch	Selectable objects	Description
	Single rebars	You can select single bars in rebar sets, and also reinforcing bar groups and single reinforcing bars.
	Pour breaks	You can select pour breaks.
	Planes	You can select construction planes.
	Distances	You can select distances that are used in custom components or in parametric modeling.
	Tasks	You can select Task Manager tasks.
		Switch Direct Modification on or off.
		Hide (page 225) selected switches from the toolbar.

Analysis model switches

The following switches can be used to select objects in an analysis model:

Switch	Selectable objects	Description
	Loads	You can select point, line, area, uniform, and temperature loads.
	Analysis parts	You can select analysis parts.
	Nodes	You can select analysis nodes.
	Rigid links	You can select analysis rigid links.

Selection switches in drawings

Similar selection switches are available in drawings:

Switch	Selectable objects	Description
	Any objects	Turns all switches on. You can select all object types, single dimensions

Switch	Selectable objects	Description
		of a dimension set, or single grid lines of a grid.
	Lines	You can select drawing objects such as lines, arcs, circles, rectangles, polylines, polygons, and clouds.
	Text	You can select any text in drawings.
	Marks	You can select all kinds of marks and associative notes in drawings. This selection switch also selects weld marks.
	Parts	You can select parts, such as columns, beams, and plates in drawings.
	Section symbols	You can select section symbols in drawings.
	Welds	You can select welds in drawings. To select weld marks, use the Select drawing marks selection switch.
	Views	You can select drawing views.
	Dimensions	You can select drawing dimensions. You can select an entire group of dimensions by selecting one dimension in the group.
	Single dimensions	You can select single drawing dimensions.
	Grids	You can select grids in drawings.
	Grid lines	You can select single grid lines in drawings.
	Detail marks	You can select details marks in drawings.

Switch	Selectable objects	Description
	Plugins	You can select custom plugins in drawings.

Alternatively, you can control the selection switches with **Quick Launch**. Start typing the name of the selection switch, for example, `select`, and click the name of the selection switch on the search results list to activate the switch.

See also

[Tips for selecting objects \(page 129\)](#)

Select assemblies, cast units, and nested objects

You can select either assemblies or cast units, or single objects in nested assemblies or nested components.

Select assemblies and cast units

Use the **Select assemblies** selection switch to select [assemblies \(page 386\)](#) and [cast units \(page 396\)](#).

1. Ensure that the  **Select assemblies selection switch (page 122)** is active.
2. Select a part.
Tekla Structures selects the entire cast unit or assembly that contains the selected part.

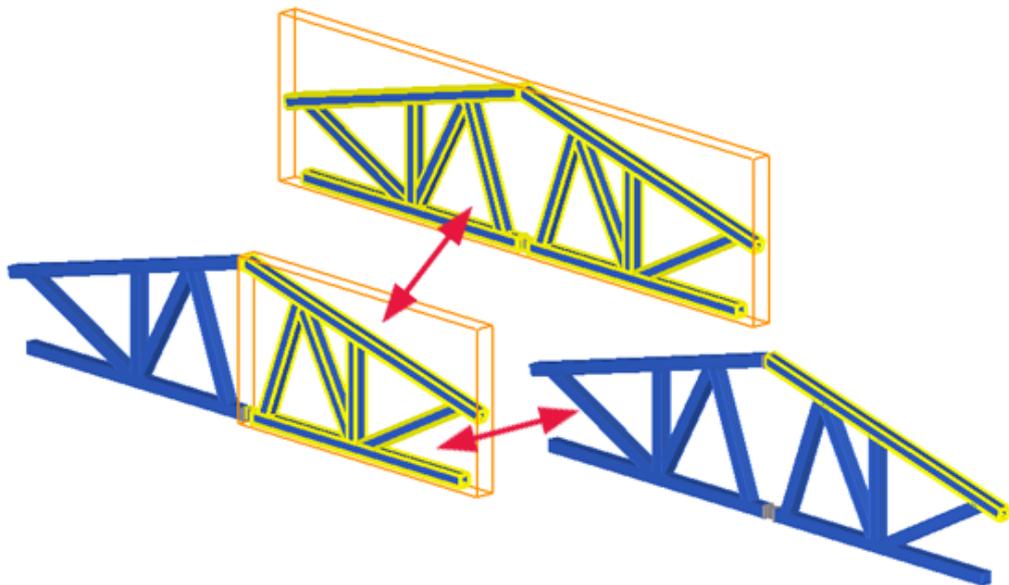
Select nested objects

You can select nested assemblies and components. The active selection switch defines on which level you start and toward which direction you move in the component or [assembly hierarchy \(page 390\)](#). The status bar shows the steps you take in the hierarchy.

1. Ensure that the correct [selection switch \(page 122\)](#) is active.
 -  : to start from the assemblies on the highest level, move to their sub-assemblies, and finally select single parts, bolts, and so on
 -  : to start from single objects and move to bigger and bigger nested assemblies

-  : to start from the components on the highest level, move to their sub-components, and finally select single parts, bolts, and so on
 -  : to start from single objects and move to bigger and bigger nested components
2. Hold down the **Shift** key.
 3. Scroll with the mouse wheel.

The orange highlighting indicates the assembly or component that you can select.



Select reference models, reference model objects and assemblies

You can select either entire reference models, or single objects and assemblies that are part of a reference model. The use of selection switches differs in each case.

Select an entire reference model

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select components** selection switch.
3. Select the reference model.

Select a reference model object

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select objects in components** selection switch.
3. Select the desired object in the reference model.

Select a reference model assembly

1. Activate the  **Select reference models** selection switch.
2. Activate the  **Select assemblies** selection switch.
3. Select the desired assembly in the reference model.

Tips for selecting objects

Here are listed some tips that can help when you are selecting objects.

Switch rollover highlight on or off

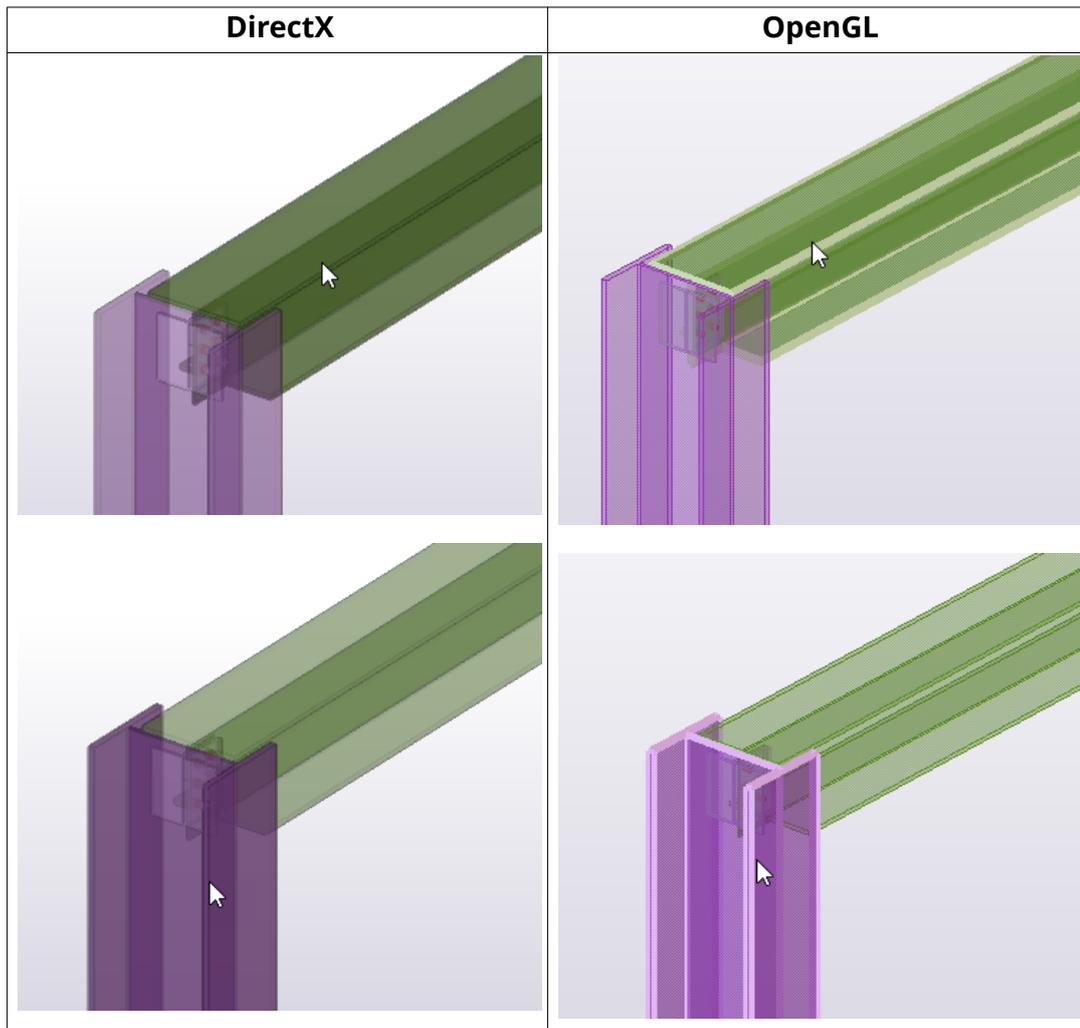
By default, Tekla Structures highlights the objects that you can select. You can switch the highlighting on or off.

To switch the rollover highlight on or off, on the **File** menu, click **Settings** and select or clear the **Rollover highlight** check box. Alternatively, you can press **H**.

Depending on the [rendering engine \(page 67\)](#) you are using, OpenGL or DirectX, Tekla Structures highlights the objects differently when rollover highlight is on.

In the example below, the [rendering of parts \(page 584\)](#) is set to **Parts shaded wireframe**.

DirectX	OpenGL
Tekla Structures highlights the objects by showing them with darker object color. For example:	Tekla Structures highlights the objects by showing them with light edge line color. For example:



Select on right-click

You can change the settings so that you can select objects also with the right mouse button.

1. On the **File** menu, click **Settings** and select the following check boxes:
 - **Select on right-click**
 - **Rollover highlight**
2. Right-click an object to select it. Tekla Structures highlights the object and shows the related shortcut menu.

If you cannot select objects

If you cannot select desired objects in the model, check the selection switches and the filter settings.

- Check that you have switched on all the needed [selection switches](#) (page 122).

- If you still cannot select the objects, check the selection filter settings. You can select a different filter or modify the current filter.

Interrupt object selection

You can have Tekla Structures interrupt the object selection process if the selection takes over a defined period of time. For example, if you are working on a large model and you accidentally select all or part of the model, you can interrupt the selection if it takes over 5000 milliseconds (5 seconds) to complete.

1. Define the time after which Tekla Structures asks if you want to interrupt object selection.
 - a. On the **File** menu, click **Settings** --> **Advanced Options** and go to the **Modeling Properties** category.
 - b. Modify the advanced option XS_OBJECT_SELECTION_CONFIRMATION. The default value is 5000 milliseconds.
 - c. Click **OK**.
2. [Select \(page 116\)](#) all or part of the model.
3. When Tekla Structures asks if you want to interrupt object selection, click **Cancel**.

1.6 Copy and move objects

The basic functionality of copying and moving objects is the same in models and drawings. You can copy and move objects linearly, with rotation, and with mirroring.

- [Copy objects \(page 133\)](#)
- [Move objects \(page 144\)](#)
- [Rotate objects \(page 150\)](#)
- [Mirror objects \(page 154\)](#)

Duplicate objects

Two objects are considered duplicates if they have the same properties and location. Tekla Structures checks for duplicate objects when you copy and move objects or create new objects in the same location as an existing object. If duplicates are found, you can choose whether to keep or delete them.

Use the advanced option XS_DUPLICATE_CHECK_LIMIT_FOR_COPY_AND_MOVE to define the maximum number of objects that can be counted as duplicates while copying or moving objects.

NOTE Tekla Structures does not check for duplicates when you copy objects using a modeling tool, such as the **Array of objects (29)** component.

Connected objects

When you copy objects that are connected to another part (such as connections and bolts), Tekla Structures tries to find suitable new main parts for these copied objects. If none are found, some of the connected objects may not be copied at all. After copying, check that all of the objects have been copied correctly.

Assemblies and cast units

If you copy or move objects from an assembly or cast unit, Tekla Structures copies the assembly structure if possible. For example, sub-assemblies are copied as sub-assemblies if a parent object is found.

Reinforcement and surface treatment

If you copy or move reinforcement or surface treatments, and want them to adapt to the part they are copied or moved to:

- The reinforcement handle or surface treatment handles must be in part corners.
- The parts between which you copy or move must have the same number of cross section corners.
- Circular parts must have the same cross section dimensions.

Drawing objects

You can copy and move drawing objects between drawing views that have different scales.

Copy and move efficiently

You can keep the **Move** and **Copy** dialog boxes open if you are going to use them often.

After you have run the **Copy - Linear**, **Copy - Mirror**, **Copy - Rotate**, or the **Move - Linear**, **Move - Mirror** or **Move - Rotate** command, interrupt the command and leave the dialog box open. When you want to continue copying or moving, click the dialog box to activate it and continue to copy or move objects.

Show or hide "Do not show this message again"

Tekla Structures displays warning messages when necessary, for example, when you are about to copy or move objects outside the work area. However, you have the option to hide future warnings of the same type. You can have Tekla Structures show these warnings again.

- To hide future warnings of the same type, select the **Do not show this message again** check box.
- To re-display the warnings, hold down the **Shift** key while running a command that should normally induce a warning message.

Copy objects

You can copy objects in a number of different ways. When you copy an object, Tekla Structures copies all objects connected to it, including the components.

Copy by picking two points

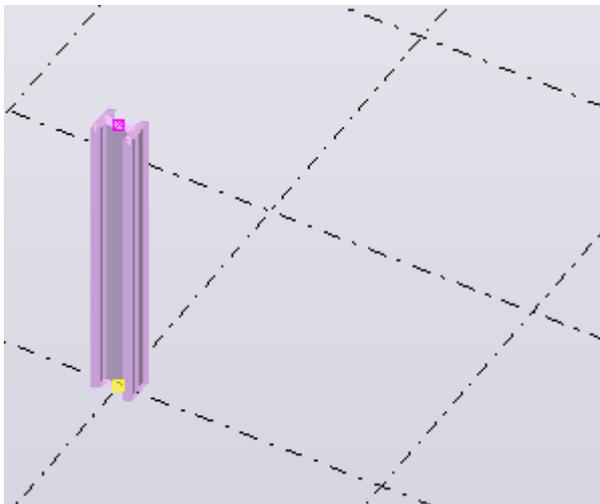
The basic way to copy objects in a model or drawing is by defining the origin and one or more destination points.

1. Select the object you want to copy.
2. Run the **Copy** command:

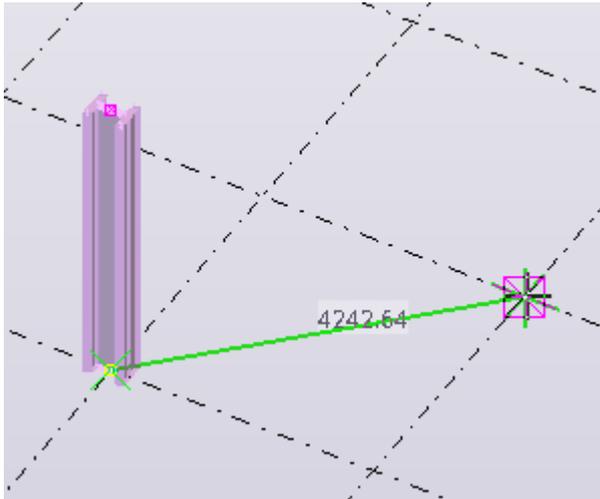
- In the model, on the **Edit** tab, click  **Copy**.

- In the drawing, on the **Drawing** tab, click  **Copy --> Copy**.

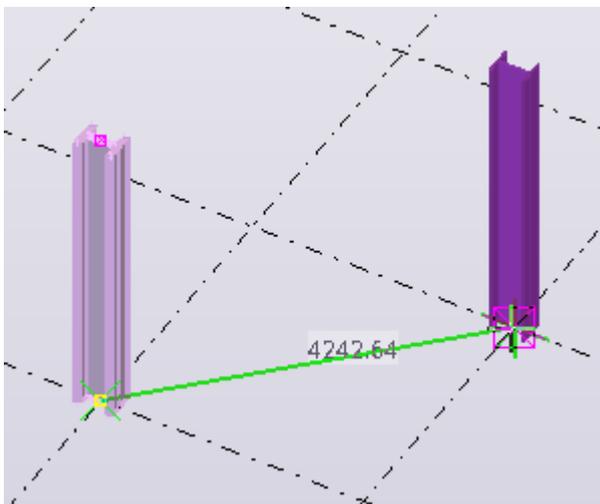
3. Pick the origin for copying.



4. Pick one or more destination points.



The objects are copied immediately. The **Copy** command remains active.



5. If you want to undo the latest copy operation, click the  **Undo** button on the top left corner of the Tekla Structures main window.
The **Copy** command still remains active.
6. To stop copying, press **Esc**.

Copy linearly

In the model, you can create multiple copies of an object in the same linear direction.

1. Select the objects you want to copy.
2. On the **Edit** tab, click  **Copy special --> Linear**.
The **Copy - Linear** dialog box opens.

- Pick two points, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.
You can also use a formula to calculate the x, y, and z displacements. For example:

dY =3*1250

- Enter the number of copies.
- Click **Copy**.
- To stop copying, press **Esc**.

TIP If the dialog box is open but the command is no longer active, click the **Pick...** button to re-activate the command.

Copy by specifying a distance from origin

You can copy objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog box to specify the distance.

- Select the objects you want to copy.
- Run the **Copy** command:

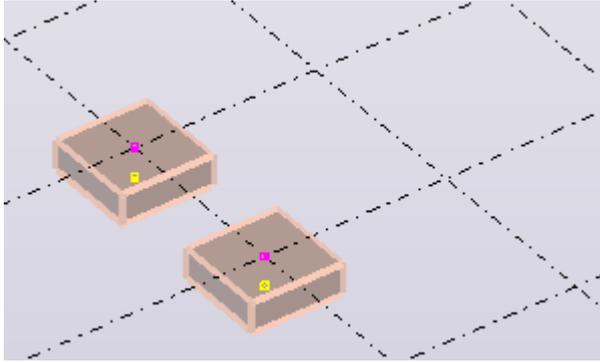
- In the model, on the **Edit** tab, click  **Copy**.
- In the drawing, on the **Drawing** tab, click  **Copy** --> **Copy**.

- Pick the origin for copying.
- Move the cursor in the direction you want to copy the objects, but do not pick the point.
- Type the distance.
When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.
- Click **OK**.

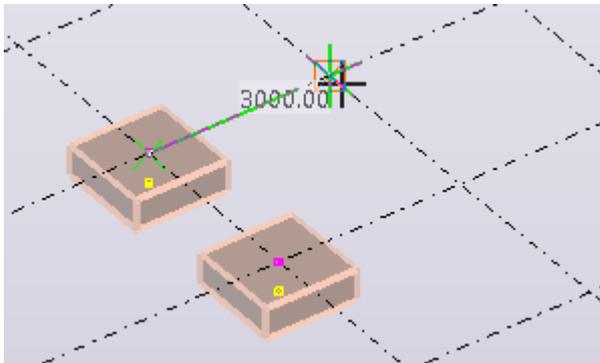
Copy using drag-and-drop

You can copy objects by using drag-and-drop.

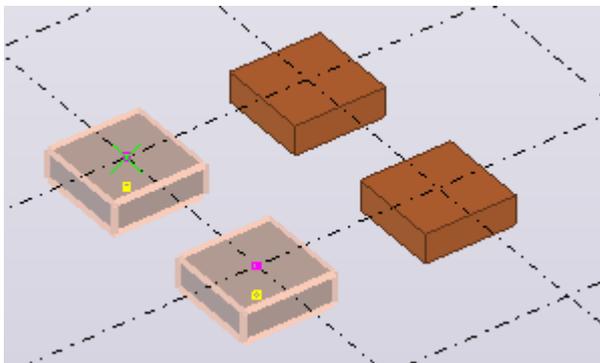
- On the **File** menu, click **Settings** and select the **Drag & drop** check box to activate the command.
- Select the objects you want to copy.



3. Hold down the **Ctrl** key and drag the objects to a new location. The point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.



Tekla Structures copies the objects:



NOTE To copy grid labels in a drawing, first select the grid label and then either activate the  **Select grid line** [selection switch \(page 122\)](#) or select the grid label handle.

Copy objects to another object

In the model, you can copy objects from an object to other similar objects. This is useful, for example, when you detail previously modeled parts. The objects

that you can copy between can have different dimensions, length, and rotation.

1. Select the objects you want to copy.
2. On the **Edit** tab, click  **Copy special --> To another object.**
3. Select the object to copy from (source object).
4. Select the objects to copy to (target object).

Copy all content to another object

In the model, you can copy objects from an assembly or cast unit to other similar assemblies or cast units without individually selecting each object to copy. This is useful, for example, when you have detailed an assembly and want to copy all details to another similar assembly.

1. Ensure that the  **Select assemblies selection switch** ([page 122](#)) is active.
2. Select the assembly or cast unit to copy from (source object).
3. On the **Edit** tab, click  **Copy special --> All content to another object.**
4. Select the assemblies or cast units to copy to (target objects).

As a result, Tekla Structures copies the following objects:

- Secondary parts
- Reinforcement, bolts, and welds
- Cuts, fittings, and edge chamfers
- Sub-assemblies
- Components

NOTE Tekla Structures does not copy pour breaks, or secondary parts created by a component that has also created the assembly main part. If some of the objects to be copied already exist in the assembly or cast unit to copy to, Tekla Structures may create duplicate objects. Tekla Structures warns you about duplicate secondary parts, reinforcement, and sub-assemblies, but not about duplicate bolts, welds, cuts, or components.

Copy to another plane

In the model, you can copy objects from the first plane you specify to the second (and third, etc.) plane you specify. The position of the copied objects

relative to the second (and third, etc.) plane remains the same as the position of the original objects relative to the first plane.

1. Select the objects you want to copy.
2. On the **Edit** tab, click  **Copy special --> To another plane.**
3. Pick the point of origin of the first plane.
4. Pick a point on the first plane in the positive x direction.
5. Pick a point on the first plane in the positive y direction.
6. Repeat steps 3–5 for all destination planes.

Copy from another model

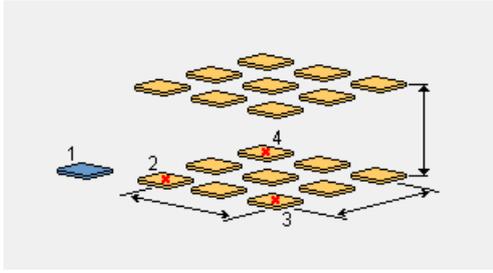
You can copy objects from another model based on phase numbers. Note that Tekla Structures copies secondary parts from the model only if they belong to the same phase as their main part. This also applies to component objects.

1. On the **Edit** tab, click  **Copy special --> From another model.**
The **Copy from Model** dialog box opens.
2. In the **Model directories** list, select the model to copy from.
This is the source model. Note that the target model must have been created using the same or newer version of Tekla Structures as the source model. You cannot copy from a newer version to an older version.
3. In the **Phase numbers** box, enter the numbers of the phases from which to copy objects, separated by spaces.
For example, 2 7.
4. Click **Copy**.
5. Close the dialog box.

NOTE If you copy pour breaks from another model, the copied pour breaks automatically adapt to the target model. Always check that the copied pour breaks have adapted correctly.

Copy objects using linear array tool

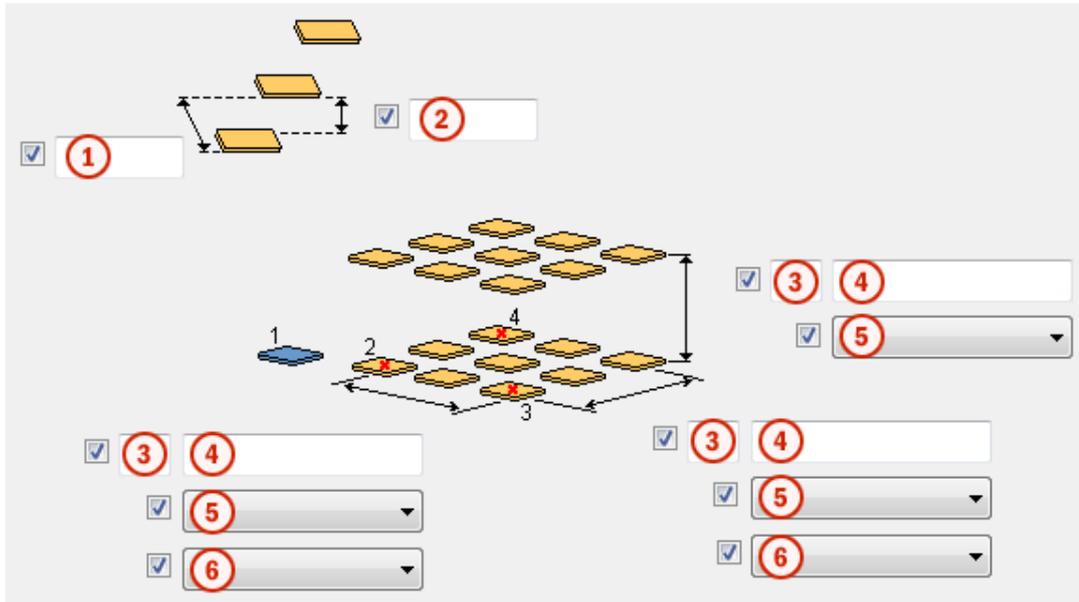
Use **Linear array tool** to copy selected objects linearly along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.



How to use Linear array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
 2. Search for **Linear array tool**, and then double-click to open it.
 3. Select the **Copy method**. The options are:
 - **Selected objects only**
This is the default. Only the selected objects are copied.
 - **All associated objects**
Selected objects and all objects associated with them are copied. For example, cuts and fittings applied to a part.
 - **Advanced**
This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.
 4. Select the **Copy origin**. The options are:
 - **Object to be copied**
This is the default. Copies are relative to the input objects.
 - **Origin point**
Copies are relative to the input origin point.
 5. Define the settings.
 6. Select the objects to copy.
 7. Click **OK** to close the dialog box.
 8. Click the middle mouse button.
 9. Pick origin point.
 10. Pick axis direction X.
 11. Pick axis direction Y.
- The selected objects are copied.

How to define the settings

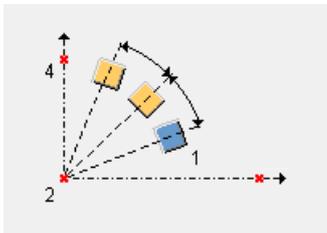


1	Offset along the Y axis. The default value is 0 mm.
2	Offset along the Z axis. The default value is 0 mm.
3	Number of copies. The default value is 0.
4	Space between copies. The default value is 0 mm. Use the space character to separate values. Enter a value for each space between copies. This option is not available if you select Equal as the spacing method.
5	Copy direction. The options are: <ul style="list-style-type: none"> • Normal (default) Spacing values are calculated from the origin in positive direction along the axis. • Reverse Spacing values are calculated from the origin in negative direction along the axis. • Centered Copies are centered on the origin. • Mirror Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.
6	

6	<p>Spacing method. The options are:</p> <ul style="list-style-type: none"> • Equal (default) Copies are equally spaced based on the length of the X or Y axis. • Specified Copies are spaced according to the number and spacing values given.
---	--

Copy objects using radial array tool

Use **Radial array tool** to copy selected objects radially along multiple directions at defined intervals or spacing. Tekla Structures does not check for duplicates when you copy objects using this method.

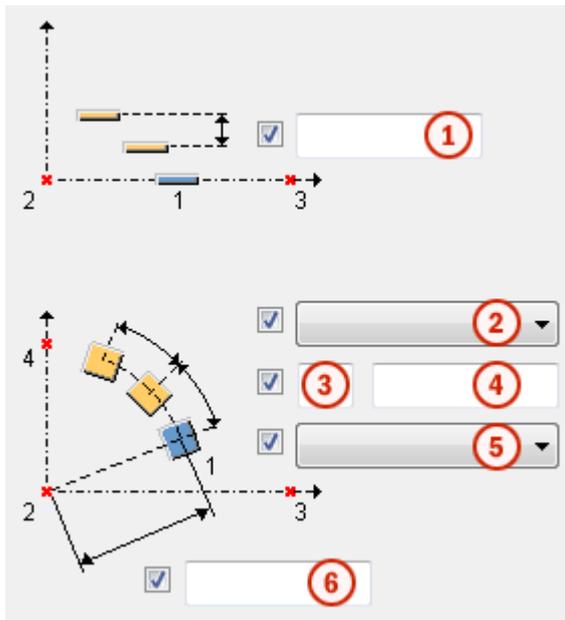


How to use Radial array tool

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **Radial array tool**, and then double-click to open it.
3. Select the **Copy method**. The options are:
 - **Selected objects only**
This is the default. Only the selected objects are copied.
 - **All associated objects**
Selected objects and all objects associated with them are copied. For example, cuts, welds, and bolts.
 - **Advanced**
This option is similar to **All associated objects**, but works better with modifications. For example, when you have stairs that have posts welded to the steps, and you modify the distance between steps.
4. Select the **Rotate copies** option.
The default is **Yes**.
5. Define the rotation axis.
The default is **X**.
6. Define the settings.

7. Select the objects to copy.
 8. Click **OK** to close the dialog box.
 9. Click the middle mouse button.
 10. Pick origin point.
 11. Pick axis direction X.
 12. Pick axis direction Y.
- The selected objects are copied.

How to define the settings



1	Distance between copies. The default value is 0.
2	Rotation. The options are: <ul style="list-style-type: none"> • Angle (default) The copies are rotated by angle. • Distance The copies are rotated by distance.
3	Number of angles or distances. The default value is 0.
4	Space between copies. Use the space character to separate values. Enter a value for each space between copies.

5	<p>Copy direction. The options are:</p> <ul style="list-style-type: none"> • Normal (default) Spacing values are calculated from the origin in positive direction along the axis. • Reverse Spacing values are calculated from the origin in negative direction along the axis. • Centered Copies are centered on the origin. • Mirror Spacing values are calculated from the origin in both positive and negative direction. Mirrored copying doubles the number of copies.
6	<p>Radial distance.</p> <p>The radial distance should be equivalent to the distance you picked when applying the component.</p> <p>If the radial distance is smaller or greater than the picked distance, the spacing between the copied objects is not the same as given in the Space between copies box (4).</p> <p>Tekla Structures calculates the rotation angle according to the dialog box values (spacing and radial distance), and the rotation angle overrides the spacing given in the dialog box.</p>

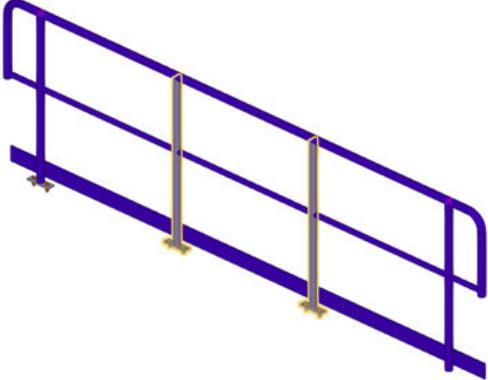
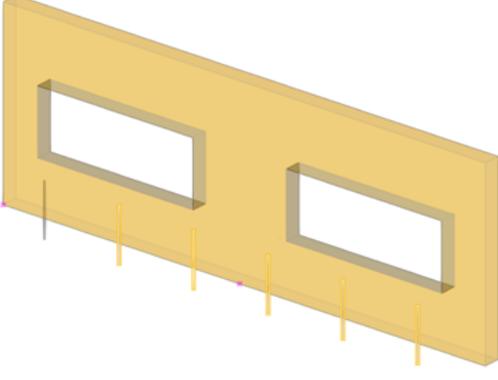
Copy objects using Array of objects (29) component

Use the **Array of objects (29)** component to copy model objects along a line. If you modify the original object, Tekla Structures also changes the copied objects.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for the **Array of objects (29)** component, and then double-click to open it.
3. Define the settings:
 - **Number of copies:** Enter the number of copies you want to create.
 - **Spacing values:** Define the spacing of the objects.
 - **Copy to the opposite direction:** Select **Yes** if you want to copy in the direction opposite to the points you pick.
 - **Start point for copying:** Choose either the object to be copied or the first input point.

- **Copy at equal distances (Ignore spacing values):** Select **Yes** if you want to create the objects at equal distances. **Spacing value** will be ignored.
4. Click **OK** to save the settings.
 5. Select the objects to copy.
 6. Click the middle mouse button to finish selecting.
 7. Pick a point to indicate the start of the line along which to arrange copied objects.
 8. Pick a point to indicate the end of the line.

Examples

Example	Description
	An array of steel objects.
	An array of concrete objects.

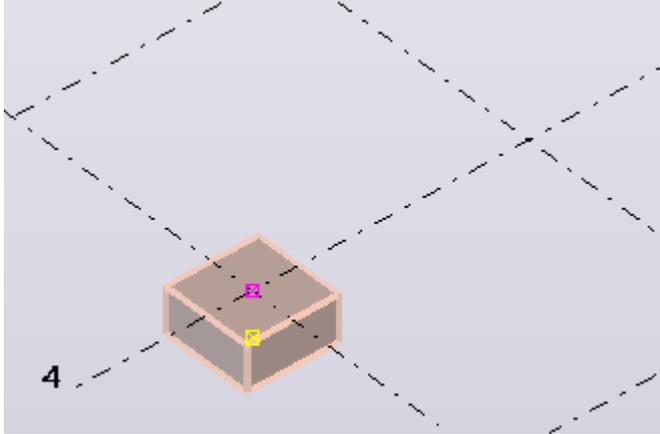
Move objects

You can move objects in a number of different ways, especially in models. When you move an object, Tekla Structures also copies all objects connected to it, including the components.

Move by picking two points

The basic way to move objects in a model or drawing is by defining the origin and one or more destination points.

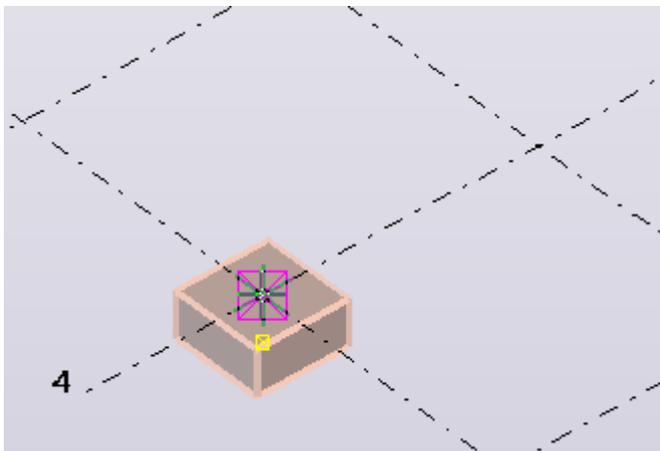
1. Select the object you want to move.



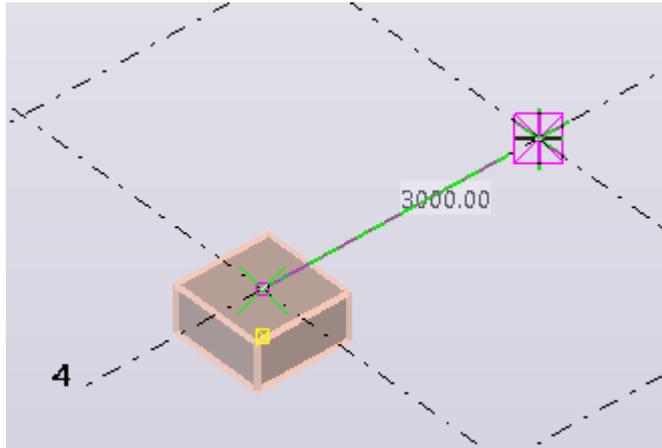
2. Run the **Move** command:

- In the model, on the **Edit** tab, click  **Move**.
- In the drawing, on the **Drawing** tab, click  **Move --> Move**

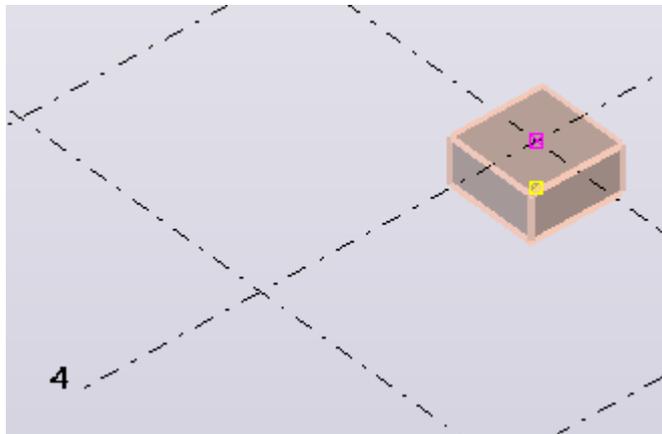
3. Pick the origin for moving.



4. Pick the destination point.



The object is moved immediately. The **Move** command does not remain active.



Move linearly

You can move objects linearly to a new position in the model.

1. Select the objects you want to move.

2. On the **Edit** tab, click  **Move special --> Linear**.

The **Move - Linear** dialog box opens.

3. Pick two points in the model, or enter the coordinates in the **dX**, **dY**, and **dZ** boxes.

You can also use a formula to calculate the x, y, and z displacements. For example:

dY = 3*1250

4. Click **Move**.

TIP If the dialog box is open but the command is not active anymore, click the **Pick...** button to re-activate the command.

Move by specifying a distance from origin

You can move objects to a new position in the model or drawing by specifying a distance from the origin. Use the **Enter a Numeric Location** dialog box to specify the distance.

1. Select the objects you want to move.
2. Run the **Move** command:

- In the model, on the **Edit** tab, click  **Move**.

- In the drawing, on the **Drawing** tab, click  **Move --> Move**.

3. Pick the origin for moving.
4. Move the cursor in the direction you want to move the objects, but do not pick the point.
5. Type the distance.

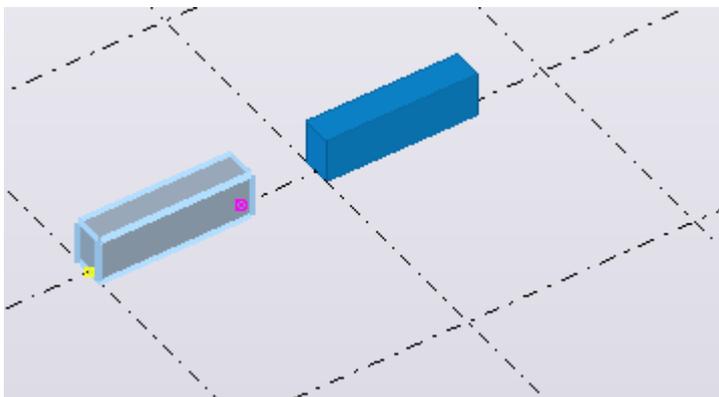
When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box automatically.

6. Click **OK**.

Move using drag-and-drop

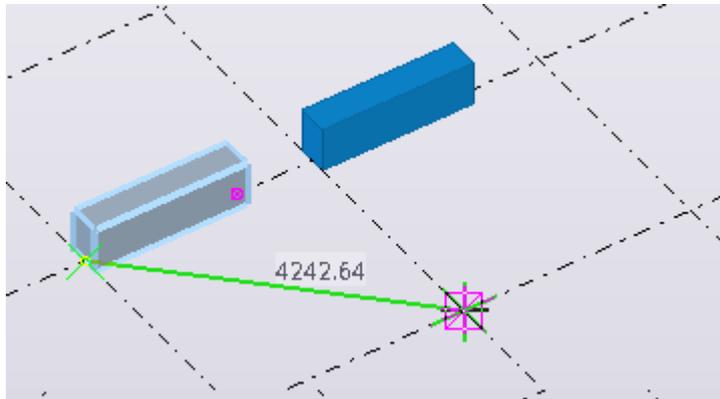
You can move objects by dragging them to a new location.

1. On the **File** menu, click **Settings** and select the **Drag & drop** check box to activate the command.
2. Select the objects you want to move.

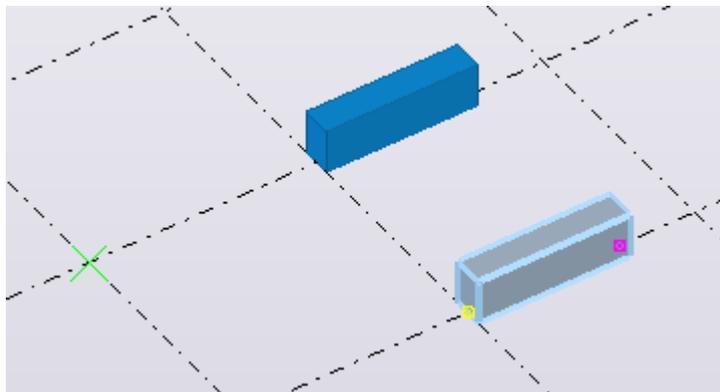


3. Drag the objects to a new location.

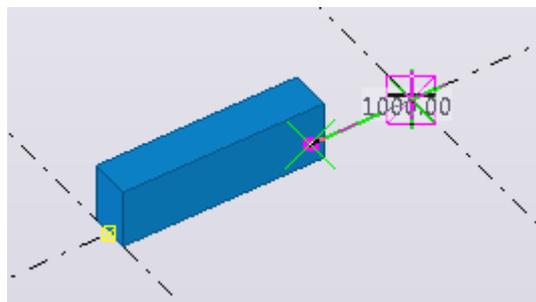
The point you start dragging from (center, corner, or middle point) affects the object's alignment in the new location.



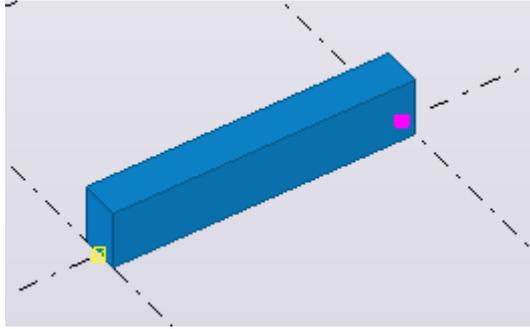
The objects are moved immediately.



4. To move an end point using drag-and-drop:
 - a. Select the handle.
 - b. Hold down the left mouse button, and drag the handle to a new location.



The end point is moved accordingly:



NOTE For some objects, you may need to switch on **Smart select** to drag from handles without selecting them first. To switch it on, click **File** --> **Settings** and select the **Smart select** check box.

NOTE To move grid labels in a drawing, first select the grid label and then either activate the  **Select grid line selection switch** (page 122) or select the grid label handle.

Move to another plane

In a model, you can move objects from the first plane you specify to another plane, which you specify by picking three points. The moved objects remain in the same position on the second plane as the original objects on the first plane.

1. Select the objects you want to move.
2. On the **Edit** tab, click  **Move special** --> **To another plane**.
3. Pick the point of origin of the first plane.
4. Pick a point on the first plane in the positive x direction.
5. Pick a point on the first plane in the positive y direction.
6. Repeat steps 3-5 for the destination plane.

Move objects to another object

In a model, you can move objects from an object to other, similar objects. This is useful, for example, when you detail previously modeled parts. The objects that you move between can have different dimensions, length, and rotation.

1. Select the objects you want to move.
2. On the **Edit** tab, click  **Move special** --> **To another object**.
3. Select the object to move from (source object).

4. Select the objects to move to (target object).

Rotate objects

You can copy or move an object in a model by rotating it around any line you choose. In a drawing, you can copy or move an object by rotating it around a given line on the work plane.

NOTE Positive rotation is according to the [right-hand rule \(page 50\)](#) (clockwise when looking from the start point of the rotation axis).

Rotate around a line

Use the **line** option in the **Rotate** dialog box when you want to copy and rotate, or move and rotate objects around any given line in the model.

1. Select the objects you want to copy or move.
2. Activate the rotation command.

- To copy and rotate, go to the **Edit** tab and click  **Copy special --> Rotate**.

The **Copy - Rotate** dialog box opens.

- To move and rotate, go to the **Edit** tab and click  **Move special --> Rotate**.

The **Move - Rotate** dialog box opens.

3. In the **Around** list, select **line**.
4. Pick the start point of the rotation axis, or enter its coordinates.
5. Pick the end point of the rotation axis, or enter its coordinates.
6. If you are copying, enter the number of copies.
7. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.
8. Enter the rotation angle.
9. Click **Copy** or **Move**.

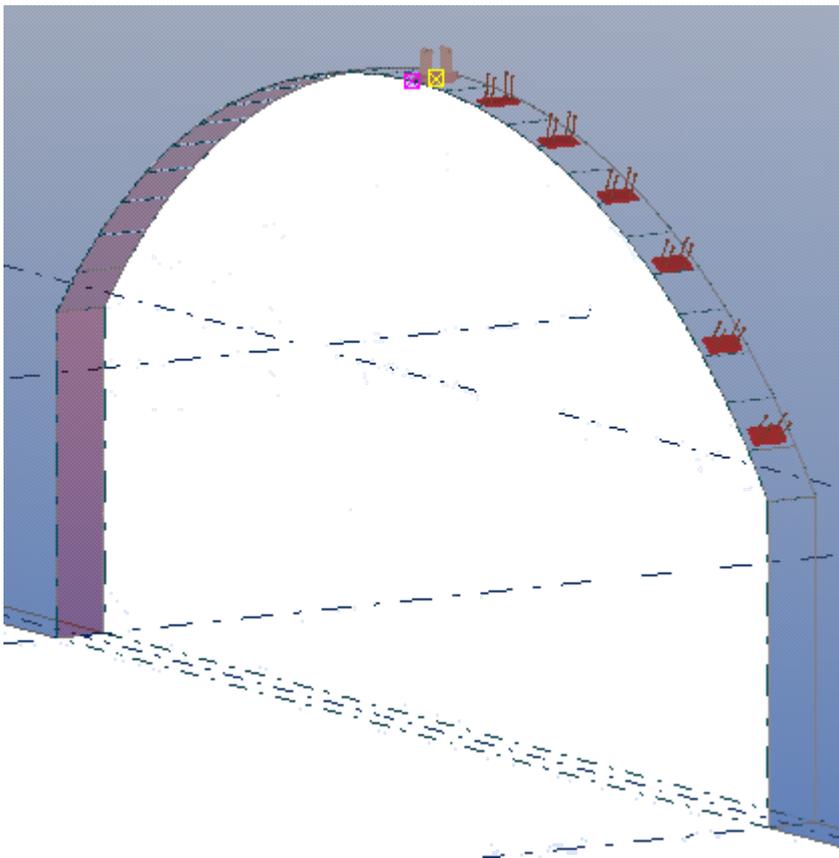
The objects are rotated accordingly.

Example

In this example, a fitting plate is copied and rotated around a construction line that is located at the following coordinates.

Origin	
X0	18000.00
Y0	23847.50
Z0	-900.00
X1	18000.00
Y1	24000.00
Z1	-900.00

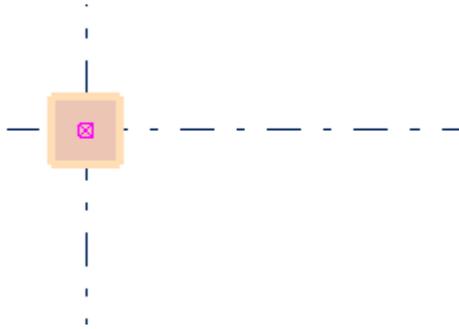
As a result, the copied fitting plates follow the curve of the concrete panel.



Rotate around the z axis

Use the **Z** option in the **Rotate** dialog box when you want to copy and rotate, or move and rotate objects around the z axis in the model.

1. Select the objects you want to copy or move. For example:



2. Activate the rotation command.

- To copy and rotate, go to the **Edit** tab and click  **Copy special** --> **Rotate**.

The **Copy - Rotate** dialog box opens.

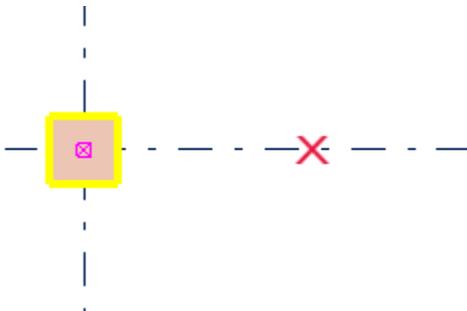
- To move and rotate, go to the **Edit** tab and click  **Move special** --> **Rotate**.

The **Move - Rotate** dialog box opens.

3. Select **Z** in the **Around** list.

4. Pick a point to define the rotation axis, or enter its coordinates.

In the example below, the red cross indicates the picked point.



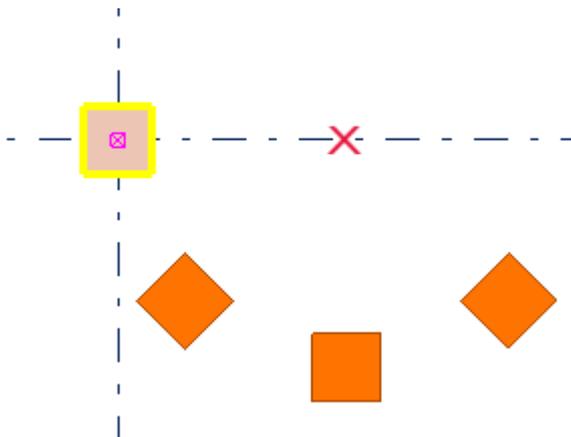
5. If you are copying, enter the number of copies.

6. If needed, enter the **dZ** value, which is the difference in position between the original and copied object in the z direction.

7. Enter the rotation angle. For example:

Copy	
Number of copies	3
dZ	0.00
Rotation	
Angle	45.00000
Around	Z

- Click **Copy** or **Move**.
The objects are rotated accordingly.



Rotate drawing objects

Use this option when you want to rotate drawing objects on the work plane.

- Select the objects you want to copy or move.
- Activate the rotation command.

- To copy and rotate, go to the **Drawing** tab and click  **Copy** --> **Rotate**.

The **Copy - Rotate** dialog box opens.

- To move and rotate, go to the **Drawing** tab and click  **Move** --> **Rotate**.

The **Move - Rotate** dialog box opens.

- Pick a point, or enter its coordinates.
- If you are copying, enter the number of copies.
- Enter the rotation angle.
- Click **Copy** or **Move**.

Rotation settings

Use the **Copy - Rotate** and **Move - Rotate** dialog boxes to view and modify the settings that are used when you rotate objects in Tekla Structures. The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

Option	Description
X0	The x and y coordinates of the start point of the rotation axis.
Y0	
Origin angle	The angle of the rotation axis when rotating around a line on the work plane.
Number of copies	Define the number of copies created.
dZ	The difference in position between the original and copied object in the z direction.
Rotation angle	The rotation angle between the original and new position.
Around	Define whether the rotation axis is a line on the work plane or in the z direction.

Mirror objects

When you copy or move objects, you can mirror them through a plane that is perpendicular to the work plane and passes through a line you specify.

Note that Tekla Structures cannot create mirrored copies of object properties. For example, the **Copy special > Mirror** command does not fully mirror objects if they include components with asymmetrically positioned parts, or reinforcement objects with asymmetric properties such as spacing.

Mirror model objects

Use this method to copy and mirror, or move and mirror objects in a model.

1. Select the objects you want to copy or move.
2. Activate the mirroring command.

- To copy and mirror, go to the **Edit** tab and click  **Copy special --> Mirror**.

The **Copy - Mirror** dialog box opens.

- To move and mirror, go to the **Edit** tab and click  **Move special --> Mirror.**

The **Move - Mirror** dialog box opens.

3. Pick the start point of the mirroring plane, or enter its coordinates.
4. Pick the end point of the mirroring plane, or enter its coordinates.
5. Enter the angle.
6. Click **Copy** or **Move**.

Mirror drawing objects

Use this method to copy and mirror, or move and mirror objects in a drawing.

1. Select the objects you want to copy or move.
2. Activate the mirroring command.

- To copy and mirror, go to the **Edit** tab and click  **Copy special --> Mirror.**

The **Copy - Mirror** dialog box opens.

- To move and mirror, go to the **Edit** tab and click  **Move special --> Mirror.**

The **Move - Mirror** dialog box opens.

3. Pick the start point of the mirroring plane, or enter its coordinates.
4. Pick the end point of the mirroring plane, or enter its coordinates.
5. Enter the angle.
6. Click **Copy** or **Move**.

1.7 Filter objects

Use filters to restrict what can be selected or what is visible in a view. You can create filters of your own, or you can use any of the standard filters available in Tekla Structures.

Here are some examples of what filters can be used for:

- **To select a large number of objects**

Use selection filters when you need to change a specific object property that is common for many objects. The rest of the objects will not be affected, even if you try to include them in the selection.

- **To check the model**

Use view filters to ensure that beams are called beams, columns are called columns, and so on. You can highlight several groups of objects, one at a time, to check that all required objects are included in a given group.

- **To hide objects**

Use view filters to temporarily hide the columns in a view so that it easier to select all the beams, for example.

- **To find objects**

You can create a selection filter to find all locations where ½" reinforcing bars are in the model, for example. Once the filter is active, you can make an area selection that includes the entire model. All specified reinforcing bars become selected, but the other objects will not be affected.

See also

[Use existing filters \(page 156\)](#)

[Create new filters \(page 158\)](#)

[Filtering techniques \(page 166\)](#)

[Examples of filters \(page 186\)](#)

Use existing filters

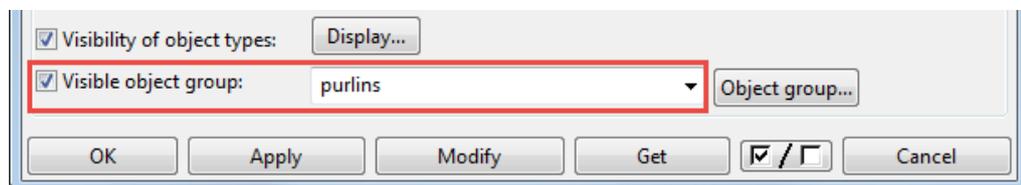
Before creating new custom filters, check out the existing view and selection filters available in Tekla Structures.

How to use a view filter

Use view filters to define which objects are displayed in a model view.

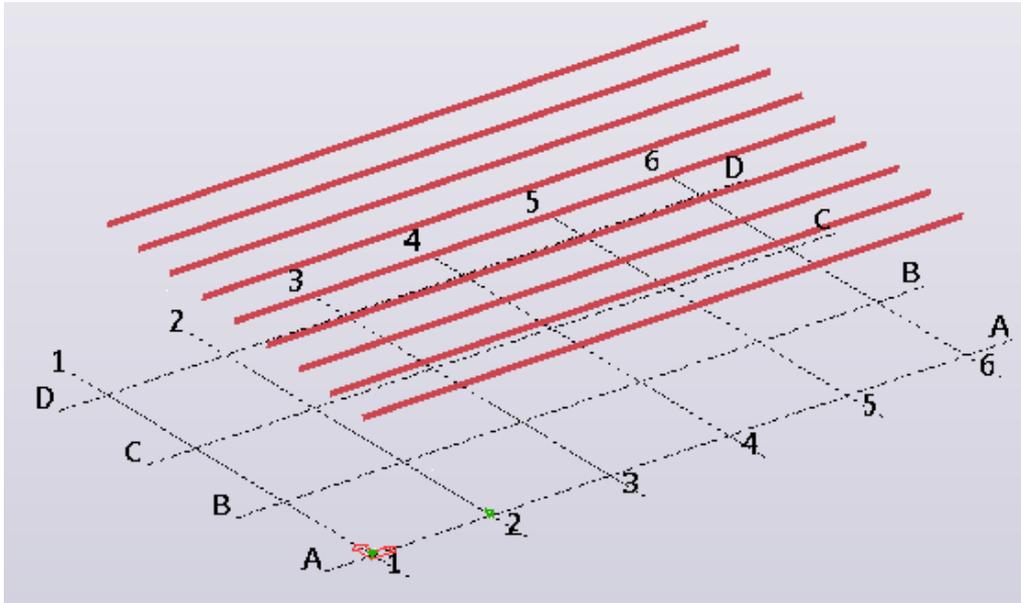
1. Double-click the view to open the **View Properties** dialog box.
2. Select a filter from the **Visible object group** list.

For example, select **purlins**.



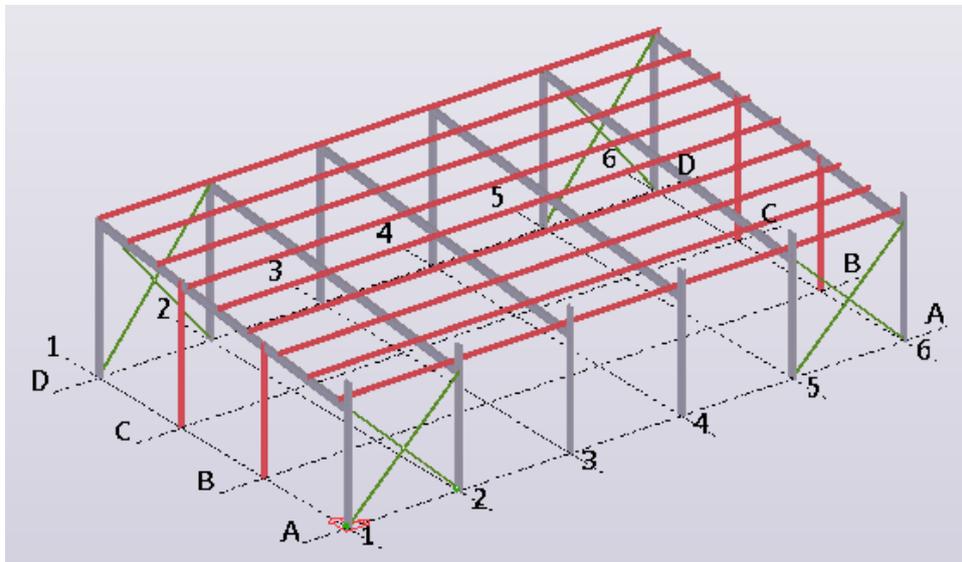
3. Click **Modify**.

Now only the objects defined by the filter are visible. For example, the purlins:



4. To stop using the filter:
 - a. Double-click the view to open the **View Properties** dialog box.
 - b. In the **Visible object group** list, select the **standard** filter.
 - c. Click **Modify**.

All objects are visible again:



NOTE If you cannot see all desired objects (page 48), note that the work area, view depth, view setup, and object representation settings also affect the visibility of objects.

How to use a selection filter

Use selection filters to define which objects can be selected in the model. Note that an object must be visible in the model to be selectable.

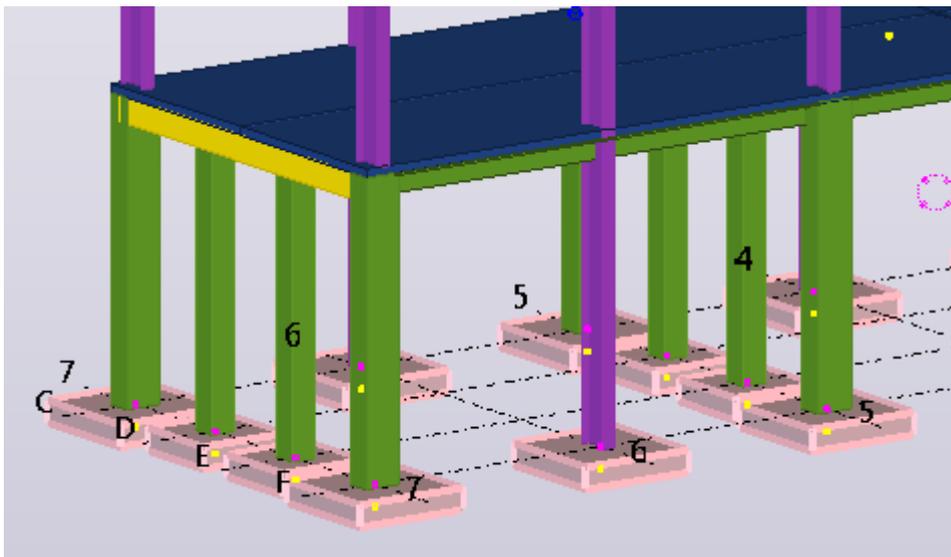
1. On the **Selecting** toolbar, select a filter from the  list.

The list is by default located at the bottom of the Tekla Structures main window.

For example, select the **Name - Footing** filter.

2. Select the desired objects in the model.

You can select multiple objects or even the entire model at once. Now that the filter is active, only objects defined by the filter become selected. For example, if the **Name - Footing** filter is active, only footings are selectable and the rest of the objects stay intact:



3. If you cannot select all objects defined by the selection filter, check your view filter settings and ensure that you have switched on all the needed [selection switches \(page 122\)](#).
4. To stop using the filter, go to the **Selecting** toolbar and select the **standard** filter.

All objects are selectable again.

Create new filters

You can create custom filters to define which objects are visible and selectable in the model and drawings. Add new filter rules, one on each row, to define which objects should be included or excluded.

Create a view filter

You can create your own custom filters to define which objects are visible in a model.

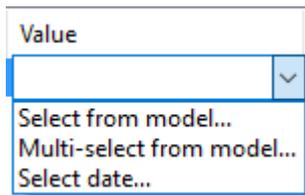
1. Double-click the view to open the **View Properties** dialog box.
2. Click **Object group**.
The **Object Group - View Filter** dialog box opens, showing the currently active filter.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

You have the following options:

- Part
- Component
- Bolt
- Weld
- Reinforcing bar
- Surface
- Assembly
- Construction object
- Load
- Template
- Reference assembly
- Reference object
- Location breakdown structure
- Pour object
- Pour break
- Pour unit
- Task
- Object

6. In the **Property** list, select a suitable [object property \(page 169\)](#).
The options vary depending on the object category you chose in step 5.
7. In the **Condition** list, select a suitable [condition \(page 166\)](#).
8. In the **Value** list, type a value.

Alternatively, to use the current value of an existing object, click **Select from model...** and select the desired object from the model. To use the values of multiple objects, click **Multi-select from model**, select the objects from the model, and then click the middle mouse button. For date values, also the **Select date...** option is available.



The values can be complete strings, such as the profile name UC310*97. You can also use incomplete strings together with [wildcards \(page 186\)](#). For example, the value UC* will match with all parts whose profile name begins with the characters UC*. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, 12 5). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, "custom panel"), or use a question mark (for example, custom?panel) to replace the space.

9. Repeat steps 4–8 to create as many filter rules as needed.
You can apply several filter rules at the same time.
10. Use the [And/Or options and brackets \(page 166\)](#) to define how multiple rows work together.
11. Select the check boxes next to all filter rules that you want to enable.
If the check box is selected, the filter rule is enabled and effective. For example:

-	(Category	Property
<input checked="" type="checkbox"/>	-	Object	Object type
<input checked="" type="checkbox"/>	-	Part	Phase

By default, each new rule is disabled.

12. Define the filter type.
 - a. Click  to display more settings.

- b. Select or clear the check boxes to define where the filter will be used.
For example, the same filter may be used both as a view filter and as a selection filter.

13. Enter a unique name in the box next to the **Save as** button.

-
- NOTE**
- Filters are case sensitive.
 - Do not use spaces in filter names.
 - We recommend that you use _ (underscores) in your naming convention.
 - To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.
-

14. Click **Save as** to save the filter.

15. To apply the filter to the current view, click **Modify**.

Create a selection filter

You can create your own custom filters to help you select objects in a model.

1. On the **Selecting** toolbar, click  to open the **Object Group - Selection Filter** dialog box.



2. Follow the instructions above on how to create a view filter.
The same instructions apply to selection filters.

Create a drawing filter

For general arrangement drawings, you can create drawing filters that affect the whole drawing, not just a specific view. Drawing filters select objects in the whole drawing.

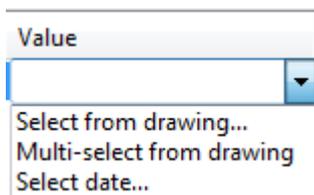
You can use drawing filters together with saved object property files when you create and apply object level settings in the whole drawing. For example, you might create a filter that selects all beams, then save an object property file that defines that the part color is blue, and then create and apply an object level settings file that changes all beams to blue in the whole drawing.

1. In a GA drawing, on the **Drawing** tab, click **Properties --> Drawing**.
2. Click **Filter**.
3. Click **New filter** to create a new filter from scratch.
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select an object category.

You have the following options:

- Part
 - Component
 - Bolt
 - Weld
 - Reinforcing bar
 - Surface
 - Assembly
 - Construction object
 - Template
 - Reference assembly
 - Reference object
 - Location breakdown structure
 - Pour object
 - Pour break
 - Pour unit
 - Task
 - Object
6. In the **Property** list, select a suitable [object property \(page 169\)](#).
The options vary depending on the object category you chose in step 5.
 7. In the **Condition** list, select a suitable [condition \(page 166\)](#).
 8. In the **Value** list, type a value.

Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.



The values can be complete strings, such as the profile name UC310*97. You can also use incomplete strings together with [wildcards \(page 186\)](#). For example, the value UC* will match with all parts whose profile name

begins with the characters UC*. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, 12 5). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, "custom panel"), or use a question mark (for example, custom?panel) to replace the space.

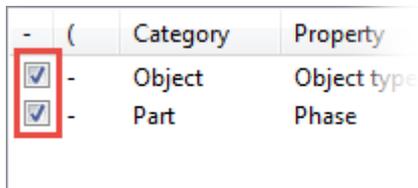
9. Repeat steps 4–8 to create as many filter rules as needed.

You can apply several filter rules at the same time.

10. Use the **And/Or options and brackets** (page 166) to define how multiple rows work together.

11. Select the check boxes next to all filter rules that you want to enable.

If the check box is selected, the filter rule is enabled and effective. For example:



-	(Category	Property
<input checked="" type="checkbox"/>	-	Object	Object type
<input checked="" type="checkbox"/>	-	Part	Phase

By default, each new rule is disabled.

12. Define the filter type.

- a. Click  to display more settings.
- b. Select or clear the check boxes to define where the filter will be used.

For example, the same drawing filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

13. Enter a unique name in the box next to the **Save as** button.

-
- NOTE**
- Filters are case sensitive.
 - Do not use spaces in filter names.
 - We recommend that you use _ (underscores) in your naming convention.
 - To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.
-

14. Click **Save as** to save the filter.

15. When you are done, click **Cancel** to close the filter properties dialog box.

Create a drawing view filter

You can create your own custom view filters to help you select a specific group of view objects in a drawing view.

You can use drawing view filters for changing the appearance of a certain object group, or for selecting which objects are hidden in a drawing view.

You can also use drawing view filters together with saved object property files when you create and apply object level settings in the selected view. For example, you might create a view filter that selects all columns in a view, then save an object property file that defines that the part color is red, and then create and apply an object level settings file that changes all columns to red in the selected view.

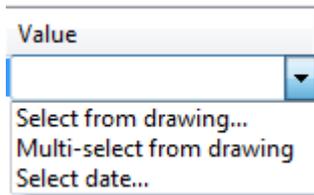
1. Open a drawing.
2. Double-click a drawing view frame.
3. Click **Filter**.
4. Click **New filter** to create a new filter from scratch.
5. Click **Add row** to add a new filter rule.
6. In the **Category** list, select an object category.

You have the following options:

- Part
- Component
- Bolt
- Weld
- Reinforcing bar
- Surface
- Assembly
- Construction object
- Template
- Reference assembly
- Reference object
- Location breakdown structure
- Pour object
- Pour break
- Pour unit
- Task
- Object

7. In the **Property** list, select a suitable [object property \(page 169\)](#).
The options vary depending on the object category you chose in step 5.
8. In the **Condition** list, select a suitable [condition \(page 166\)](#).
9. In the **Value** list, type a value.

Alternatively, to use the current value of an existing object, click **Select from drawing** and select the desired object from the drawing. To use the values of multiple objects, click **Multi-select from drawing**, select the objects from the drawing, and then click the middle mouse button. For date values, also the **Select date...** option is available.



The values can be complete strings, such as the profile name UC310*97. You can also use incomplete strings together with [wildcards \(page 186\)](#). For example, the value UC* will match with all parts whose profile name begins with the characters UC*. Empty values are matched to empty object properties.

If you use multiple values, separate the strings with blank spaces (for example, 12 5). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, "custom panel"), or use a question mark (for example, custom?panel) to replace the space.

10. Repeat steps 4–8 to create as many filter rules as needed.
You can apply several filter rules at the same time.
11. Use the [And/Or options and brackets \(page 166\)](#) to define how multiple rows work together.
12. Select the check boxes next to all filter rules that you want to enable.
If the check box is selected, the filter rule is enabled and effective. For example:

-	(Category	Property
<input checked="" type="checkbox"/>	-	Object	Object type
<input checked="" type="checkbox"/>	-	Part	Phase

By default, each new rule is disabled.

13. Define the filter type.
 - a. Click to display more settings.

- b. Select or clear the check boxes to define where the filter will be used.
For example, the same drawing view filter may be used both as a model view filter and as a model selection filter, and also as an Organizer filter.

14. Enter a unique name in the box next to the **Save as** button.

- NOTE**
- Filters are case sensitive.
 - Do not use spaces in filter names.
 - We recommend that you use _ (underscores) in your naming convention.
 - To have the filter appear at the top of the list, right after the standard filter, use capital letters in the filter name.
-

15. When you are done, click **Cancel** to close the filter properties dialog box.

Create a drawing selection filter

You can create your own custom filters to help you select objects in a drawing.

You can use selection filters in drawings if you want to hide certain parts from drawings or drawing views, or change the part color or representation for certain parts.

Also, if you have some different looking part marks for different types of parts, you can select the specific parts using a selection filter and then only modify part marks for those parts.

1. In an open drawing, on the **Selecting** toolbar, click  (**Ctrl+G**).
The **Selection filter** dialog box opens.
2. Follow the instructions above on how to create a drawing or drawing view filter.
The same instructions apply to drawing selection filters.
3. Click **Apply** or **OK** to select the parts according to the filter.

Filtering techniques

By using conditions, brackets, and the **And/Or** options you can create filters that can be as complex as needed.

Conditions

Use conditions to define how filter criteria relate to one another. Note that when you create filters, you are always defining what should be **shown** (or be selectable) in the model or drawing. Therefore, if you enter "Component name does not contain gusset", you are telling Tekla Structures to show all

components whose name **does not contain** the word "gusset". Tekla Structures then hides all components which have the word "gusset" in their name.

Condition	Description
Equals	Use this condition when the filter value must be matched exactly. For example, "Part name equals BEAM."
Does not equal	Filters out objects that contain the value you enter. For example, "Part profile does not equal BL200*20" means that Tekla Structures will hide (or not select) the objects whose profile is BL200*20. The rest of the objects will be shown (or selected).
Begins with	Finds all objects that start with the value you enter. For example, "Component name begins with purlin."
Does not begin with	Filters out objects that start with the value you enter. For example, "Component name does not begin with shear" means that Tekla Structures will hide (or not select) the objects whose name begins with the word "shear". The rest of the objects will be shown (or selected).
Ends with	Finds all objects that end with the value you enter. For example, "Component name ends with plate."
Does not end with	Filters out objects that end with the value you enter. For example, "Component name does not end with angle" means that Tekla Structures will hide (or not select) the objects whose name ends with the word "angle". The rest of the objects will be shown or selected.
Contains	Finds all objects that include the value you enter. For example, "Component name contains plate" would find base plate and shear plate simple.
Does not contain	Filters out objects that include the value you enter. For example, "Component name does not contain

Condition	Description
	<p>"gusset" means that Tekla Structures will hide (or not select) the objects whose name contains the word "gusset". The rest of the objects will be shown or selected.</p>
Greater than	<p>Finds all objects that exceed the value you enter. For example, "Template attribute LENGTH is greater than 5000."</p> <p>This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.</p>
Greater or equal	<p>Finds all objects that match or exceed the value you enter.</p> <p>This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.</p>
Less than	<p>Finds all objects that are less than the value you enter.</p> <p>This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.</p>
Less or equal	<p>Finds all objects that match or are less than the value you enter.</p> <p>This property can only be used with numerical data, such as the part start number, class, phase, or LENGTH.</p>
Later than	<p>Available for dates only. The date must be later than the one you have defined. For example, "Object's approval date later than 4/10/2017."</p>
Later than or equal	<p>Available for dates only. The date must be later than or equal to the one you have defined.</p>
Earlier than	<p>Available for dates only. The date must be earlier than the one you have defined. For example, "Object's approval date earlier than 2/18/2017."</p>
Earlier than or equal	<p>Available for dates only. The date must be earlier than or equal to the one you have defined.</p>

And/Or options

Use the **And/Or** options when you create filter rules that consist of several rows.

Option	Description
And	Use to find objects that match both values. When you create filter rules between objects whose Category settings differ, use the And option when possible to avoid potential problems with more complex rules.
Or	Use to find objects that match either value.
empty (= And)	Empty means the same as And .

Brackets

You can use single, double, and triple brackets to create more complex filter rules.

Example 1. Use the format "A and (B or C)" to find objects that match the first filter rule and **either** of the last two rules.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Part	Name	Equals	BRACING	-	And
<input checked="" type="checkbox"/>	(Part	Phase	Equals	1	-	Or
<input checked="" type="checkbox"/>	-	Part	Phase	Equals	3)	Or

Example 2. Use the format "(A and B) or C" to find objects that match both of the first two rules **or** the third.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	(Part	Name	Equals	COLUMN	-	And
<input checked="" type="checkbox"/>	-	Part	Profile	Equals	IPE300)	Or
<input checked="" type="checkbox"/>	-	Part	Material	Equals	S235JR	-	Or

Object properties in filtering

You can select from a wide variety of object properties when creating new filters. In the tables below, the properties are listed according to their object category. In addition to these, almost all categories contain user-defined attributes and template attributes that can also be used in filtering.

Category: Object

Use the **Object** category to filter objects based on their object level properties.

Property	Description
GUID	To filter objects based on their globally unique identifier (GUID). For example, "Object GUID begins with ID7554C9EB-C8B4."
Phase	To filter objects based on their phase number. For example, "Object phase does not equal 3."
Object type	<p>To filter objects based on their type. Select an object type from the Value list, or use the Select from model... or Multi-select from model option.</p> <p>NOTE: We recommend that you include one filter rule for the Object type property in each filter you create. This ensures that only objects of this type are filtered in. If you leave the object type out, the filtering outcome will be different, and objects that do not match the category in further filter rules may be filtered in.</p> <p>The following object types can be selected from the list:</p> <ul style="list-style-type: none">• Assembly• Bolt group• Connection• Part• Pour break• Pour object• Pour unit• Reference object• Reinforcing bar• Surface• Surface treatment• Weld <p>The following object types are shown as numeric values only:</p>

Property	Description
	<ul style="list-style-type: none"> • 1 = point • 9 = fitting • 11 = polygon cut • 12 = line cut • 24 = construction line • 30 = construction plane • 38 = added material • 42 = construction circle • 48 = reference model • 70 = edge chamfer • 76 = analysis part
Is component	To filter objects based on whether or not they are components. The options are Yes and No . For example, "Object is component equals Yes."

Some object types are not visible directly, but only when the objects that make them up are visible. For example, assemblies are visible when parts are visible, and pour units are visible when pour objects are visible. Therefore, using the object type **Assembly** or **Pour unit** alone in a view filter in the model or in drawings does not show anything. However, selection filters are able to select object types like assemblies and pour units.

Category: Part

Use the **Part** category to filter [parts \(page 235\)](#) based on their common properties.

Property	Description
Name	To filter objects based on their name. For example, "Part name equals SLAB."
Profile	To filter objects based on their profile. For example, "Part profile does not equal L20*2."
Material	To filter objects based on their material grade. For example, "Part material equals C25/30."
Finish	To filter objects based on how the part surface has been treated. For example, "Part finish equals "FP - Fire proofing"."

Property	Description
Prefix	To filter objects based on their numbering prefix. For example, "Part prefix equals P."
Start number	To filter objects based on their start number. For example, "Part start number greater than 100."
Numbering series	To filter objects based on their numbering series information. For example, "Part numbering series equals TP/1." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in File menu --> Settings --> Options --> Numbering.
Position number	To filter objects based on their position number. For example, "Part position number does not equal P/5." Note that position number separator can be a dot (.), comma (,), slash (/), or hyphen (-), depending on what you have defined in File menu --> Settings --> Options --> Numbering.
Class	To filter objects based on their class number. For example, "Part class equals 210."
Phase	To filter objects based on their phase number. For example, "Part phase equals 1 2."
Lot	To filter objects based on their lot number. For example, "Part lot greater than 1."
Primary part	To filter objects based on whether they are main or secondary parts in an assembly or cast unit. 1 = primary part, 0 = secondary part. For example, "Part primary part equals 1."
Pour phase	To filter parts based on their pour phase. For example, "Pour phase does not equal 0."

Category: Component

Use the **Component** category to filter components based on their common properties.

Property	Description
Name	To filter components based on their name. For example, "Component name equals "shear plate simple"."
Connection code	To filter components based on their connection code, which can be either a text string or a number. For example, "Component connection code equals 200_2."
Running number	To filter components based on their unique running number. For example, "Component running number less than 150."
Phase	To filter components based on their phase number. For example, "Component phase equals 2."
Is conceptual	To filter components based on their type. Components can be either detailed or conceptual. Yes = conceptual, No = detailed. For example, "Component is conceptual equals Yes."

Category: Bolt

Use the **Bolt** category to filter bolts based on their common properties.

Property	Description
Size	To filter bolts based on their diameter. For example, "Bolt size less than 20.00."
Standard	To filter bolts based on their bolt assembly standard/grade. For example, "Bolt standard equals 7990."
Site/Workshop	To filter bolts based on their assembly type. Site = 0, Workshop = 1. For example, "Bolt Site/Workshop equals 1."

Property	Description
Phase	To filter bolts based on their phase number. For example, "Bolt phase equals 3 4."
Length	To filter bolts based on their length. For example, "Bolt length greater than 50.00."

Category: Weld

Use the **Weld** property to filter [welds \(page 346\)](#) based on their common properties.

Property	Description
Size above line Size below line	To filter welds based on their size. For example, "Weld size above line equals 5.00."
Reference text	To filter welds based on their reference text, which is a user-definable value in the Weld properties. For example, "Weld reference text contains 12345."
Phase	To filter welds based on their phase number. For example, "Weld phase equals 3."
Type above line Type below line	To filter welds based on their weld type (page 354) . Select the type from the Value list.
Length above line Length below line	To filter welds based on their length value. For example, "Weld length greater than 0.00."
Welding site	To filter welds based on where they should be made. The options are Site and Workshop .
Position number	To filter welds based on their unique position number. For example, "Weld position number is greater than 100."
Angle above line Angle below line	To filter welds based on the angle of weld preparation, bevels, or groove. For example, "Weld angle below line greater than 0.000."
Contour above line Contour below line	To filter welds based on their fill type contour. The options are None , Flush , Convex , and Concave . For

Property	Description
	example, "Weld contour above line does not equal None ."
Effective throat above line Effective throat below line	To filter welds based on their weld size used in weld strength calculation. For example, "Weld effective throat above line equals 0.500."
Finish above line Finish below line	To filter welds based on how they have been treated. The options are None, Grind, Machine, Chip, Finished weld, and Smooth transition.
Increment amount above line Increment amount below line	To filter welds based on their amount of increments. For example, "Weld increment amount above line greater than 0."
Intermittent type	To filter welds based on their shape. The options are Continuous, Intermittent, and Staggered intermittent.
Pitch above line Pitch below line	To filter welds based on their spacing of weld increments.
Root face thickness above line Root face thickness below line	To filter welds based on their root face thickness, in other words, the height of the narrowest part inside the root opening.
Root opening above line Root opening below line	To filter welds based on the space found between the welded parts.
Size prefix above line Size prefix below line	To filter welds based on their weld size prefix. For example, "Weld size prefix above line equals a." The standard ISO 2553 prefixes are a (Design throat thickness), s (Penetration throat thickness), and z (Leg length).
User defined cross-section	To filter welds based on whether or not they contain user-defined cross-sections. The options are Yes and No .
Electrode classification	To filter welds based on their weld electrode classification. The options are (empty), 35, 52, 50, E60XX, E70XX, E80XX, and E90XX.
Electrode strength	To filter welds based on their electrode strength. For example,

Property	Description
	"Weld electrode strength greater than 0.000."
Electrode coefficient	To filter welds based on their electrode coefficient.
Process type	To filter welds based on their welding process type. The options are SMAW, SAW, GMAW, FCAW, ESW, and EGW.
NDT inspection	To filter welds based on their non-destructive testing and inspection level. The options are A, B, C, D, and E.
Is around weld	To filter welds based on whether only one edge or the entire perimeter of a face is welded. No = edge, Yes = around.

Category: Reinforcing bar

Use the **Reinforcing bar** category to filter [reinforcing bars \(page 451\)](#) based on their common properties.

Property	Description
Name	To filter reinforcing bars based on their name. For example, "Reinforcing bar name equals "STIRRUP"."
Class	To filter reinforcing bars based on their class number. For example, "Reinforcing bar class equals 3."
Size	To filter reinforcing bars based on their size. The size property depends on the environment and may contain letters and special characters. For example, in the US imperial environment, "Reinforcing bar size equals #18."
Diameter	To filter reinforcing bars based on their diameter. Diameter is the nominal diameter of the bar, not actual. For example, "Reinforcing bar diameter less than 12."
Length	To filter reinforcing bars based on their total length. For example, "Reinforcing bar length greater than 5000.00."

Property	Description
Material	To filter reinforcing bars based on their material grade. For example, "Reinforcing bar material does not equal Undefined."
Prefix	To filter reinforcing bars based on their numbering prefix. For example, "Reinforcing bar prefix equals R."
Start number	To filter reinforcing bars based on their start number. For example, "Reinforcing bar start number greater than 1."
Numbering series	To filter reinforcing bars based on their numbering series information. For example, "Reinforcing bar numbering series equals R/1."
Position number	To filter reinforcing bars based on their position number. For example, "Reinforcing bar position number equals R/3."
Phase	To filter reinforcing bars based on their phase number. For example, "Reinforcing bar phase equals 2."
Shape	To filter reinforcing bars based on their bending shape (page 535) . For example, "Reinforcing bar shape does not equal 2_1."

Category: Surface

Use the **Surface** category to filter [surfaces \(page 386\)](#) based on their common properties.

Property	Description
Name	To filter surfaces based on their name. For example, "Surface name equals SURFACE."
Type	To filter surfaces based on their type. The options are Formwork and Concrete finish .
Class	To filter surfaces based on their class number. For example, "Surface class does not equal 13."

Property	Description
Phase	To filter surfaces based on their phase number. For example, "Surface phase equals 3 4."

Category: Assembly

Use the **Assembly** category to filter [assemblies \(page 386\)](#) and [cast units \(page 396\)](#) based on their common properties.

Property	Description
Name	To filter assemblies and cast units based on their name. For example, "Assembly name does not contain RAFTER."
GUID	To filter assemblies based on their globally unique identifier (GUID). For example, "Assembly GUID equals ID89F414A7-ECA6-4B14-99CB-6985B84E64CB."
Prefix	To filter assemblies and cast units based on their numbering prefix. For example, "Assembly prefix equals A."
Start number	To filter assemblies and cast units based on their start number. For example, "Assembly start number greater than 1."
Position number	To filter assemblies and cast units based on their position number. For example, "Assembly position number equals A/13."
Phase	To filter assemblies and cast units based on their phase number. For example, "Assembly phase does not equal 1."
Assembly level	To filter assemblies and cast units based on their position in the assembly hierarchy (page 390) . The larger the value, the lower the position in the assembly hierarchy. 0 is the top-most level, and 1 is the first sub-assembly level. For example, to check if the model contains sub-assemblies, use the filter rule "Assembly level greater than or equal to 1."

Property	Description
Assembly type	To filter assemblies and cast units based on their type. <ul style="list-style-type: none"> • 0 = precast • 1 = cast in place • 2 = steel • 3 = timber • 6 = miscellaneous
Assembly series	To filter assemblies and cast units based on their numbering series information. For example, "Assembly series equals C/1."

Category: Construction object

Use the **Construction object** category to filter construction objects based on their common properties.

Property	Description
Phase	To filter construction objects based on their phase number. For example, "Construction object phase does not equal 1."
Type	To filter construction objects based on their type. The options are Line, Arc, Circle, Plane, and Polycurve.

Category: Load

Use the **Load** category to filter loads based on their common properties.

Property	Description
Load group	To filter loads based on which load group they belong to. For example, "Load group does not equal DefaultGroup."
Load type	To filter loads based on their type. The options are line, point, area, uniform, and temperature. Note that wind loads are handled as area loads in filtering. Use the  Select components and  Select

Property	Description
	objects in components selection switches to select wind loads.
Phase	To filter loads based on their phase number. For example, "Load phase does not equal 1."

Category: Template

Use the **Template** category to filter parts and other objects by using template attributes.

With this category, you can type the name of any template attribute or user-defined attribute directly in the **Property** box, even if it is not on the list. Use the `ASSEMBLY.`, `CAST_UNIT.`, or `POUR_UNIT.` prefix in front of the property name to access higher hierarchy level attributes, and the `USERDEFINED.` prefix to access user-defined attributes.

For example, to filter objects that are one hierarchy level below the cast-unit with the user-defined attribute **User field 1**, type `CAST_UNIT.USERDEFINED.USER_FIELD_1` in the **Property** box.

In some cases, you can filter objects based on properties of other objects from lower hierarchy levels. This is possible when there is only one lower level object to be accessed. For example, there is only one main part in each assembly or cast unit, so you can access the main part properties from the assembly or cast unit level using the `MAINPART` prefix. Similarly, there can only be one pour object in each pour unit, so you can access the pour object properties from the pour unit level using the `POUR_OBJECT` prefix.

For example, to filter objects in assemblies whose main part has a certain name, type `ASSEMBLY.MAINPART.NAME` in the **Property** box.

For example, to filter all reinforcing bars belonging to pour units with a certain pour object type, type `POUR_UNIT.POUR_OBJECT.POUR_TYPE` in the **Property** box.

Category: Reference assembly

Use the **Reference assembly** category to filter reference model assemblies based on their common properties.

Property	Description
Creation	
GUID	To filter reference model assemblies based on their globally unique identifier (GUID). For example, "Reference assembly GUID equals IDA51E6BFF-DAB9-4A56-970C-7486EF17B7B7."

Property	Description
Phase	To filter reference model assemblies based on their phase number. For example, "Reference assembly phase equals 2."
Lot	To filter reference model assemblies based on their lot number. For example, "Reference assembly lot greater than 1."
Description	To filter reference model assemblies based on their description, which is a user-definable value in the Reference object dialog box. For example, "Reference assembly description contains "architect model"."
Info text	To filter reference model assemblies based on their info text, which is a user-definable value in the Reference object dialog box. For example, "Reference assembly info text contains revised."
Locked	To filter reference model assemblies based on whether or not they are locked. 0 = No, 1 = Yes, 2 = Organization.
Logical name	To filter reference model assemblies based on their logical name, which is a user-definable value in the Reference object dialog box. For example, "Reference assembly logical name equals "MEP heating system"."

Category: Reference object

Use the **Reference object** category to filter reference model objects based on their common properties.

Property	Description
Creation	
GUID	To filter reference model objects based on their globally unique identifier (GUID).
Phase	To filter reference model objects based on their phase number. For

Property	Description
	example, "Reference object phase does not equal 1."
Lot	To filter reference model objects based on their lot number. For example, "Reference object lot equals 1."
Description	To filter reference model objects based on their description, which is a user-definable value in the Reference object dialog box. For example, "Reference object description contains "architect model"."
Info text	To filter reference model objects based on their info text, which is a user-definable value in the Reference object dialog box. For example, "Reference object info text contains revised."
Locked	To filter reference model objects based on whether or not they are locked. 0 = No, 1 = Yes, 2 = Organization.
Logical name	To filter reference model objects based on their logical name, which is a user-definable value in the Reference object dialog box. For example, "Reference object logical name contains "3rd floor"."

TIP You can filter reference model object attributes using the **Template** category and the `EXTERNAL.` prefix in the **Property** box. For example, "Reference object `EXTERNAL.Material` equals A572."

Category: Location breakdown structure

Use the **Location breakdown structure** category to filter objects based on their location categories, which can be defined in **Organizer**.

Property	Description
Site	To filter objects based on which site category they belong to. For example, "Location breakdown structure site equals "Site 2"."
Building	To filter objects based on which building category they belong to. For

Property	Description
	example, "Location breakdown structure building does not equal "Building A"."
Section	To filter objects based on which section category they belong to. For example, "Location breakdown structure section equals Ramp."
Floor	To filter objects based on which floor they are located on. For example, "Location breakdown structure floor equals "Floor 4"."

Category: Pour object

Use the **Pour object** category to filter [pour objects \(page 408\)](#) based on their common properties.

Property	Description
Pour number	To filter pour objects based on their pour number. For example, "Pour number equals 5."
Pour type	To filter pour objects based on their type. For example, "Pour type equals WALL."
Concrete mixture	To filter pour objects based on the properties of their concrete mix, for example, maximum grain size of aggregate, and/or plasticity of fresh concrete.
Material	To filter pour objects based on their material grade. For example, "Material equals C35/45 ."
Pour phase	To filter pour objects based on their pour phase. For example, "Pour phase does not equal 0."

Category: Pour break

Use the **Pour break** category to filter [pour breaks \(page 415\)](#) based on their common properties.

Property	Description
Creation	
ID number	To filter pour breaks based on their ID number. For example, "ID number equals 25237."

Property	Description
Phase	To filter pour breaks based on their phase. For example, "Pour break phase equals 2 3."
Pour break type	To filter pour breaks based on their type. For example, "Pour break type equals "Watertight construction joint"."

Category: Pour unit

Use the **Pour unit** category to filter pour units based on their common properties.

Property	Description
Name	To filter pour units based on their name. For example, "Pour unit name contains beam."
GUID	To filter pour units based on their globally unique identifier (GUID). For example, "Pour unit GUID contains 8505."

Category: Task

Use the **Task** category to filter scheduled tasks based on their common properties.

Property	Description
Name	To filter scheduled tasks based on their name. For example, "Task name contains floors."
Planned start date	To filter scheduled tasks based on their planned start date. For example, "Task planned start date earlier than Review date."
Planned end date	To filter scheduled tasks based on their planned start date. For example, "Task planned end date later than or equal to 10/13/2017."
Actual start date	To filter scheduled tasks based on their actual start date.
Actual end date	To filter scheduled tasks based on their actual end date.
Completeness	To filter scheduled tasks based on their completeness. The value is a

Property	Description
	percentage. For example, "Task completeness is 75."
Critical	To filter scheduled tasks based on how critical they are. A task can be critical only if it has been imported from external software. 1 = Critical, 0 = Not critical. Note that this property is not visible in Task manager .
Local	To filter scheduled tasks based on whether they were created in Task manager or imported from external software. 1 = Created in Task Manager, 0 = Imported.
Contractor	To filter scheduled tasks based on the contractor. For example, "Task contractor equals "Contractor A"."
Scenario	To filter scheduled tasks based on the scenario they belong to. For example, "Task scenario equals "Scenario 1"."
Task type	To filter scheduled tasks based on their type. For example, "Task type does not equal "A - Floor tiling"."

Template attributes in filtering

Use the following units when filtering template attributes, even when using the US Imperial environment:

- **mm** for length
- **mm2** for area
- **kg** for weight
- **degree** for angle

TIP To check which unit Tekla Structures uses for a particular template attribute, use the **Select from model...** option in the **Value** list in the filtering dialog box.

See also

[Create new filters \(page 158\)](#)

Wildcards

A wildcard is a symbol that stands for one or more characters. You can use wildcards to shorten strings, for example in filtering.

Wildcard	Description	Example
* (asterisk)	Matches any number of characters	HE* matches all parts with a profile name that begins with the characters "HE". You can also this symbol at the beginning of a word: *BRAC*.
? (question mark)	Matches a single character	HE?400 matches parts with profile names such as HEA400, HEB400, and HEC400.
[] (square brackets)	Matches whatever is enclosed in the brackets	L[78]X4X1/2 matches parts with the profile names L7X4X1/2 and L8X4X1/2.

NOTE The characters * and ? may also be used in object names in Tekla Structures. If the object name you want to filter contains * or ?, you need to enclose these characters in square brackets. For example, to find the profile P100*10, enter P100[*]10 in the filter field.

See also

[Filter objects \(page 155\)](#)

Examples of filters

Here are some examples of filters that you can create. The same filtering techniques can be used for view, selection, and drawing filters.

Filter parts based on their name

Create a filter that only shows parts with a certain name.

1. [Create a new view filter. \(page 158\)](#)
2. Click **Add row** three times to add three filter rules.
3. In the first filter rule, define that the object type must be part:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Part**.
 - e. In the **And/Or** list, select **And**.

4. In the second and third filter rules, define that the part name must be BEAM or COLUMN:
 - a. In the **Category** list, select **Part**.
 - b. In the **Property** list, select **Name**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** box, enter the part names, BEAM and COLUMN.
 - e. In the **And/Or** list, select **Or**.
5. Include the second and third filter rule in brackets. The filter is now looking for parts that are named either BEAM or COLUMN.
6. Enter a unique name in the box next to the **Save as** button.
7. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Object	Object type	Equals	<input checked="" type="checkbox"/> Part	-	And
<input checked="" type="checkbox"/>	(Part	Name	Equals	COLUMN	-	Or
<input checked="" type="checkbox"/>	-	Part	Name	Equals	BEAM)	

Filter main parts

Create a filter that selects the main parts only.

1. [Create a new selection filter. \(page 158\)](#)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object type must be part:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Part**.
 - e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that you only want to include main parts:
 - a. In the **Category** list, select **Part**.
 - b. In the **Property** list, select **Primary part**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** box, enter 1.
 In this context, 1 means the main parts, and 0 would mean the secondary parts.
5. Enter a unique name in the box next to the **Save as** button.

6. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Object	Object type	Equals	<input checked="" type="checkbox"/> Part	-	And
<input checked="" type="checkbox"/>	-	Part	Primary part	Equals	1	-	And

Filter bolts based on their size

Create a filter that only shows bolts of certain sizes.

1. [Create a new view filter. \(page 158\)](#)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object type must be bolt:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Bolt group**.
 - e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the bolt size must be 12.00 or 16.00:
 - a. In the **Category** list, select **Bolt**.
 - b. In the **Property** list, select **Size**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** box, enter the bolt sizes, 12.00 and 16.00.
Separate the strings with a blank space.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Object	Object type	Equals	Bolt group	-	And
<input checked="" type="checkbox"/>	-	Bolt	Size	Equals	12.00 16.00	-	And

Filter parts based on their assembly type

Create a filter based on assembly types. For example, you can create a filter that only shows cast-in-place and precast columns. Steel columns and any other columns or parts are hidden. The same filtering technique can be used for steel, concrete, timber, and miscellaneous parts.

1. [Create a new filter. \(page 158\)](#)
2. Click **Add row** four times to add four filter rules.
3. In the first filter rule, define that the object type must be part:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Part**.
 - e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the part name must be COLUMN:
 - a. In the **Category** list, select **Part**.
 - b. In the **Property** list, select **Name**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** box, enter the part name, COLUMN.
 - e. In the **And/Or** list, select **And**.
5. Include the first and second filter rule in brackets.
6. In the third and fourth filter rules, define that the assembly type must be precast or cast in place:
 - a. In the **Category** list, select **Assembly**.
 - b. In the **Property** list, select **Assembly type**.
 - c. In the **Value** box, enter the assembly types, 0 and 1.

Value	Assembly type
0	precast
1	cast in place
2	steel
3	timber
6	miscellaneous

- d. In the **And/Or** list, select **Or**.
7. Include the third and fourth filter rule in brackets. The filter is now looking for concrete parts that are named COLUMN.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	(Object	Object type	Equals	<input checked="" type="checkbox"/> Part	-	And
<input checked="" type="checkbox"/>	-	Part	Name	Equals	COLUMN)	And
<input checked="" type="checkbox"/>	(Assembly	Assembly type	Equals	1	-	Or
<input checked="" type="checkbox"/>	-	Assembly	Assembly type	Equals	0)	

Filter sub-assemblies

Create a filter that only selects parts that belong to a sub-assembly.

1. [Create a selection filter. \(page 158\)](#)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select `ASSEMBLY.HIERARCHY_LEVEL`.
5. In the **Condition** list, select **Does not equal**.
6. In the **Value** box, enter 0.

In this context, 0 means that the part does not belong to any sub-assembly, and 1 would mean that the part does. The filter will show only those parts whose value is **not** 0.

7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Template	ASSEMBLY.HIERARCHY_LEVEL	Does not equal	0	-	And

Filter reference model objects

Create a filter based on reference model object properties.

1. [Create an empty view or selection filter. \(page 158\)](#)
2. Click **Add row** to add a new filter rule.
3. In the **Category** list, select **Template**.
4. In the **Property** list, select the desired template attribute [or enter one of your own \(page 169\)](#).

TIP To find the attribute name used by the reference model, select a reference model object, right-click and select one of the **Inquire** commands. Find the property name in the **Inquire object** dialog box and copy it.

5. Add the prefix `EXTERNAL.` in front of the template attribute name.
6. In the **Condition** list, select **Equals**.
7. In the **Value** box, enter the desired value or click **Select from model...** to select the object in the model.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as**.

	(Category	Property	Condition	Value
<input checked="" type="checkbox"/>		Template	EXTERNAL.MATERIAL->NAME	Equals	Insulation

Filter parts within component

Create a filter that selects all parts within a component.

1. [Create an empty selection filter. \(page 158\)](#)
2. Click **Add row** twice to add two filter rules.
3. In the first filter rule, define that the object must be a component:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Is component**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Yes**.
 - e. In the **And/Or** list, select **And**.
4. In the second filter rule, define that the object type must be part:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Part**.
5. Enter a unique name in the box next to the **Save as** button.
6. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Object	Is component	Equals	Yes	-	And
<input checked="" type="checkbox"/>	-	Object	Object type	Equals	Part	-	And

Filter reinforcement in pour units based on pour object type

Create a filter that only shows all reinforcement that belongs to pour units with a pour object of a certain type.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to TRUE.
2. [Calculate the pour units. \(page 410\)](#)
3. [Create a new view filter. \(page 158\)](#)
4. Click **Add row** twice to add two filter rules.
5. In the first filter rule, define the pour object type.
 - a. In the **Category** list, select **Template**.
 - b. In the **Property** box, enter POUR_UNIT.POUR_OBJECT.POUR_TYPE.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** box, enter the pour object type, for example MyType, or click **Select from model...** to select the object in the model.
 - e. In the **And/Or** list, select **And**.
6. In the second filter rule, define that the object type must be reinforcement:
 - a. In the **Category** list, select **Object**.
 - b. In the **Property** list, select **Object type**.
 - c. In the **Condition** list, select **Equals**.
 - d. In the **Value** list, select **Reinforcing bar**.
7. Enter a unique name in the box next to the **Save as** button.
8. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Template	POUR_UNIT.POUR_OBJECT.POUR_TYPE	Equals	MyType	-	And
<input checked="" type="checkbox"/>	-	Object	Object type	Equals	↳ Reinforcing bar	-	

Filter all contents of a pour unit

Create a filter that selects all contents of a pour unit with a certain name.

1. Ensure that you have XS_ENABLE_POUR_MANAGEMENT set to TRUE.
2. [Calculate the pour units. \(page 410\)](#)
3. [Create a selection filter. \(page 158\)](#)
4. Click **Add row** to add a new filter rule.
5. In the **Category** list, select **Pour unit**.
6. In the **Property** list, select **Name**.
7. In the **Condition** list, select **Equals**.
8. In the **Value** box, enter the pour unit name, for example MyName.
9. Enter a unique name in the box next to the **Save as** button.

10. Click **Save as**.

-	(Category	Property	Condition	Value)	And/Or
<input checked="" type="checkbox"/>	-	Pour unit	Name	Equals	MyName	-	

Copy and remove filters

You can copy custom filters to another model by manually copying filter files to the `attributes` folder under the desired model folder. You can also manually remove unneeded filters from the same folder. If you want to make a filter available in all models, copy the file to your company's project or firm folder.

Copy a filter to another model

1. Select the filter you want to copy.

The filters you have created are located in the `attributes` folder under the current model folder. You can recognize different filter types based on their file name extension:

File name extension	Filter type
.VObjGrp	Model view filter
.SObjGrp	Model selection filter
.PObjGrp	Object group filter
.vf	Drawing view filter
.vnf	Drawing view level neighbor part filter
.wdf	Single-part drawing filter
.wdnf	Single-part drawing neighbor part filter
.adf	Assembly drawing filter
.adnf	Assembly drawing neighbor part filter
.cuf	Cast unit drawing filter
.cunf	Cast unit drawing neighbor part filter
.gdf	General arrangement drawing filter
.gdnf	General arrangement drawing neighbor part filter
.dsf	Drawing selection filter

2. To make the filter available in another model, copy the file to the `attributes` folder of the destination model folder.

3. To make the filter available in all models, copy the file to your company's project or firm folder.
4. Restart Tekla Structures.

Remove a filter

1. Remove the filter file from the model's `attributes` folder.
2. Restart Tekla Structures.

Select values from the model

You can select object properties and dates directly from the model. This can be useful when creating view filters, selection filters, and object groups.

Before you start, create an empty view or selection filter, or an object group.

1. Create [an empty view or selection filter \(page 158\)](#), or [an object group \(page 591\)](#).
2. Click **Add row**.
3. Select options from the **Category** and **Property** lists.
4. In the **Value** list, select one of the options.

The availability of options depends on your selection in the **Property** list. You can select dates from the model only if the property is a date.

- a. To select an object property, click **Select from model...** and then select an object.
- b. To select a date, click **Select date...** to open the **Select Date** dialog box, and then select one of the options.

You can either select a date from the calendar, select the review date, or define the number of days before or after the review date. The review date is the same as **Review date** in the **Project Status Visualization** dialog box.

1.8 Customize the basic user interface elements

You can customize the basic user interface elements according to your needs.

For example, the following user interface elements are customizable:

- keyboard shortcuts
- ribbon
- property pane

- [toolbars \(page 225\)](#)
- contextual toolbar

The customizations can be distributed to other users in the company.

Customize the keyboard shortcuts

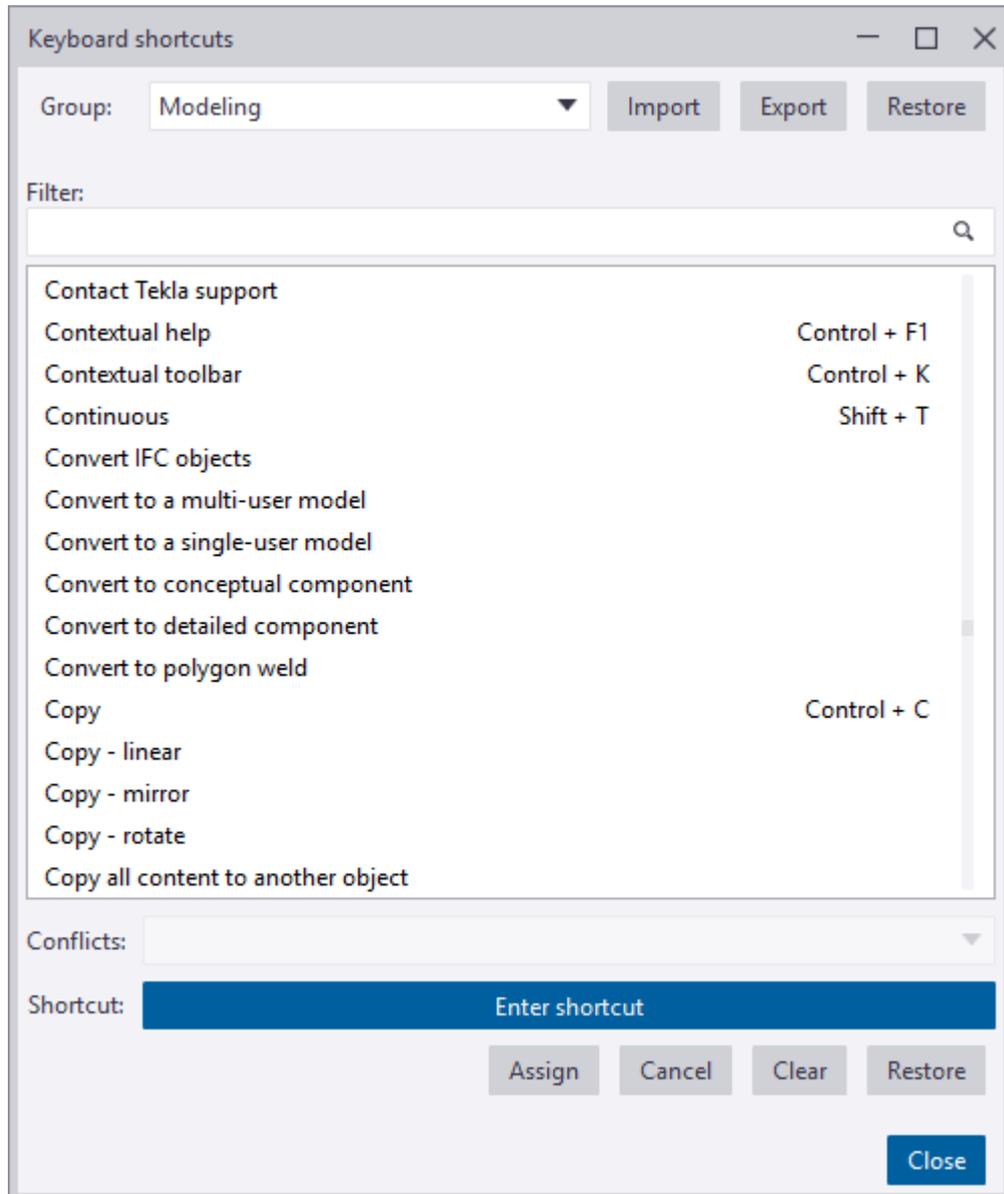
In the **Keyboard shortcuts** dialog box, you can view a list of all shortcuts available in Tekla Structures. You can define new keyboard shortcuts and remove existing ones. After customization, you can export the keyboard shortcuts and share them with your co-workers.

Define new keyboard shortcuts

You can assign keyboard shortcuts to any command, macro, or component. You can even change the default keyboard shortcuts, if needed.

1. On the **File** menu, click **Settings --> Keyboard shortcuts**.

The **Keyboard shortcuts** dialog box opens.



2. In the **Group** list, select the shortcut group you want to modify.
A list of commands and shortcuts appears.
3. If you want to search for a particular command or keyboard shortcut, enter some text in the **Filter** box.

For example:

- Type `grid` to only see the commands whose name contains the word "grid".
- Type `+` to get a list of shortcuts that consist of two parts (such as **Ctrl +S**).

- Type ", " to get a list of shortcuts that consist of two consecutive keys (such as **M, N**).
4. Select a command from the list.
 5. Click **Enter shortcut**.
 6. On the keyboard, enter the combination of keys you would like to use as the shortcut.
 7. Check the **Conflicts** box to see if the keyboard shortcut is already assigned to another command.
If the shortcut is already in use, enter a different combination of keys.

NOTE If you reassign a keyboard shortcut that is already used, it will no longer be associated with the command it was originally assigned to.

8. Click **Assign** to save the keyboard shortcut.

Clear and reset shortcuts

You can remove any existing shortcut. You can also reset all shortcuts back to the defaults.

1. On the **File** menu, click **Settings --> Keyboard shortcuts**.
2. To remove a keyboard shortcut, select the command from the list and click **Clear**.
3. To reset all the keyboard shortcuts to the defaults, click the **Restore** button.

Export keyboard shortcuts

You can export your customized keyboard shortcuts and share them with your co-workers.

1. On the **File** menu, click **Settings --> Keyboard shortcuts**.
2. Click **Export**.
3. Enter a file name and location.
4. Click **Save** to export the keyboard shortcuts.
5. To share your keyboard shortcuts with other users, send them the exported file.

Import keyboard shortcuts

You can import keyboard shortcuts from a file. Use this method to import keyboard shortcuts from Tekla Structures 2016 or newer.

1. On the **File** menu, click **Settings --> Keyboard shortcuts**.

2. Click **Import**.
3. Browse for the shortcuts file you want to import. For example, `.. \Users \<user> \AppData \Local \Trimble \Tekla Structures \<version> \Settings \KeyboardShortcuts_4.xml`.
4. Click **Open** to import the keyboard shortcuts.

Customize the ribbon

With **Ribbon editor** you can customize the ribbon according to your needs. You can change the size and shape of any button, for example. You can create user-defined commands, and bring your favorite components and extensions to the ribbon for an easy access.

To open **Ribbon editor**, click **File menu** --> **Settings** --> **Customize** --> **Ribbon** .

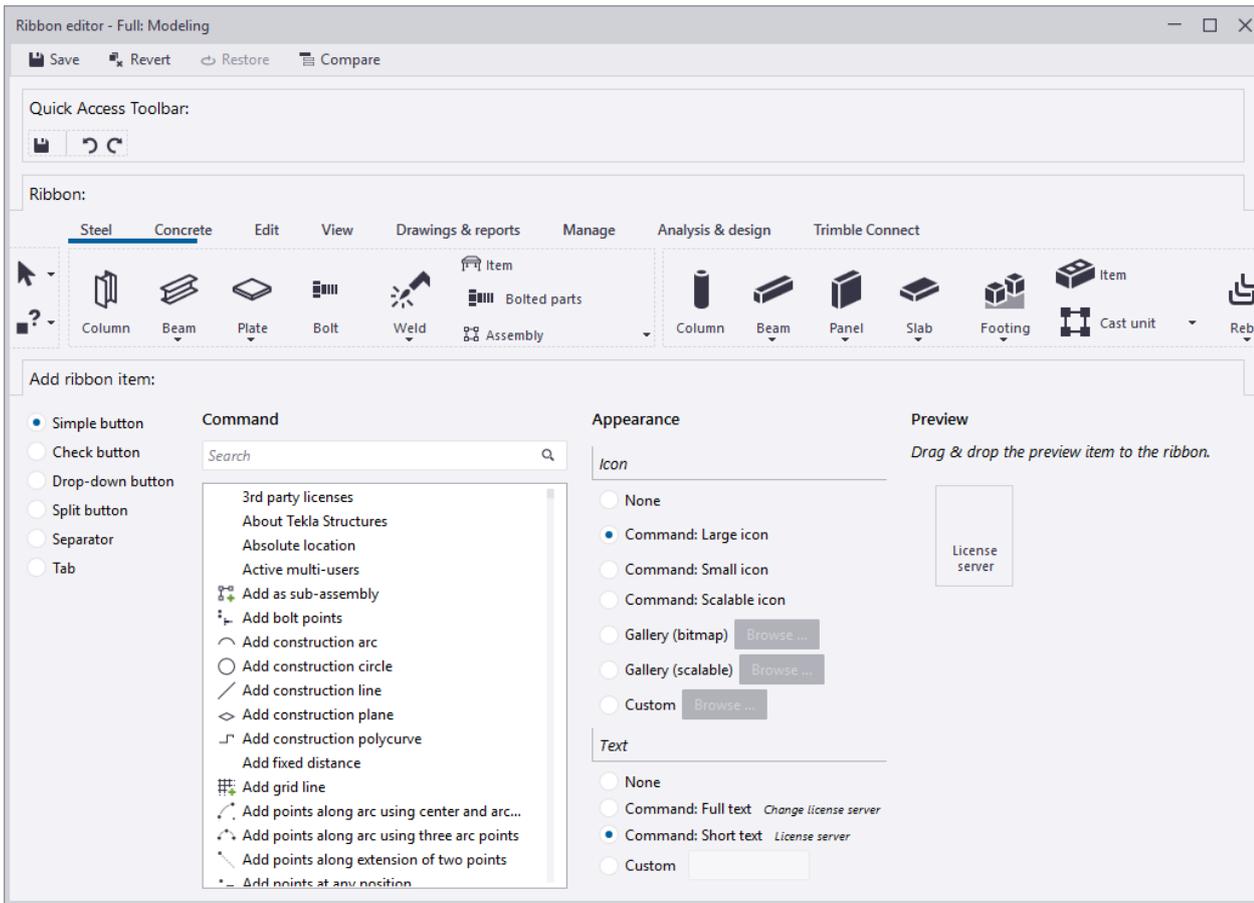
With **Ribbon editor**, you can

- add new buttons to the ribbon
- move existing buttons on the ribbon
- resize the buttons on the ribbon
- change the icons and texts of the buttons
- remove buttons you do not need
- create new commands and add buttons for them
- add separator bars to the ribbon
- add new tabs

NOTE If you want to customize the modeling ribbon, open **Ribbon editor** in the modeling mode.

If you want to customize the drawing ribbon, open **Ribbon editor** in the drawing mode.

You can only customize the ribbons that are available in your configuration.



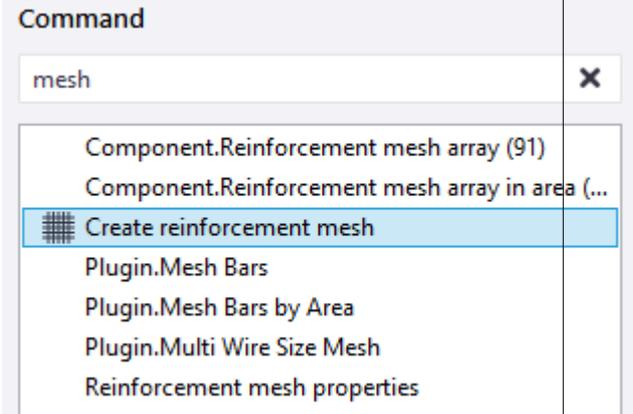
The customized ribbons are saved to `.. \Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UI\Ribbons` folder. If you cannot find the folder, ensure that you are able to view the hidden files and folders on your computer.

Company administrators can distribute the customized ribbons or tabs to the whole organization, in the same way as customized property pane layouts.

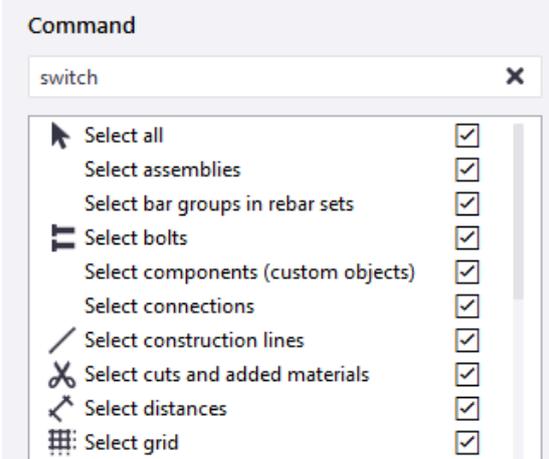
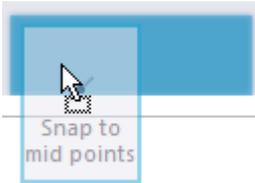
Add a button to the ribbon

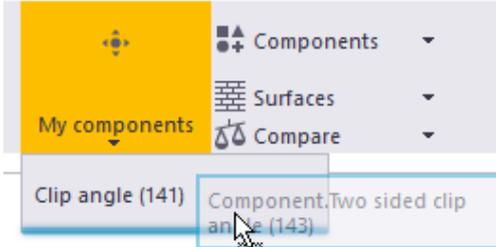
You can add buttons simply by selecting a button type and its appearance and then dragging the command to the ribbon or to the **Quick Access Toolbar**.

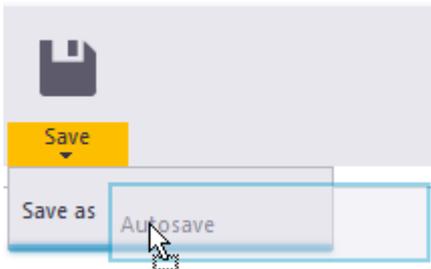
To	Do this
Add a button for a single command	<ol style="list-style-type: none"> 1. In the Add ribbon item list, select Simple button. 2. In the Command list, select the command you want to add to the ribbon.

To	Do this
	<p>You can also add components, macros, and extensions. Browse through the list or use the Search box to filter content. For example, type <code>mesh</code> to find the Create reinforcement mesh command and other mesh related components:</p>  <p>3. In the Appearance list:</p> <ul style="list-style-type: none"> • Select whether the button has an icon. Select the size of the icon, or whether you want to use a scalable icon, or browse for the icon image. • Select whether the button has any text.

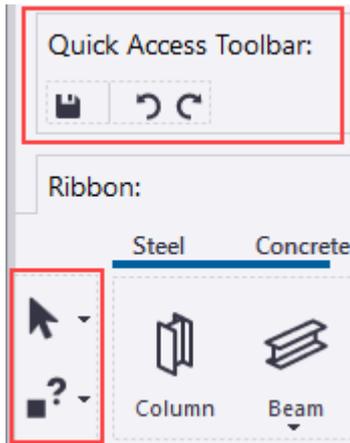
To	Do this
	<div data-bbox="863 271 1362 1016" data-label="Image"> </div> <p data-bbox="759 1039 1342 1142">4. Preview shows what the button looks like. Modify the button appearance, if needed.</p> <p data-bbox="759 1160 1375 1279">5. Drag the button to the ribbon. The blue color indicates the place where the button will be inserted.</p> <div data-bbox="820 1301 1120 1563" data-label="Image"> </div>
<p data-bbox="309 1585 737 1682">Add a toggle button that switches a particular command on or off</p>	<p data-bbox="759 1585 1367 1753">Use this to add any switch from the File menu --> Settings --> Switches to the ribbon, for example. You can also add single snap switches and select switches to the ribbon.</p> <ol data-bbox="759 1771 1375 1919" style="list-style-type: none"> <li data-bbox="759 1771 1289 1839">1. In the Add ribbon item list, select Check button. <li data-bbox="759 1856 1375 1919">2. In the Command list, select a command that can be switched on or off.

To	Do this
	<p>Commands that can be switched on or off have a check box next to them.</p>  <ol style="list-style-type: none"> In the Appearance list: <ul style="list-style-type: none"> Select whether the button has an icon. Select the size of the icon, or whether you want to use a scalable icon, or browse for the icon image. Select whether the button has any text. Preview shows what the button looks like. Modify the button appearance, if needed. Drag the button to the ribbon. The blue color indicates the place where the button will be inserted. 
Add a drop-down button with a group of commands underneath it	<ol style="list-style-type: none"> In the Add ribbon item list, select Drop-down button. In the Appearance list: <ul style="list-style-type: none"> Select whether the button has an icon. Browse for the icon image. Select whether the button has any text.

To	Do this
	<p>3. Preview shows what the button looks like. Modify the button appearance, if needed.</p> <p>4. Drag the button to the ribbon.</p> <p>The button is now an empty placeholder for single commands. You need to add commands to the drop-down button to make it work.</p> <ol style="list-style-type: none"> In the Add ribbon item list, select Simple button. In the Command list, select the command you want to add to the drop-down list. In the Appearance list, set the appearance of the button. Drag the button to the drop-down list. <p>The blue color indicates the place where the button will be inserted. If you hover over a down arrow, a list will open and you can drag commands to the list. The list will remain open until you click the down arrow again.</p>  <p>e. Add as many commands as needed to the drop-down button.</p>
Add a button for a single command, plus a drop-down button with a group of commands underneath it	<ol style="list-style-type: none"> In the Add ribbon item list, select Split button. In the Command list, select the command you want to add to the ribbon as the main button for the whole split button. In the Appearance list:

To	Do this
	<ul style="list-style-type: none"> • Select whether the button has an icon. Select the size of the icon, or whether you want to use a scalable icon, or browse for the icon image. • Select whether the button has any text. <p>4. Preview shows what the button looks like. Modify the button appearance, if needed.</p> <p>5. Drag the button to the ribbon.</p> <p>The button now has a single command. You need to add commands to the drop-down list.</p> <ol style="list-style-type: none"> a. In the Add ribbon item list, select Simple button. b. In the Command list, select the command you want to add to the drop-down list. c. In the Appearance list, set the appearance of the button. d. Drag the button to the drop-down list. <p>The blue color indicates the place where the button will be inserted. If you hover over a down arrow, a list will open and you can drag commands to the list. The list will remain open until you click the down arrow again.</p>  <ol style="list-style-type: none"> e. Add as many commands as needed to the drop-down button.

You can also drag commands to the **Quick Access Toolbar**, which is located above the ribbon, or to the fixed container on the left side of the ribbon:

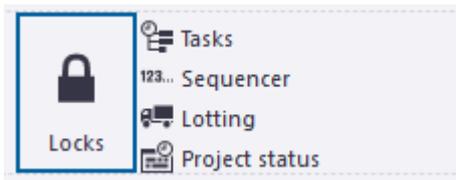


Move a button

You can rearrange buttons on the ribbon. Note that you cannot move drop-down buttons underneath each other.

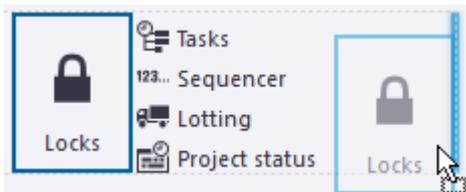
1. Select the button you want to move.

The button becomes highlighted:



2. Drag the button to a new location.

The blue color indicates the place where the button will be inserted. For example:



Resize a button

You can change the size of existing buttons.

1. Select the button you want to resize:



2. Move the mouse pointer over any side or corner of the button to display a white arrow symbol:



3. Drag with the arrow to define a new size:



The size of the button changes accordingly. The other buttons are automatically moved forward on the ribbon, if needed.



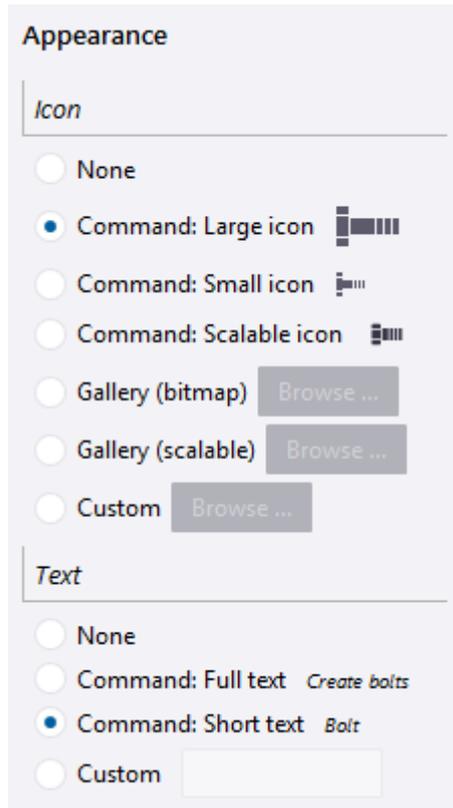
4. Double-click the button to expand it.
The button now fully occupies the empty space around it:



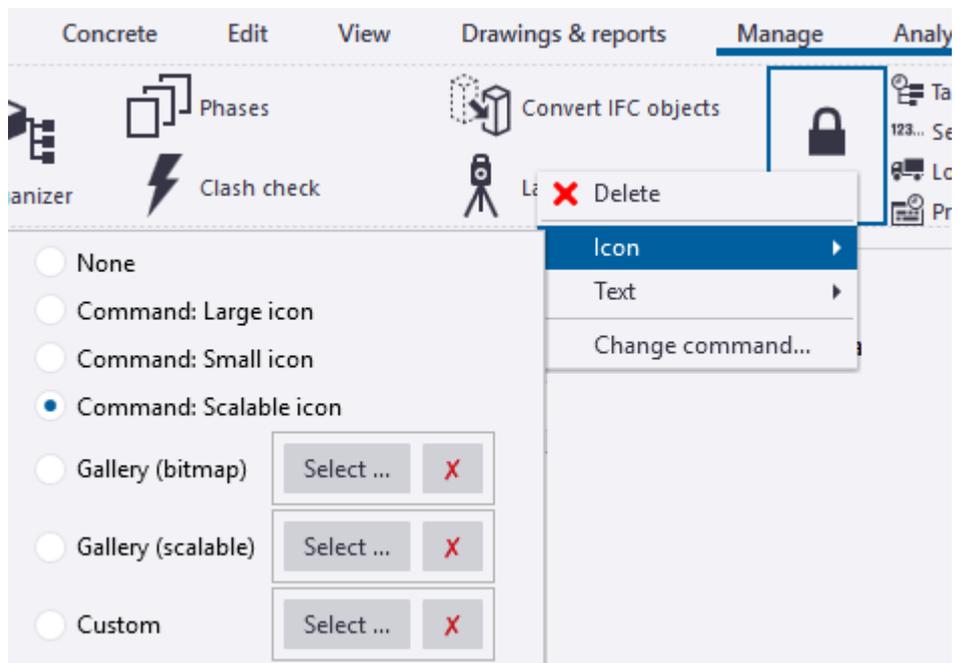
Change the appearance of a button

You can change the appearance of any button.

1. Select the button you want to modify.
 - If you are adding a new button: select the command in the **Command** list.
The current properties of the button are shown in the **Appearance** list.



- If the button already exists on the ribbon: right-click the selected button on the ribbon.



2. To change the icon, select one of the options:
 - a. **None**: no icon is used for the button

- b. **Command: Large icon:** the default large icon (32x32) is used
 - c. **Command: Small icon:** the default small icon (16x16) is used
 - d. **Command: Scalable icon:** the scalable vector icon is used
 - e. **Gallery (bitmap):** select a large or small bitmap icon from the Tekla Structures icon gallery
 - f. **Gallery (scalable):** select a scalable icon from the Tekla Structures icon gallery
 - g. **Custom:** define a custom icon by selecting a suitable image file. The recommended size is 32x32 pixels for large buttons and 16x16 pixels for small buttons. If you have problems with your custom image not appearing the right size, check the DPI setting of the image file. A DPI of 96 is recommended.
3. To change the name, select one of the options:
- **None:** no name is used for the button
 - **Command: Full text:** the default full version of the name is used
 - **Command: Short text:** the default short version of the name is used
 - **Custom:** enter a custom name for the button

Create a user-defined command with Command editor

You can create user-defined commands and link them to any file or URL. Create the user-defined commands with **Command editor**.

1. Click **File menu --> Settings --> Customize --> User-defined commands** to open **Command editor**.
2. Click **New**.
3. Enter a unique ID for the command, and then click **OK**.

For example, create a link to the **Tekla Discussion Forum**. Enter `OpenTeklaDiscussionForum` as the ID of the command.

A new column with more properties appears.

Source	User
ID	<i>i</i> OpenTeklaDiscussionForum
Full name	<i>i</i> <input type="text" value="My command"/>
Short name	<i>i</i> <input type="text" value="Command"/>
Large icon	<input type="text" value="..."/> <input type="text" value="X"/>
Small icon	<input type="text" value="..."/> <input type="text" value="X"/>
Scalable icon	<i>i</i> <input type="text" value="..."/> <input type="text" value="X"/>
Tooltip	<input type="text"/>
Action	<i>i</i> <input type="text" value="File or URL"/>
Availability	<input checked="" type="checkbox"/> All <input checked="" type="checkbox"/> Modeling <input checked="" type="checkbox"/> Drawing <input checked="" type="checkbox"/> Import

- In the **Full name** and **Short name** boxes, enter a name for the command.

This name will be visible in the Tekla Structures user interface. You can define two alternative names: a full name and a short version. For example, enter `Tekla Discussion Forum` as the full name of the command, and `Forum` as the short version.

- In the **Large icon**, **Small icon**, and **Scalable icon** settings, select an icon for the command.

You can define three alternative icons: a large one and a small one, or a scalable vector icon.

You can use your own icon or select a suitable icon from the Tekla Structures icon gallery.

- In the **Tooltip** box enter a tooltip for the command.

For example, enter `Go to the Tekla discussion forum.`

- In the **Action** box define a file or URL.

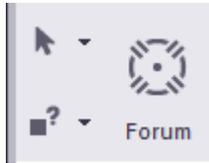
For example, enter `https://forum.tekla.com.`

- In **Availability**, select the mode where the command will be available.

- Click **Save** to save the new command.

- Go to **Ribbon editor**.

11. Select the button type.
12. In the **Command** list, search for the new command you created.
13. In the **Appearance** list, modify the appearance of the button, if needed.
14. Drag the new button to the ribbon.



15. To modify a user-defined command, right-click the command on the ribbon and edit the command properties just like for any other command.

Add a separator bar

You can add vertical and horizontal separator bars to divide buttons into smaller groups on the ribbon.

1. In the **Add ribbon item** list, select **Separator**.
2. In the **Appearance** list, select whether to add a horizontal or a vertical bar, and the thickness of the bar.

Preview shows what the separator bar looks like.

3. Drag the preview item to the ribbon.
4. To modify the orientation or the line thickness of the bar, right-click the bar on the tab and select **Orientation** or **Thickness**.
5. To delete the bar, select the bar on the ribbon and press **Delete** on your keyboard.

Alternatively, right-click the bar on the ribbon and select **Delete**.

Remove a button

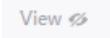
1. Select the button on the ribbon.
2. Press **Delete** on your keyboard.

Alternatively, right-click the button on the ribbon and select **Delete**.

Add, hide, and edit tabs

You can add, move and rename ribbon tabs, choose how they are aligned, and hide some tabs if you do not need them in your current project. For example, if you are only modeling steel parts, you can temporarily hide the **Concrete** tab.

To	Do this
Add a new tab	1. In the Add ribbon item list, select Tab .

To	Do this
	<ol style="list-style-type: none"> 2. Enter a name for the tab in the Text box. 3. Click Add tab to add it to the ribbon.
Change the order of tabs on the ribbon	Drag and drop the tab titles.
Select how the tabs are aligned	<p>Right-click in the ribbon area and select one of the Navigation mode options:</p> <ul style="list-style-type: none"> • Scroll visible: the ribbon movement is minimal when you switch between the tabs • Align to left: the icons start from the left side of the ribbon • Align to tab: the icons start from the left side of the current tab
Hide the tabs that you do not need in your current project	<ol style="list-style-type: none"> 1. Rest the mouse pointer on a tab title. A small eye symbol appears next to the tab title:  2. Click the eye symbol . <p>The eye symbol changes and the tab title becomes gray: </p> <p>The View tab is now hidden from the ribbon. If you slide the ribbon, hidden tabs appear as: </p> <p>To re-display the hidden tab, click the eye symbol again.</p>
Rename a tab	Right-click the tab and select Rename . Enter a new name for the tab.
Remove a tab	<p>Select the tab and press Delete.</p> <p>Alternatively, right-click the tab and select Delete.</p>

Save the ribbon

When you are happy with the changes, save the customized ribbon.

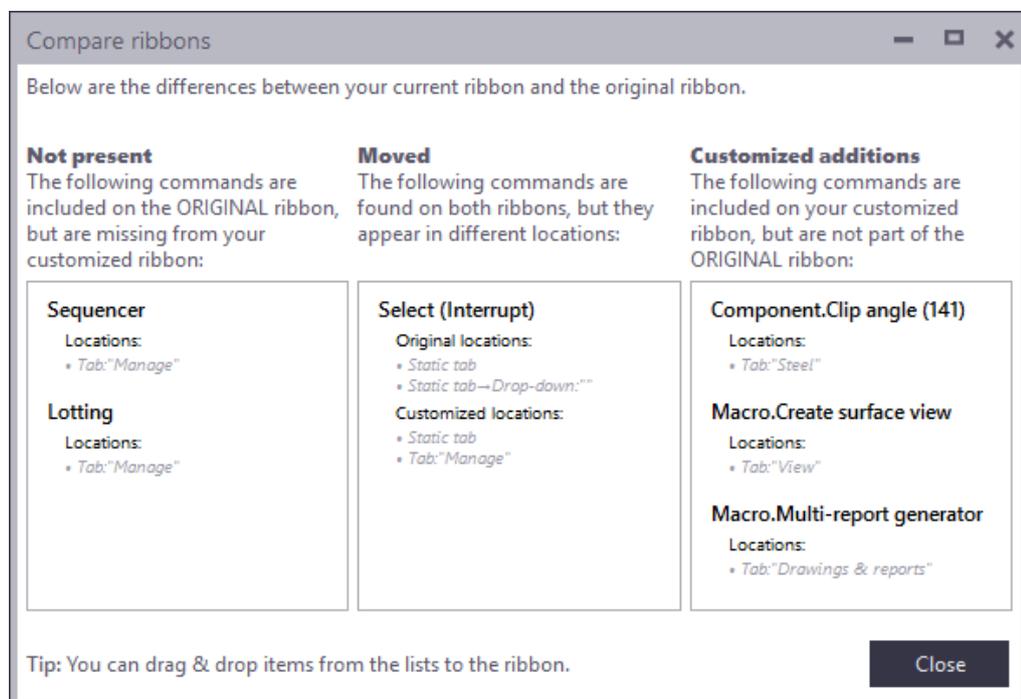
1. In **Ribbon editor**, click the **Save** button.
2. When you return to Tekla Structures and the program asks if you wish to load the new ribbon, click **Yes**. The ribbon becomes updated with the changes you made.

Check the changes

You can compare the original ribbon with the changes you have made. You can check what has been added and removed, and what has been moved to different tabs.

1. Save the customized ribbon, if you have not already done so.
2. Click the **Compare** button.
3. In the **Compare ribbons** dialog box, check the changes you have made.

For example:



- **Not present:** these commands have been removed
- **Moved:** these commands have been moved to a new place
- **Customized additions:** these commands have been added

NOTE Original ribbon refers to the ribbon file that came with the Tekla Structures installation for your current configuration.

4. If you have removed a command that you would like to get back, drag it from the **Compare ribbons** dialog box to the ribbon.

5. When you are finished, click **Close**.

Back up and restore ribbons

You can restore the default Tekla Structures ribbons at any time. Before restoring the default settings, make sure to save a backup copy of your customized ribbon, because the customizations will be permanently deleted. You can use the backup file to take your customized ribbon back into use, to copy the ribbon settings to another computer, or to share the customized ribbon with your co-workers.

1. To save a backup copy of the customized ribbon:
 - a. In **Ribbon editor**, click the **Save** button.
 - b. Go to the `.. \Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UI\Ribbons` folder.
 - c. Make a copy of the desired ribbon file and save it in another folder.
The ribbons are named according to the Tekla Structures configurations. For example, in the **Full** configuration, the name of the **Modeling** ribbon file is `albl_up_Full--main_menu.xml`.
2. Click the **Restore** button to restore the default Tekla Structures modeling or drawing ribbon.
3. To take the customized ribbon back into use:
 - a. Copy the backup file back to the `.. \Users\<user>\AppData\Local\Trimble\Tekla Structures\<version>\UI\Ribbons` folder.
 - b. When you return to Tekla Structures and the program asks if you wish to load the new ribbon, click **Yes**. The ribbon becomes updated with the changes you made.

Customize the property pane layout

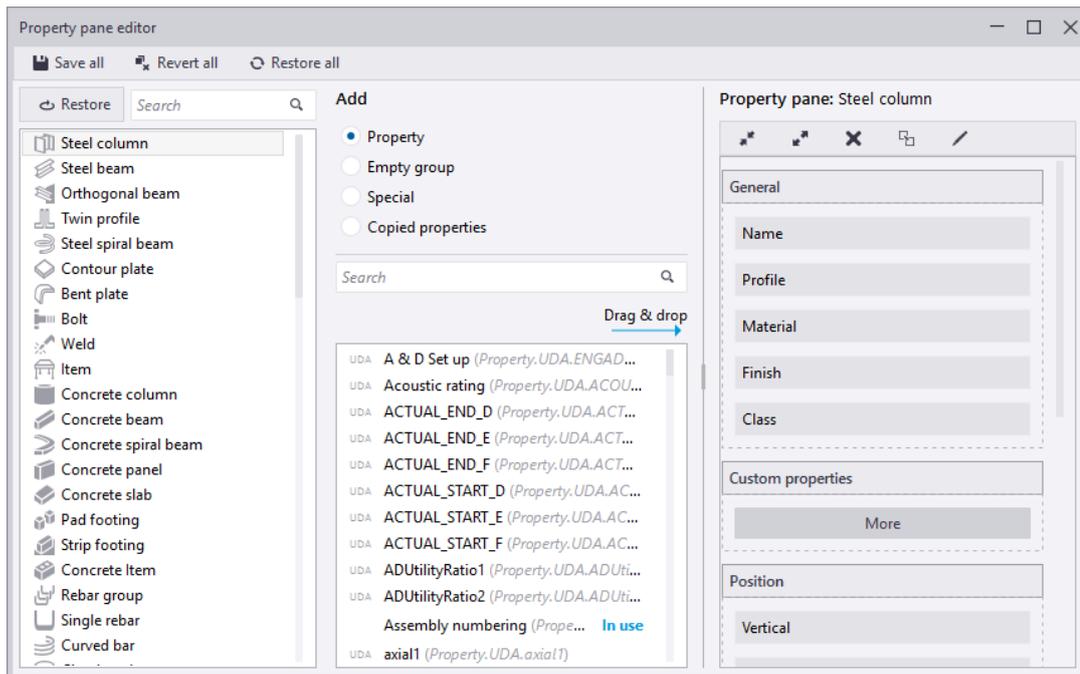
Tekla Structures shows the properties of most model objects in the property pane. You can customize the property pane to better suit your needs. You can select separately for each object type which properties you wish to see in the property pane. With **Property pane editor** you can show, hide and organize the settings in the property pane. You can also add your most needed user-defined attributes (UDAs) directly to the property pane.

To open **Property pane editor**, click **File menu --> Settings --> Customize --> Property pane** .

With **Property pane editor**, you can

- organize properties to a suitable order or to suitable groups
- remove properties you do not use or need

- create your own groups for properties that you find relevant, including user-defined attributes (UDAs)
- add properties, including user-defined attributes (UDAs), to an existing group
- create nested property groups
- rename properties or groups
- save the customized property pane layouts



The customized property pane layouts are saved to the `PropertyTemplates.xml` file in the `..Users\<user>\AppData\Local\Trimble\TeklaStructures\<version>\UI\PropertyTemplates\` folder. If you cannot find the folder, ensure that you are able to view the hidden files and folders on your computer.

Company administrators can distribute the customized property pane layouts to the whole organization, in the same way as customized ribbons or customized tabs.

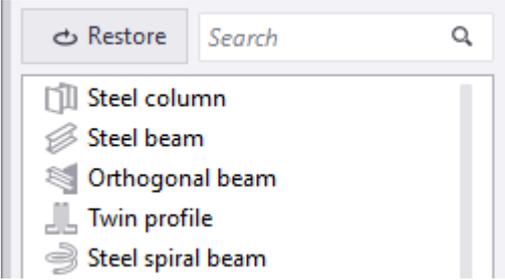
Add a property or a property group

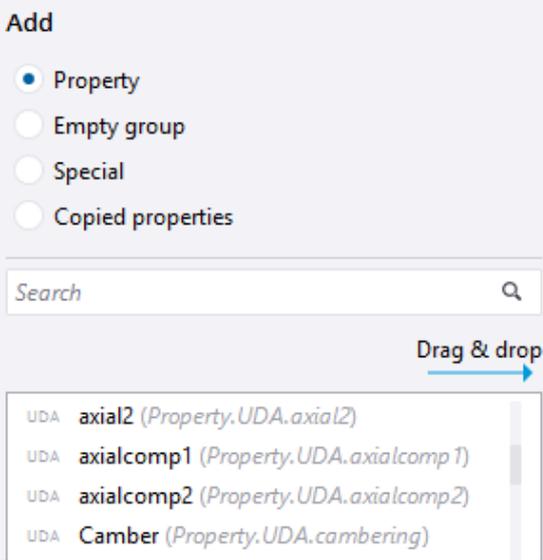
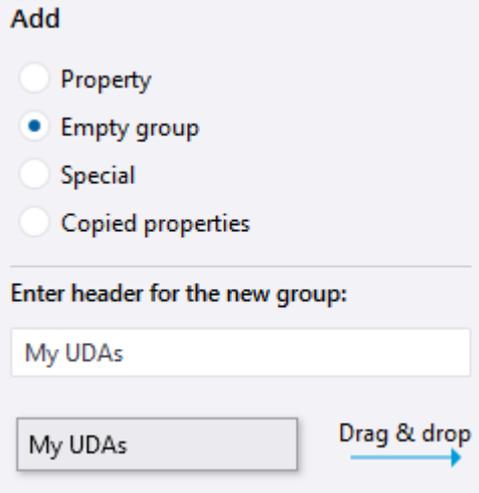
Property pane editor has three columns:

- *Object type list* on the left. The list shows all the objects which have a property pane. Select from the list the object type whose property pane layout you want to modify.

Each object type's tooltip shows the source from where the object type definition is loaded: the default source, your environment, or a user-defined customization.

- *Property list* in the middle. The list shows all the available properties and user-defined attributes (UDAs) for each object type. You can add these properties and UDAs to the property pane layout as regular properties. Properties which are already in use cannot be added for a second time but you can reorganize them in the property pane layout. Properties which are incompatible with the selected object type cannot be added.
- *Property pane layout* on the right. It shows the current layout of the property pane for the selected object type.

To	Do this
Select the object type whose property pane layout you want to modify	<p>In the object type list on the left, browse through the list or use the Search box to filter content.</p> 
Add a new property to the property pane layout	<ol style="list-style-type: none"> 1. In the middle column's Add section, select Property. 2. In the property list, select a property.

To	Do this
	<p>Use the Ctrl or Shift key to select multiple properties.</p>  <p>3. Drag the property to the property pane layout on the right.</p> <p>You can drag the property to any group in the property pane.</p>
<p>Add a new group to the property pane layout</p>	<p>1. In the middle column's Add section, select Empty group.</p> <p>2. Enter a title for the new group.</p>  <p>3. Drag the group template to the property pane layout on the right.</p>

To	Do this
	<p>You can create a new group, or insert a new group inside an existing group to create nested groups.</p> <p>You can reorganize the existing groups by dragging them.</p>
<p>Add the More button to the property pane layout</p>	<p>If you have accidentally removed the More button from the property pane layout, you can add it back to the property pane layout.</p> <ol style="list-style-type: none"> <li data-bbox="762 589 1315 658">1. In the middle column's Add section, select Special. <div data-bbox="823 678 1318 1077" style="border: 1px solid #ccc; padding: 10px; background-color: #f9f9f9;"> <p>Add</p> <p><input type="radio"/> Property</p> <p><input type="radio"/> Empty group</p> <p><input checked="" type="radio"/> Special</p> <p><input type="radio"/> Copied properties</p> <hr/> <p>Add back the 'More' button:</p> <div style="display: flex; align-items: center; gap: 10px;"> <div style="border: 1px solid #ccc; padding: 5px 15px; background-color: #e0e0e0;">More</div> → Drag & drop </div> </div> <ol style="list-style-type: none"> <li data-bbox="762 1104 1342 1173">2. Drag the More button to the property pane layout on the right.

Change the name of a property or a property group

To	Do this
<p>Rename a property or a property group</p>	<ol style="list-style-type: none"> <li data-bbox="675 1355 1362 1451">1. In the property pane layout, select the property or the property group that you want to rename. <li data-bbox="675 1473 863 1532">2. Click . <li data-bbox="675 1554 1358 1615">3. In the Rename dialog box, enter a new name and click OK. <p>Alternatively, right-click the name of the property or the property group and select Rename.</p>
<p>Restore the original name of a property or a property group</p>	<ol style="list-style-type: none"> <li data-bbox="675 1722 1347 1818">1. In the property pane layout, select the group or the property whose name you want to restore to the original. <li data-bbox="675 1841 1362 1870">2. Right-click and select Restore original name.

Copy properties from one object type to another object type

You can copy properties from one object type to another object type, for example, from steel beam to steel plate.

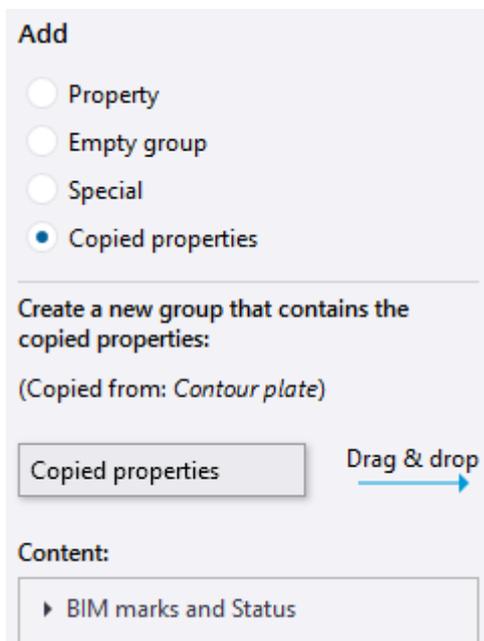
1. In the object type list on the left, select the object type from which you want to copy properties.
2. In the property pane layout on the right, select the properties you want to copy.

Use the **Ctrl** or **Shift** key to select multiple properties.

3. Click  to copy the selected properties.

Alternatively, right-click and select **Copy property**.

The copied properties are shown in the middle column, under the **Content** section.



4. In the object type list on the left, select the object type to which you want to copy the properties.
5. In the **Add** section, ensure that the **Copied properties** option is selected.
6. Drag the **Copied properties** box from the middle column to the property pane layout on the right.

The name of the copied group and the content of the group is shown in the middle column until you copy another group or close **Property pane editor**.

If you copy nested groups, all the nested groups inside the main group are copied, too.

NOTE Properties that are already in use cannot be added for a second time. If you copy properties that are already in use, the copied properties have the text **In use** in the **Content** section.

Properties that cannot be added to the selected object type have the text **Incompatible** in the **Content** section.

Remove a customization

To	Do this
Delete a property or a property group	<ol style="list-style-type: none">1. Select the property or the property group to be deleted. Use the Ctrl or Shift key to select multiple properties.2. Click . Alternatively, right-click a property or a property group, and select Delete.
Discard changes	Click the Revert all button to discard changes and to revert to the previous save.
Remove a single customization	Click the Restore button to remove the customization of a selected object type's property pane. Alternatively, right-click the selected object type and select Restore to default .
Remove all customizations	Click the Restore all button to remove the customization of all property pane layouts.

Save the changes

When you are happy with the changes, save the customized property pane layout.

1. Click the **Save all** button.
2. When you return to Tekla Structures, Tekla Structures asks if you wish to reload the changed property pane templates. Click **Yes** to take the customized property pane layout in use.

User-defined attributes (UDAs) in the customized property pane

In the property pane, the **More** button in the **Custom properties** section opens the user-defined attributes (UDAs) dialog box. If you customize the

property pane, you can add your most important UDAs directly to the property pane, so that you do not need to open separate UDA dialog boxes.

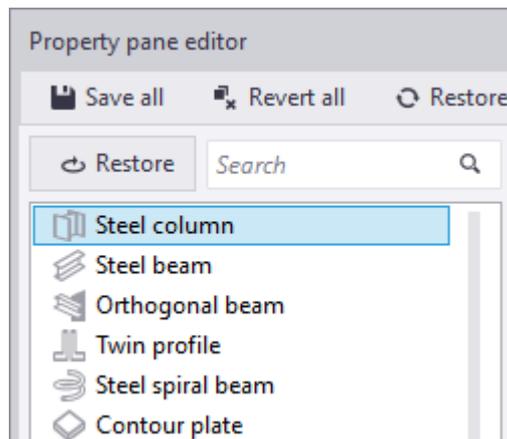
When you create or modify model objects, UDAs are automatically applied together with all the other object properties. UDAs are automatically applied regardless of them being in the property pane or in the UDA dialog boxes.

UDAs with the following unit types are supported and work correctly if they are added in the property pane: Option, String, Integer, Float, Date, Distance, Weight, Force, Moment, Angle, Factor, and Area. UDAs with other unit types need to be used through the UDA dialog boxes.

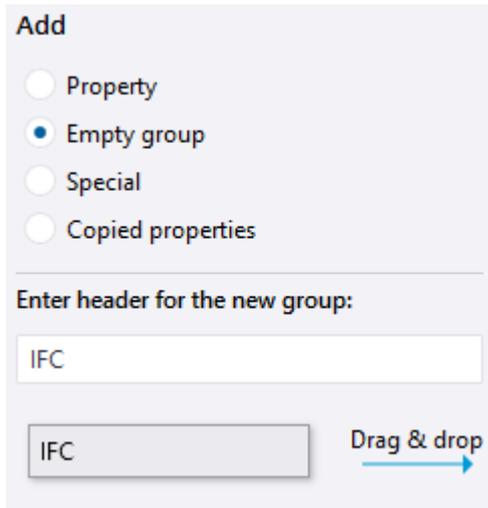
Example: How to add IFC related user-defined attributes to the property pane layout and copy them to another object type

This example shows how to add a group of IFC related user-defined attributes (UDAs) to the steel column property pane layout, and copy the group to the steel beam property pane layout.

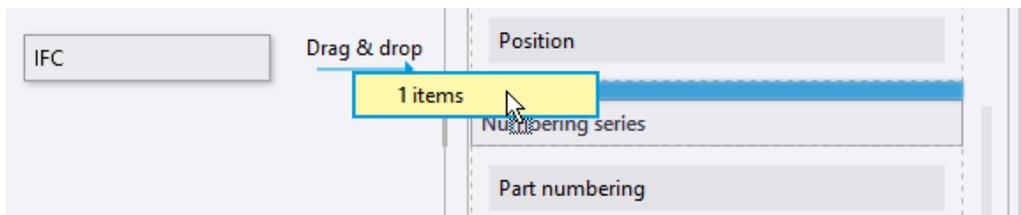
1. In the object type list, select **Steel column**.



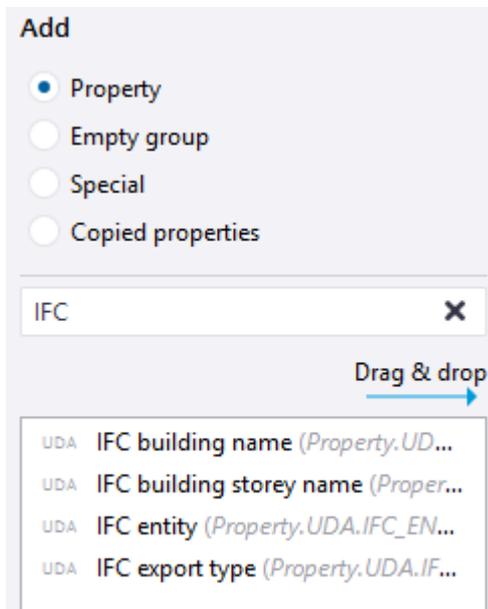
2. In the **Add** section, select **Empty group**. Enter IFC as the title for the new group.



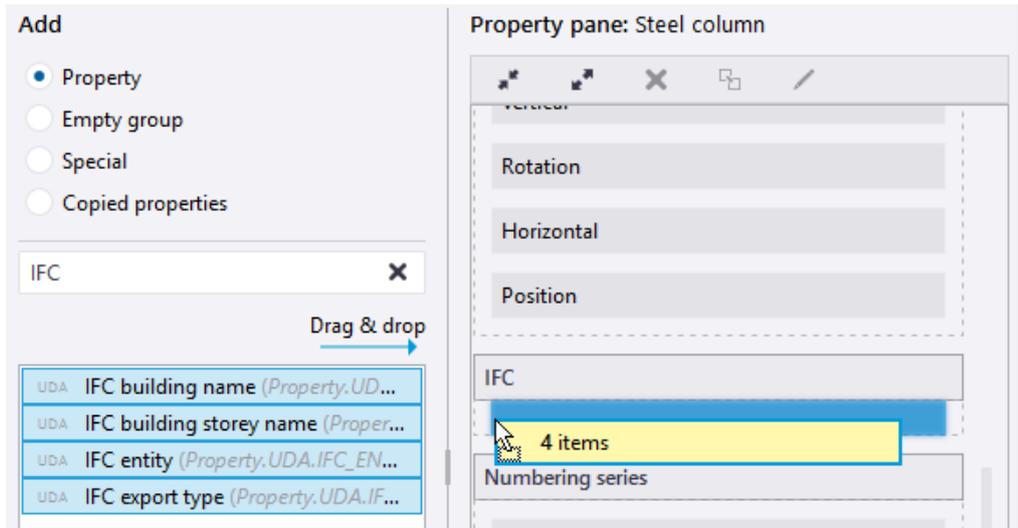
3. Drag the group template to the property pane layout on the right.



4. In the **Add** section, select **Property**. In the search, enter IFC to search the IFC related UDAs.

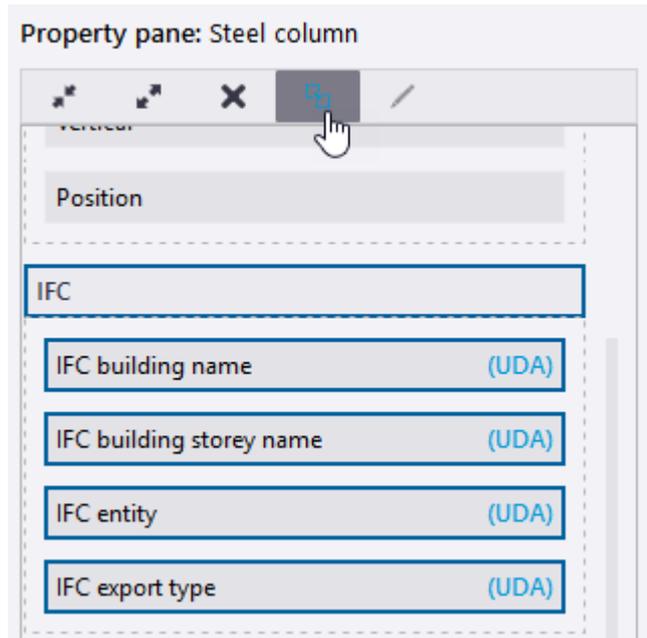


5. Select all the IFC UDAs and drag them to the group you created in the property pane layout.

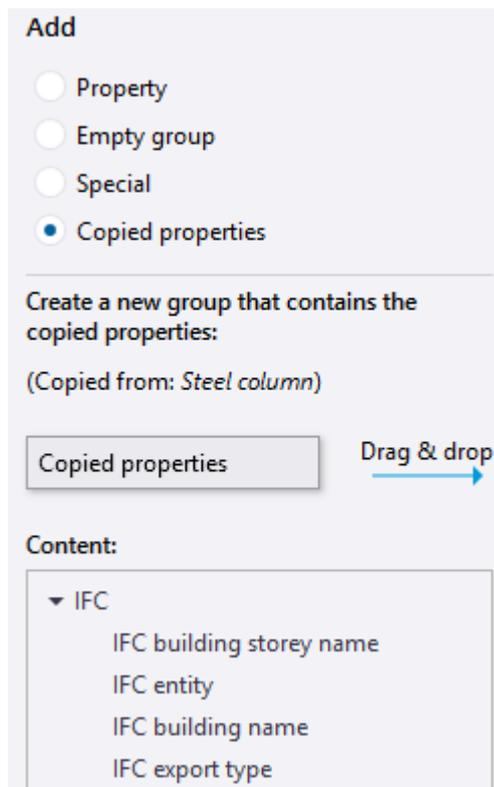


- When you have added the needed UDAs to the new group, copy the group so that you can add the group also to the steel beam property pane layout. Select the title of the group and click the **Copy selected items**

 button.

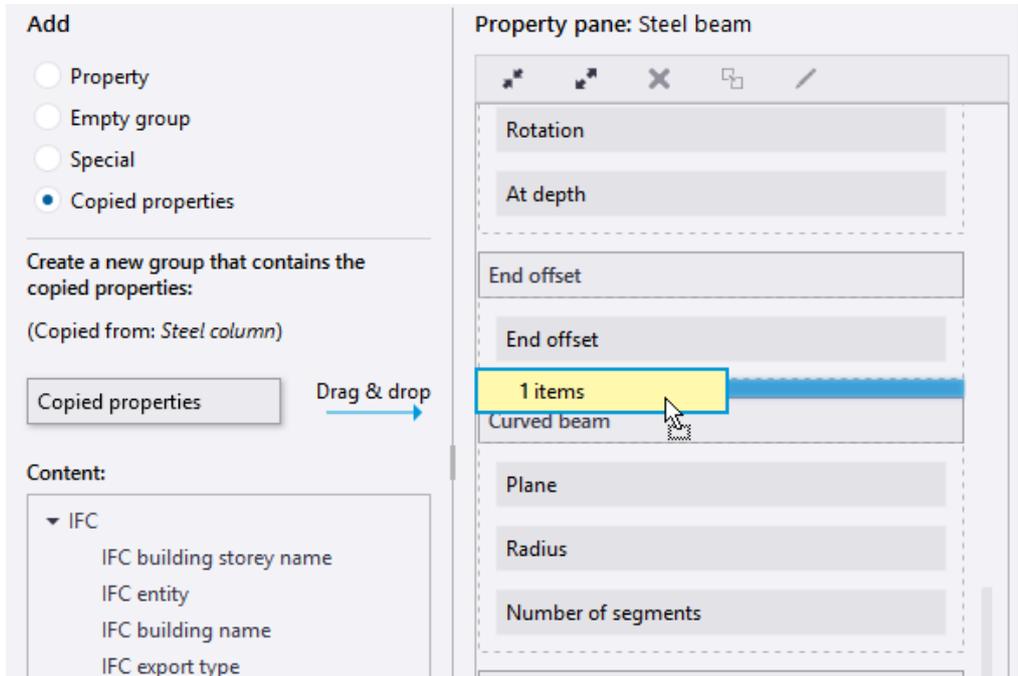


The name of the copied group and the content of the group is shown in the middle column. You can see that the properties are copied from steel column.

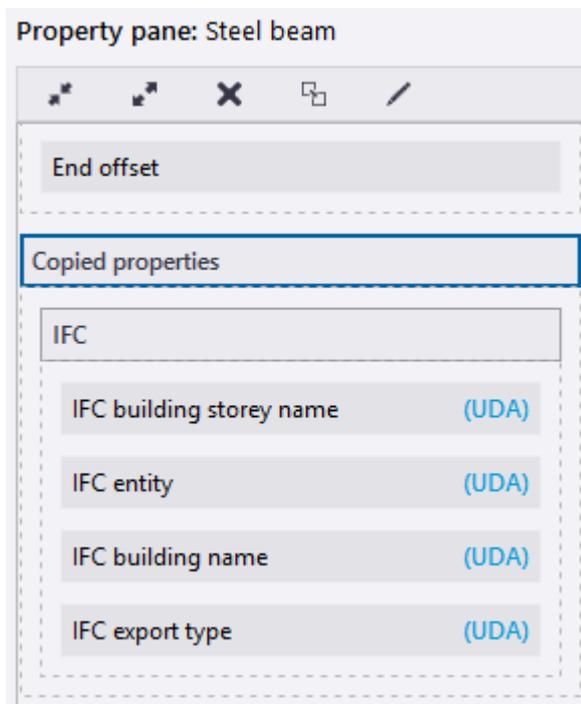


- To add the copied group in the steel beam property pane layout, select **Steel beam** in the object type list.

8. Drag the copied group from the middle column to the steel beam property pane layout on the right.



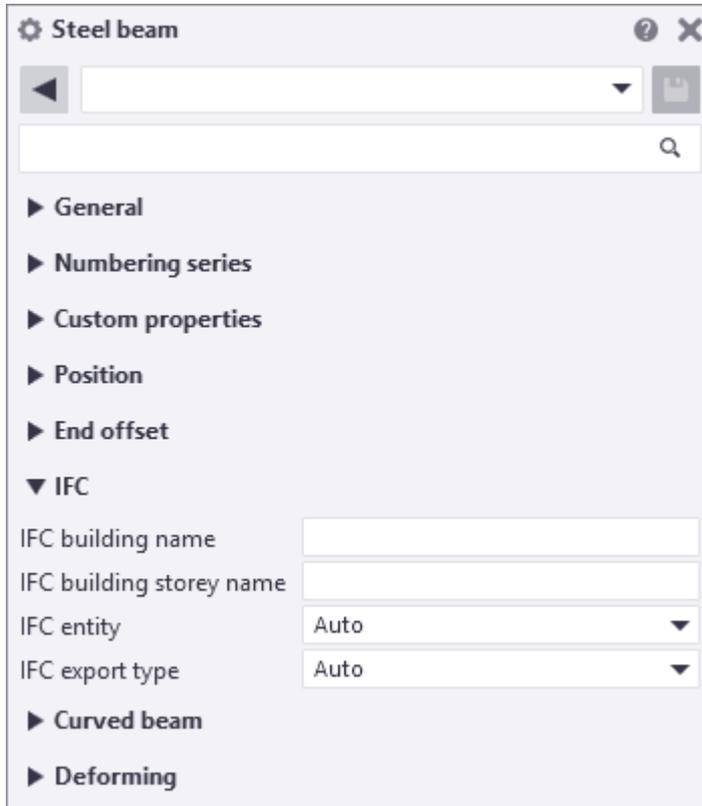
Now the IFC related UDAs are available both in the steel column property pane layout and in the steel beam property pane layout.



The copied properties are always inside the **Copied properties** group in the property pane layout. If you want to move the properties, you can

drag them outside the **Copied properties** group and then delete the empty group.

9. Click the **Save all** button to save the changes. When you return to Tekla Structures, Tekla Structures asks if you wish to reload the changed property pane templates. Click **Yes** to take the customized property pane layout in use.



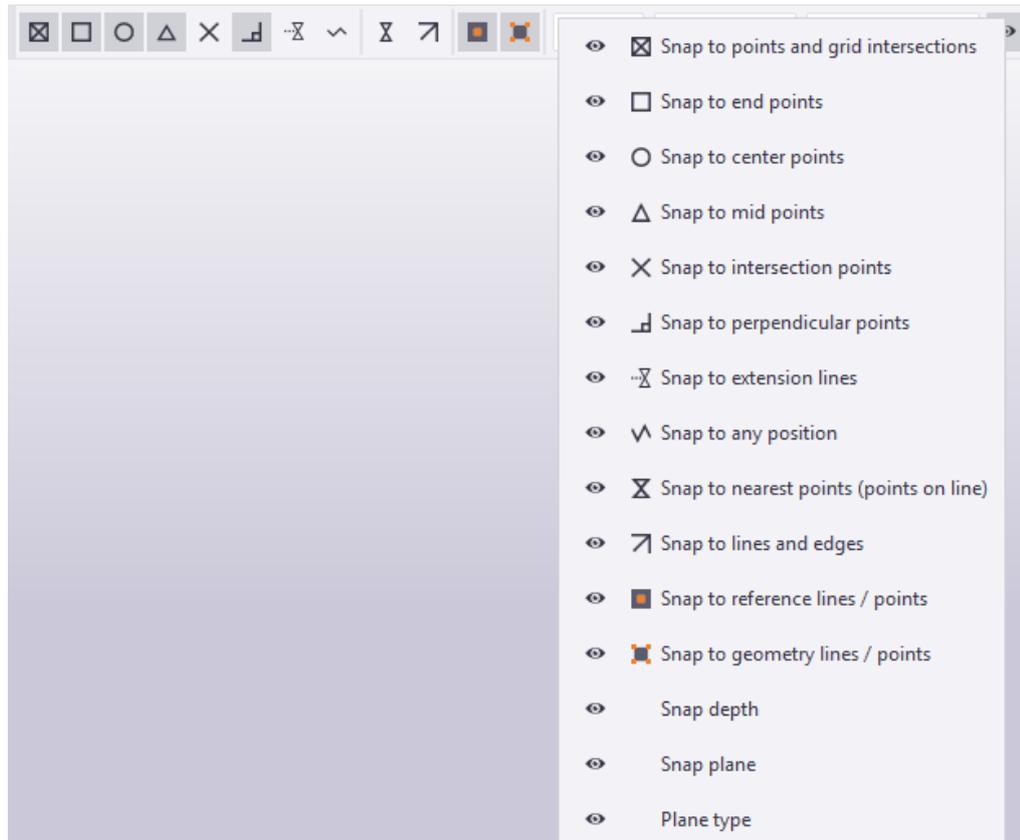
Customize the Selecting, Snapping, and Snap override toolbars

You can customize the **Selecting**, **Snapping**, and **Snap override** toolbars by hiding some of the switches. You can customize the toolbars both in the modeling mode and in the drawing mode.

Company administrators can distribute the customized toolbars to the whole organization.

Each toolbar has an eye button  that you can use for hiding the switches.

1. Click the eye button  to open a list that contains all the switches on the toolbar.



Alternatively, right-click on the toolbar to open the list.

2. To hide a switch, click the name of the switch in the list.
The selected switch becomes hidden on the toolbar and the eye button

changes in the list: .

3. To make the switch visible again, click the hidden switch in the list.

See also

[Snapping toolbar \(page 78\)](#)

[Selecting toolbar \(page 122\)](#)

Customize the contextual toolbar

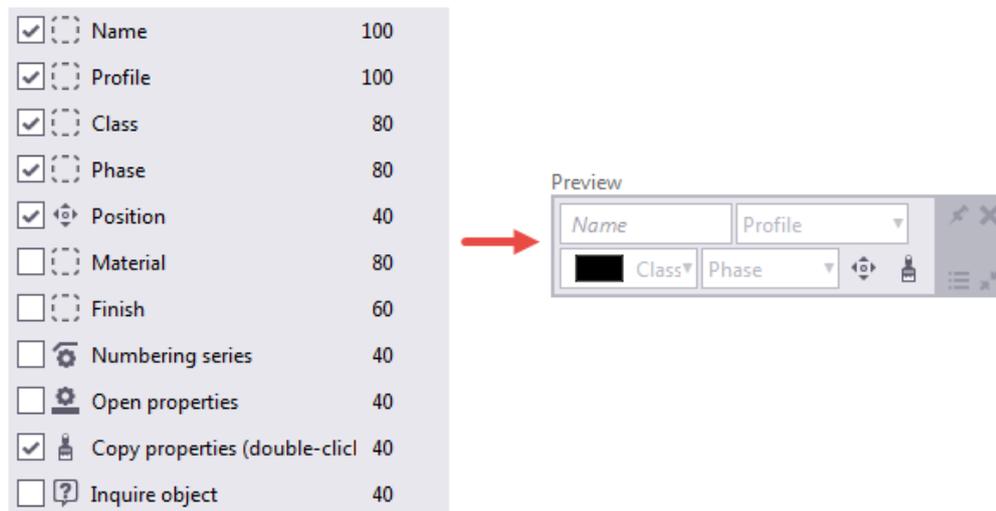
You can customize the contextual toolbar by selecting which toolbar elements are visible. You can also adjust the width of the elements, and add icons and additional titles to the elements.

Customize contextual toolbar

1. On the contextual toolbar, click .

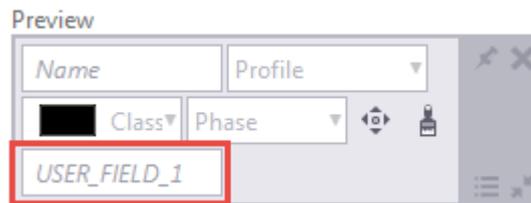
2. In the list of contextual toolbars, select the toolbar you want to customize.
3. Select and clear check boxes to define which toolbar elements you wish to show or hide.

The **Preview** area shows what the toolbar will look like. For example:



4. To modify the toolbar elements:
 - a. Click the toolbar element.
If the element can be modified, the following box appears:
 - b. Use the slider to adjust the width of the toolbar element.
 - c. To add an additional title, click the text box and enter a title.
 - d. To add an icon, click and select an icon from the list.
 - e. To remove the icon or title, click .
5. To add macros and user-defined attributes:
 - a. Select the desired macro or user-defined attribute from the list.
 - b. Click **Add**.

Tekla Structures adds the macro or user-defined attribute to the list of toolbar elements and to the **Preview** image. For example:



- c. To hide the macro or user-defined attribute, clear the corresponding check box as described in step 2.
6. Click **OK** to save the changes.

Create user profiles for contextual toolbars

You can create multiple profiles for contextual toolbars. Each profile contains the same contextual toolbars, but with different settings.

1. On the contextual toolbar, click .
2. In the **Set profiles** box, enter a name for the profile.
3. Click  to save the new profile.
4. Customize the selected contextual toolbar.
For example, remove some elements from the contextual toolbar.
5. Click **OK** to save the changes.
The user profile is now active with the settings you defined.
6. To switch to another profile:
 - a. In the **Set profiles** list, select another profile from the list.
 - b. Modify the settings.
 - c. Click **OK**.
This user profile is now active.

Back up and share contextual toolbars

We advise you to save a backup copy of your customized contextual toolbars. You can use the backup file to copy settings to another computer or to share the customizations with your co-workers.

1. Save the contextual toolbar under a user profile, with a name that you can easily recognize. For example, MyContextualToolbar.
2. Go to the `..\Users\<<user>\AppData\Local\Trimble\TeklaStructures\<<version>\ContextualToolbar\Profiles` folder.

3. Make a copy of your customized contextual toolbar and save it in the corresponding folder on another computer.
4. To open a customized contextual toolbar on another computer:
 - a. On the contextual toolbar, click .
 - b. In the **Set profiles** list, select the correct profile from the list. For example, `MyContextualToolbar`, if that is the name you used in step 1.
 - c. Click **OK**. The customizations are now active.

NOTE Alternatively, you can place the entire `ContextualToolbar` folder in your company's firm folder or in the system folder.

1.9 Tips for large models

Modeling item	Tips
Coordinate system (page 48)	<ul style="list-style-type: none"> • Do not place the model far away from the origin. The further away from the origin you model, the less precise all the computations become. • Mark global coordinates as labels instead of actually using them during modeling. • If you need to use building site coordinates, drop the first digits if they are always the same. For example, instead of coordinate 758 375 6800, use 375 6800. • Base points allow you to use another coordinate system needed for interoperability and collaboration. You can use another coordinate system for inserting reference models and exporting IFC models. When you use base points, you can keep the coordinates small and locate the model wherever needed. You can create as many base points as you need, and select one of those to be the project base point. For

Modeling item	Tips
	more information, see Base points (page 53) .
Work area (page 46) and visibility	<ul style="list-style-type: none"> • Keep the work area as small as possible. • Show only the required parts in views. • Use view filters to control the visibility of parts.
Views (page 29)	<ul style="list-style-type: none"> • Close unnecessary views. • Close all views when you save large models.
Selection switches (page 122)	<ul style="list-style-type: none"> • Switch on the Select reference models selection switch only when necessary. The speed of zooming and rotating may be affected by this switch especially in large and complicated models that contain reference models.
Round objects	<ul style="list-style-type: none"> • Create holes with the Create bolts command instead of using part cuts with round beams. • Use studs to model small straight round objects instead of small round beams. • Model lifting hooks and other embeds with reinforcement bars instead of round polybeams.
Hollow core profiles	<ul style="list-style-type: none"> • Use simple fixed (non-parametric) profiles. • Use chamfers for curved corners.
Custom components	<ul style="list-style-type: none"> • Do not create overly complex custom components. When used in great numbers they consume a lot of memory.
Numbering (page 627)	<ul style="list-style-type: none"> • Do not number the whole model in one go. Numbering all objects in large models may take a considerable amount of time.
Model database	<ul style="list-style-type: none"> • If your model file is getting large, repairing the model database can help to reduce the file size

Modeling item	Tips
	considerably and therefore help with memory problems.
Firm and Project folders	<ul style="list-style-type: none"> Save <code>Firm</code> and <code>Project</code> folders locally on the hard drive of your computer instead of a network drive. This saves time if network speed is slow. <p>When working in the multi-user mode, ensure that the folders are synchronized on all users' hard drives so that important data is not lost or changed.</p>

1.10 Create model templates

Model templates allow you to start a model with predefined company templates and settings. This can be especially useful for sub-contractors.

Only single-user models can be created with model templates. If you wish to create a multi-user model using a model template, create the model in single-user mode and then switch to multi-user mode.

By default, the model template folder is saved in your environment folder. Use the advanced option `XS_MODEL_TEMPLATE_DIRECTORY` to define a different location.

Create a new model template

You can create your own model templates and use them for creating new models. You can select which catalogs, custom components, model subfolders, drawing templates, and report templates from the model are included in the model template.

1. Create a new model.

Always start by creating a new empty model. This is because old models that have been used in live projects cannot be completely cleaned. They may contain excess information that increases the size of the model even if you delete all objects and drawings from the model.

2. Add the desired part properties, drawing properties, profiles, materials, custom components, sketches, and so on, in the model.

You can copy the needed attribute files from another model, for example.

3. On the **File** menu, click **Save as** --> **Save** .
You need to save the model to include custom components in the `xslib.db1` file. If you do not save the model, custom components will not be included in the model template.
4. On the **File** menu, click **Save as** --> **Save as model template** .
5. Enter a name for the model template.
6. Select which catalogs, drawing templates, report templates, and model subfolders to include in the model template.
For more information, see [Model template options \(page 233\)](#).
You can only select files and folders that are available in the model folder. Catalogs are typically located in an `environment` folder and they are included in the model folder only if they have been modified.
7. If you want to open the destination folder after creating the model template, select the check box.
8. Click **OK**.
You can now use the model template for creating new models.
9. When you create new models in **File** --> **New** , you can mark the important model templates as favorites, or hide the templates that you do not need.
 - a. Select a model template in the list.
 - b. Right-click and select **Favorite** or **Hidden**.
If you marked a template as **Favorite**, it is placed on top of the template list. Alternatively, use the star icon on the template to mark it as **Favorite**, or to remove the marking.
If you marked a template as **Hidden**, it is removed from the template list. Select the **Show hidden items** check box to show it again.

Modify an existing model template

To modify an existing template, save the model as a new template. Alternatively, you can modify the template by copying new or updated files directly to the model template folder.

1. Create a model using the existing model template.
2. Make the needed changes.
3. Save it as a new model template.

Download model templates

You can download, share, and store model templates using [Tekla Warehouse](#).

Model template options

Use the **Save as model template** dialog box to define which files and folders are included in the model template.

Setting	Files and folders included
Profiles	profdb.bin profitab.inp
Materials	matdb.bin
Components and sketches	ComponentCatalog.txt ComponentCatalogTreeView.txt xslib.dbl thumbnail_bitmap.arc *.dat files CustomComponentDialogFiles folder
Attribute definitions	Includes all attribute definitions of the current model.
Bolts and bolt assemblies	screwdb.db assdb.db
Reinforcement	rebar_database.inp RebarShapeRules.xml rebardatabase_config.inp rebardatabase_schedule_config.inp
Meshes	mesh_database.inp
Options	Includes all options of the current model.
Drawing templates	*.tpl files
Report templates	*.rpt files
Include model subfolders	Lists all subfolders found in the model folder. The selected folders are included in the model template. The <code>attributes</code> folder, which contains part and drawing properties, is included by default.

2 Create parts, reinforcement, and construction objects

When you know the basic principles of how to create and modify different types of model objects in Tekla Structures, you can start working on more detailed level with your model.

First, to get your model started, you need to create some [parts \(page 235\)](#). Parts are the building blocks of the physical model. You can continue working with the parts by, for example, [deforming their shape \(page 324\)](#), or by [adding some details to the parts \(page 331\)](#), such as bolts, welds, or cuts or fittings.

When you use a workshop weld or bolt to join parts together, you will learn how to [work with assemblies of steel parts \(page 386\)](#).

For concrete parts, each concrete part is considered a [separate cast unit \(page 396\)](#). For construction purposes, you may need to merge several concrete parts into one cast unit. If you model cast-in-place concrete structures, you may need to check how to [work with pours \(page 402\)](#). And once you have created a model of concrete parts, you will need to [reinforce the parts \(page 427\)](#) to gain higher strength for the parts.

Additionally, you may need to use [points or construction objects \(page 561\)](#) as modeling aids. Points and construction objects help you to place other objects in the model.

See also

[Create model objects and modify model object properties by using property pane \(page 96\)](#)

[Change how model objects are displayed \(page 581\)](#)

[Show and hide model objects \(page 581\)](#)

[Change the color and transparency of model objects \(page 593\)](#)

[Check the model \(page 599\)](#)

[Number the model \(page 627\)](#)

2.1 Create parts and modify part properties

In Tekla Structures, the term *part* refers to the basic building objects that can be modeled and detailed further. Parts are the building blocks of the physical model.

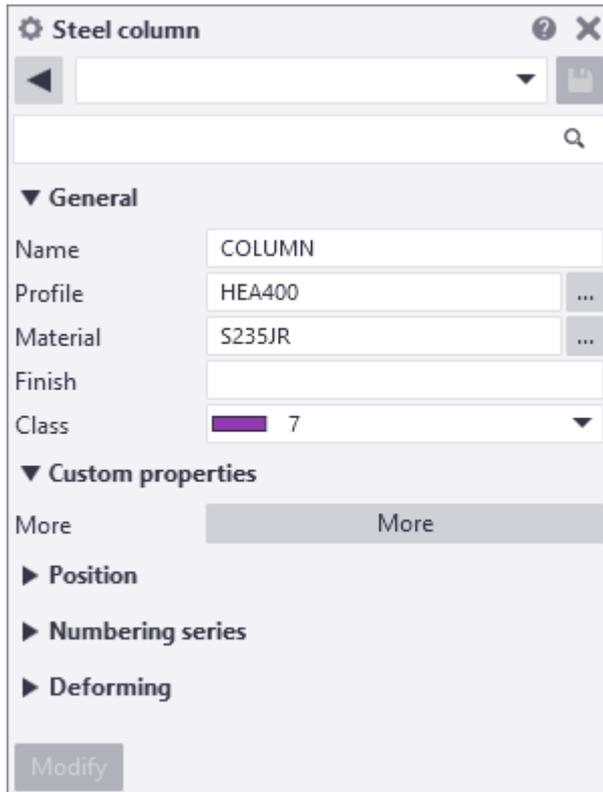
You can create steel parts and concrete parts. A special type of parts are items. Use them to model objects that would be difficult to model using basic Tekla Structures parts and commands, such as cutting.

Create steel parts by using the commands on **Steel** tab on the ribbon. Create concrete parts by using the commands on the **Concrete** tab on the ribbon.



Every part has properties that define it, such as material, profile, and [location \(page 306\)](#). Also, parts have [user-defined attributes \(UDAs\) \(page 319\)](#) that you can use to provide additional information about a part. You can use part properties in [view filters \(page 159\)](#) and [selection filters \(page 161\)](#) to, for example, select, modify, and hide parts. You can also include part properties and UDAs in drawing and report templates.

[To view and modify the part properties \(page 96\)](#), use the property pane. You can view and modify the properties of a one part type at a time, or the common properties of several, similar part types. If needed, you can [copy properties \(page 109\)](#) from one part to another part by using the  **Copy properties** button in the property pane.



Basic steel parts

The basic steel parts are

- column
- beam
- polybeam
- curved beam
- twin profile
- orthogonal beam
- spiral beam
- plate
- bent plate

Basic concrete parts

The basic concrete parts are

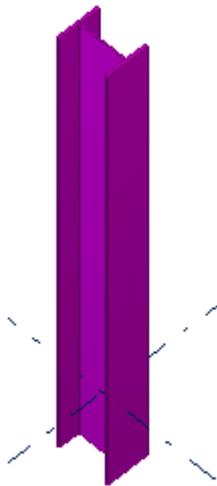
- column
- beam
- polybeam
- spiral beam

- panel
- slab
- pad footing
- strip footing

Create a steel column

1. On the **Steel** tab, click **Column**  .
2. Pick a point.

Tekla Structures creates the column using the **Steel column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Steel column** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify steel column properties

1. If the property pane is not open, double-click the column to open the **Steel column** properties.
2. [Change \(page 96\)](#) the properties as needed.

- Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Steel column properties

Use the **Steel column** properties in the property pane to view and modify the properties of a steel column. To open the properties, double-click the steel column. The file name extension of a steel column property file is *.clm.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

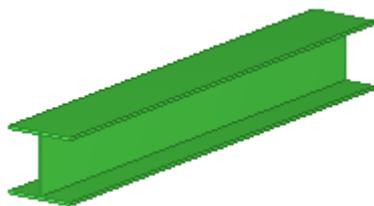
Setting	Description
General	
Name	User-definable name of the column. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the column.
Material	Material (page 318) of the column.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number for the assembly position number (page 628) .
Position	
Vertical	Vertical position (page 312) of the column, relative to column's reference point.
Rotation	Rotation (page 309) of the column around its axis on the work plane.

Setting	Description
Horizontal	Horizontal position (page 313) of the column, relative to column's reference point.
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 331) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel beam

1. On the **Steel** tab, click  .
2. Pick two points.

Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.

- In the property pane, click the **Object type list** button  and select **Steel beam** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify steel beam properties

- If the property pane is not open, double-click the beam to open the **Steel beam** properties.
- [Change \(page 96\)](#) the properties as needed.
- Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is *.prt.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	

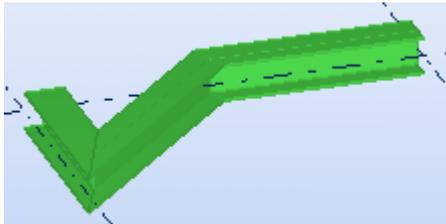
Setting	Description
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number for the assembly position number (page 628) .
Position	
On plane	Beam's position on the work plane (page 308) , relative to the beam's reference line.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
At depth	Position depth (page 310) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 315) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 315) perpendicular to the beam's reference line.
Dz	Move the beam end (page 315) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 331) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a steel polybeam

A polybeam can contain straight and curved segments.

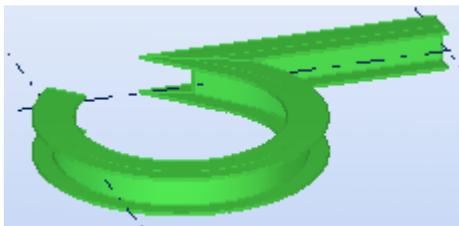
1. On the **Steel** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

Tekla Structures creates the polybeam between the points you picked, using the **Steel beam** properties in the property pane.



4. If you want to create curved segments, chamfer the corners of the polybeam.

For example:



Modify steel polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Steel beam** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is *.prt.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

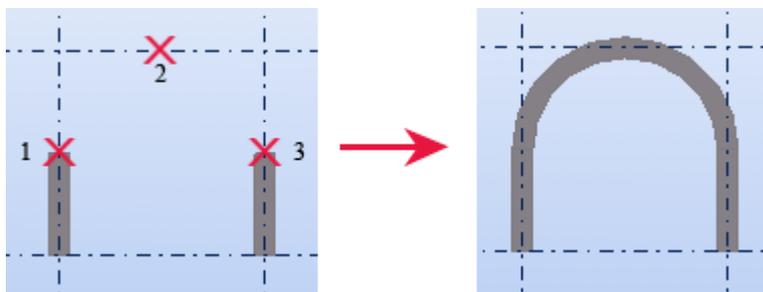
Setting	Description
General	
Name	User-definable name of a beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number for the assembly position number (page 628) .
Position	
On plane	Beam's position on the work plane (page 308) , relative to the beam's reference line.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
At depth	Position depth (page 310) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 315) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 315) perpendicular to the beam's reference line.

Setting	Description
Dz	Move the beam end (page 315) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 331) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a curved beam

1. On the **Steel** tab, click **Beam** --> **Curved beam**.
2. Pick the start point (1).
3. Pick a point on the arc (2).
4. Pick the end point (3).

Tekla Structures creates the beam between the points you picked, using the **Steel beam** properties in the property pane.



Modify curved beam properties

1. If the property pane is not open, double-click the curved beam to open the **Steel beam** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Steel beam properties

Use the **Steel beam** properties in the property pane to view and modify the properties of a steel beam, steel polybeam, or curved beam. To open the properties, double-click the steel beam. The file name extension of a beam property file is *.prt.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number for the assembly position number (page 628) .
Position	

Setting	Description
On plane	Beam's position on the work plane (page 308) , relative to the beam's reference line.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
At depth	Position depth (page 310) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 315) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 315) perpendicular to the beam's reference line.
Dz	Move the beam end (page 315) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 331) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

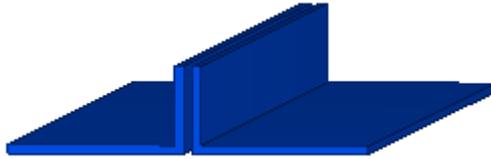
Create a twin profile

A twin profile consists of two parallel and identical beams. You define the positions of both beams by selecting the twin profile type and setting the clearance between the beams in two directions.

1. On the **Steel** tab, click **Beam** --> **Twin profile**.

- Pick two points.

Tekla Structures creates the twin profile between the points you picked, using the **Twin profile** properties in the property pane.



Alternatively, you can start the command in the property pane.

- Ensure that you have nothing selected in the model.
- In the property pane, click the **Object type list** button  and select **Twin profile** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify twin profile properties

- If the property pane is not open, double-click either of the beams to open the **Steel beam** properties.
- [Change \(page 96\)](#) the properties as needed.
- Click **Modify** to apply the changes.

Twin profile properties

Use the **Twin profile** properties in the property pane to view and modify the properties of a twin steel profile. The file name extension of a twin profile property file is *.dia.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a twin profile. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	The profile of both the beams in the twin profile.
Material	Material (page 318) of the beams.

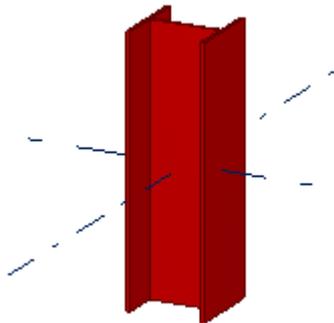
Setting	Description
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group twin profiles. For example, you can display parts of different classes in different colors.
Twin profile type	Define how the profiles are combined.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number the assembly position number (page 628) .
Position	
On plane	Twin profile's position on the work plane (page 308) , relative to the twin profile's reference line.
Rotation	Rotation (page 309) of the twin profile around its axis on the work plane.
At depth	Position depth (page 310) of the twin profile. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the twin profile (page 315) by moving the twin profile end point along the twin profile's reference line.
Clearance between members	
Horizontal	The horizontal clearance between the profiles.
Vertical	The vertical clearance between the profiles.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create an orthogonal beam

Use the **Orthogonal beam** command when you want to create a steel part that is perpendicular to the current work plane. After you have created an orthogonal beam, you can modify it as if it were beam or column.

1. On the **Steel** tab, click **Beam** --> **Orthogonal beam**.
2. Pick a point.

Tekla Structures creates the beam using the **Orthogonal beam** properties in the property pane, and at the [level \(page 323\)](#) defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Orthogonal beam** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify orthogonal beam properties

1. If the property pane is not open, double-click the orthogonal beam to open the properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Orthogonal beam properties

Use the **Orthogonal beam** properties in the property pane to view and modify the properties of an orthogonal steel beam. The file name extension of an orthogonal beam property file is `*.crs`.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 312) of the beam, relative to beam's reference point.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
Horizontal	Horizontal position (page 313) of the beam, relative to beam's reference point.
Top	Position of the second end of the beam in the global z direction.
Bottom	Position of the first end of the beam in the global z direction.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number the assembly position number (page 628) .
Custom properties	
More	Click the More button to open the user-defined attributes (page 319)

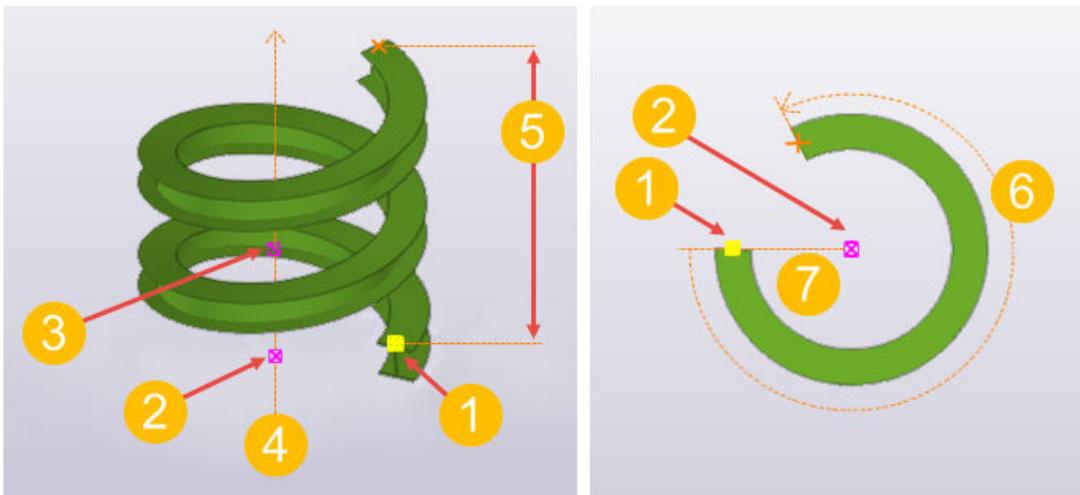
Setting	Description
	(UDAs) of the part. UDAs provide additional information about the part.

Create a steel spiral beam

Use the **Create steel spiral beam** command when you want to model spiral staircases and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.



(1) Start point (the first point picked)

(2) Center point (the second point picked)

(3) Direction of the rotation axis (the optional third point picked)

(4) Center axis

(5) Total rise: the distance from the start point to the end point, parallel to the center axis

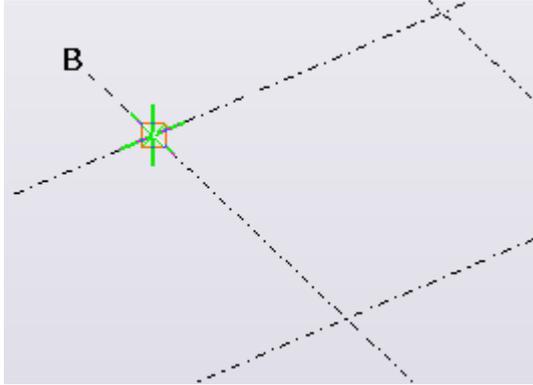
(6) Rotation angle: the rotation angle of the spiral beam, given in degrees.
Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

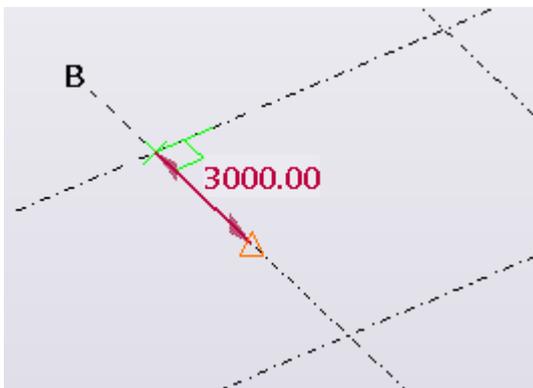
Create a spiral beam

1. On the **Steel** tab, click **Beam** --> **Spiral beam**.

- Pick the start point.



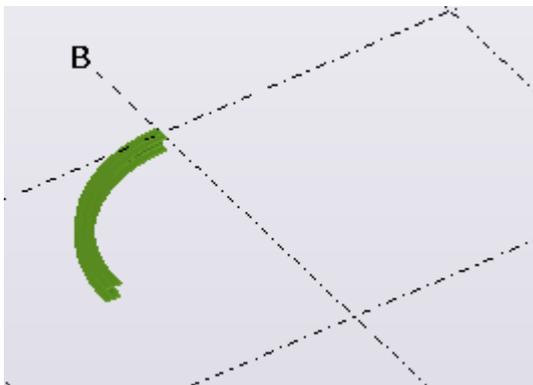
- Pick the center point.



- To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



- Click the spiral beam to select it.

The contextual toolbar appears with the following options:



- (1) Rotation angle
- (2) Total rise
- (3) Twist angle at start
- (4) Twist angle at end

6. To add more rotation, enter a bigger value in the **Rotation angle** box.
7. To make the spiral more loose, enter a bigger value in the **Total rise** box.
8. To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.
- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Create a contour plate

When you create a contour plate, the profile you select defines the thickness of the plate and the points you pick define the shape. The corners of the contour plate can be chamfered.

1. On the **Steel** tab, click  .

2. Pick the corner points of the contour plate.
3. Click the middle mouse button.

Tekla Structures creates the plate using the **Contour plate** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Contour plate** from the list.

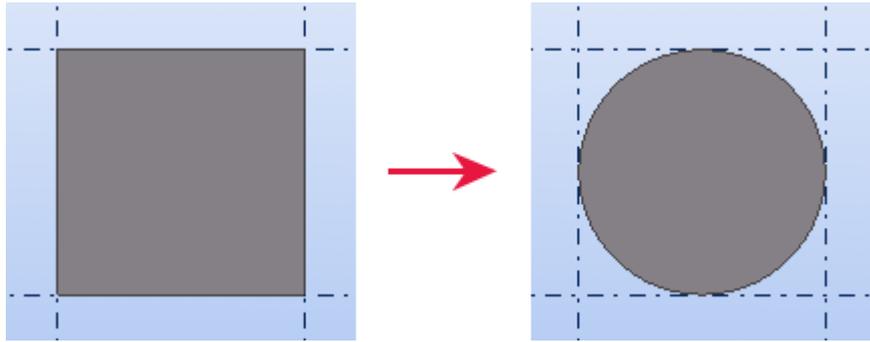
Tekla Structures starts the command and shows the properties in the property pane.

Create a round contour plate

1. Create a square contour plate with four equal sides.
2. Select the plate.
3. Select the handles of the plate.

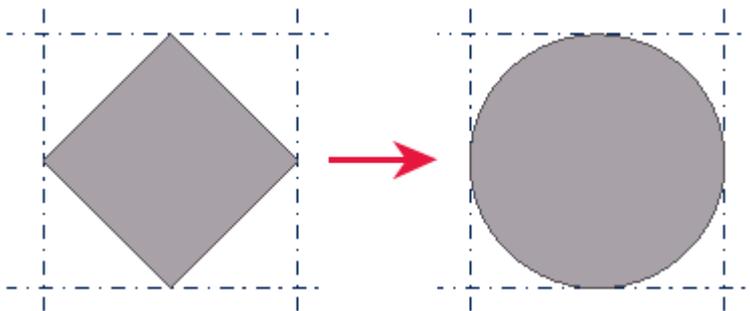
TIP To select all the handles at once, hold down the **Alt** key and drag the mouse from left to right, covering all the handles.

4. Press **Alt + Enter** to display the **Chamfer Properties** dialog box.
5. Select the round chamfer symbol  from the list.
6. Enter the chamfer radius in the **x** box. The radius must be equal to half of the side of the square.
7. Click **Modify**.



Alternative way of creating a round plate

1. Create a diamond-shaped plate with four equal sides.
2. To round the corners, use the arc point chamfer type .



Modify contour plate properties

1. If the property pane is not open, double-click the plate to open the **Contour plate** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Contour plate properties

Use the **Contour plate** properties in the property pane to view and modify the properties of a contour plate. To open the properties, double-click the contour plate. The file name extension of a contour plate property file is *.cpl.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of the contour plate. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the contour plate.
Material	Material (page 318) of the contour plate.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group contour plates. For example, you can display parts of different classes in different colors.
Numbering series	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number the assembly position number (page 628) .
Position	
At depth	Position depth (page 310) of the contour plate. The position is always perpendicular to the work plane.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a conical or a cylindrical bent plate

You can create cylindrical or conical bent steel plates either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate (for example, PL200*20). Position the parts so that there is some room on both sides, so that Tekla Structures can create a curved section between them.

After creating a cylindrical or conical bent plate, the individual parts no longer exist in the model. The bent plate gets its properties and coordinates from the first part that you selected when creating the bent plate. The first part is the main section of the bent plate. You can later change the main section, if needed.

Limitations

- Only the side faces of the part can be used for creating a bent plate.
- Chamfered or cut faces cannot be used for creating a bent plate.
- Curved beams and deformed parts cannot be used for creating a bent plate.
- Details (such as bolts, welds, cuts, chamfers, and preparations) in simple cases are supported on the curved section of the bent plate.

In addition to the cylindrical and conical bent plates you can also create [stand-alone bent plates \(page 271\)](#), which do not require any input parts.

Create a cylindrical bent plate

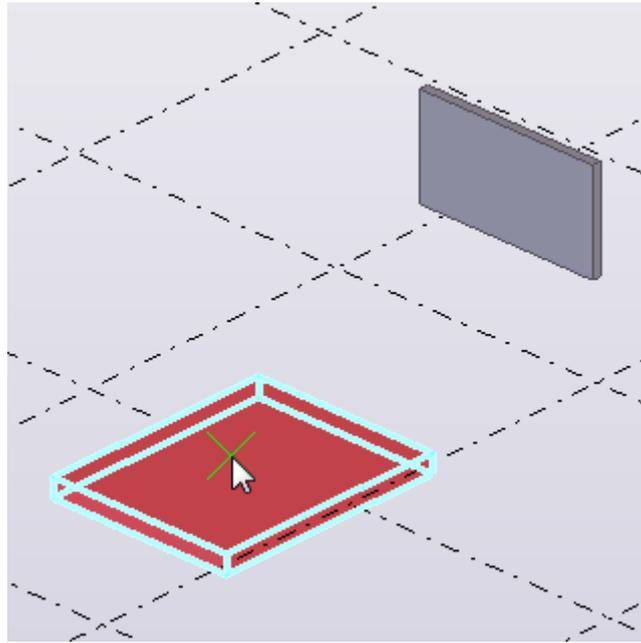
You can create a cylindrical bent plate by selecting two steel parts or two part faces. Cylindrical bent plate has a radius that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

You can create cylindrical bent plates also when the selected parts are intersecting.

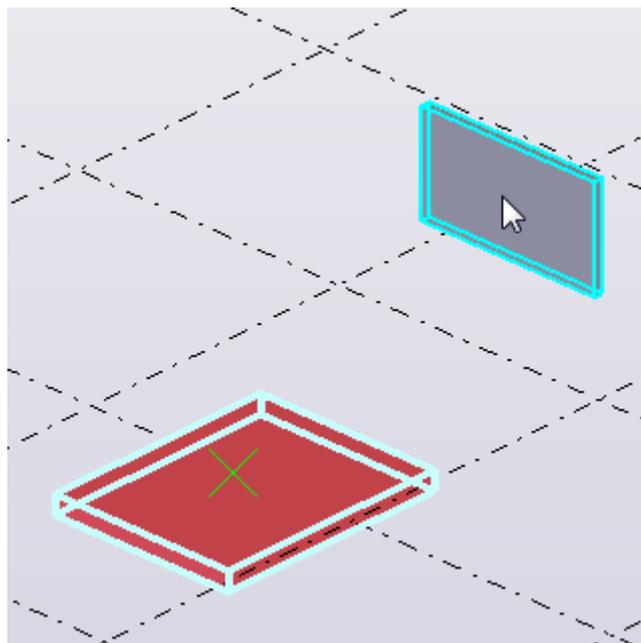
1. On the **Steel** tab, click **Plate** --> **Create cylindrical bent plate** .
2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

Additionally, you can enter the radius of the cylindrical bent plate. If no radius is entered, Tekla Structures creates the bent plate using the default radius.

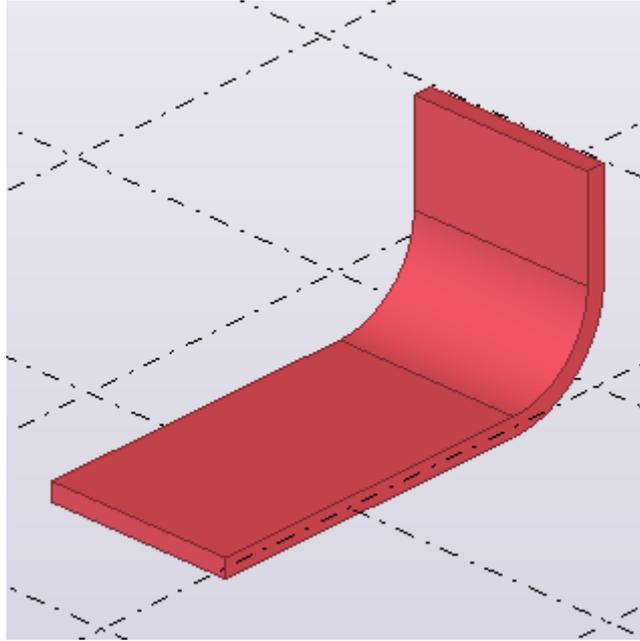
- If you selected **By parts**:
 - a. Select the first part.



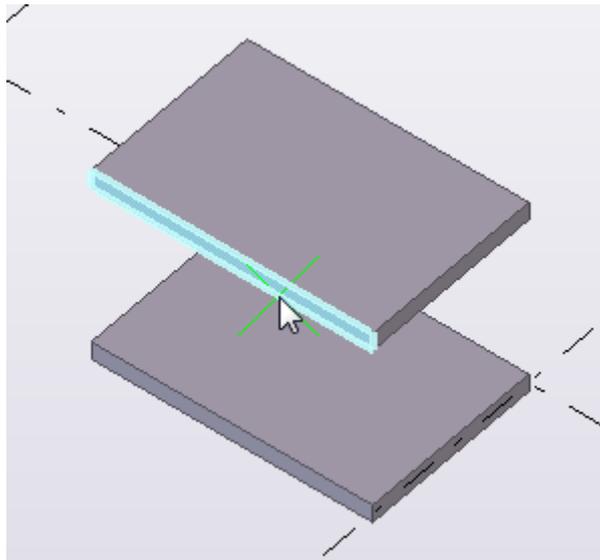
b. Select the second part.



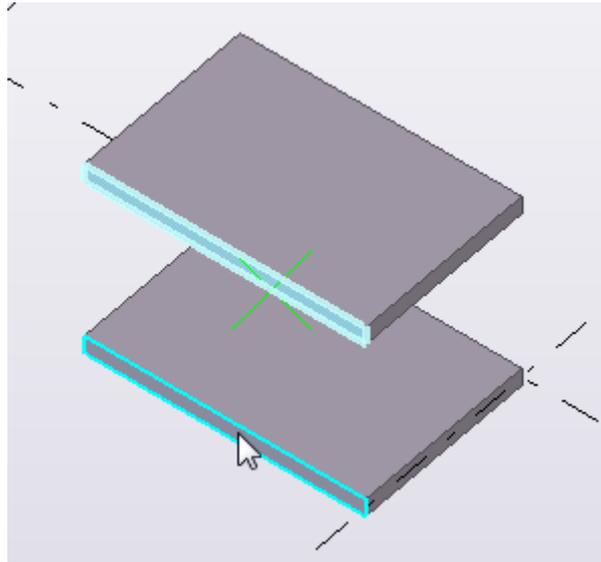
Tekla Structures creates the cylindrical bent plate.



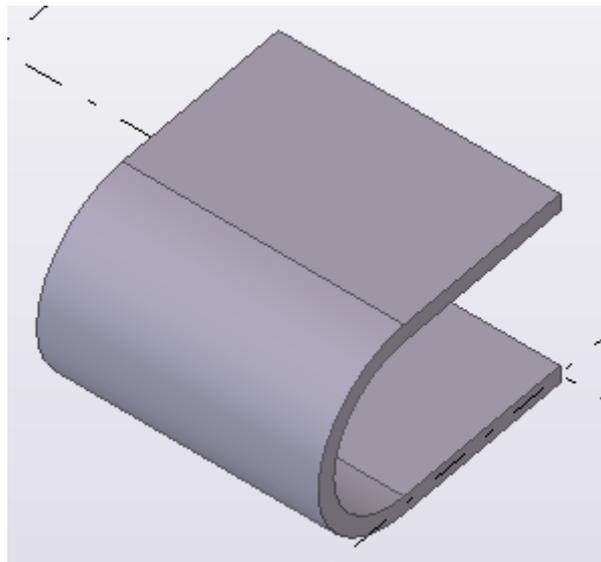
- If you selected **By faces**:
 - a. Select the first part face.



- b. Select the second part face.



Tekla Structures creates the cylindrical bent plate.



Create a conical bent plate

You can create a conical bent plate by selecting two steel parts or two part faces. Conical bent plate has two radiuses that you can modify. The bent plate properties, such as the ID, thickness, class, and material of the plate, are determined by the first part you select.

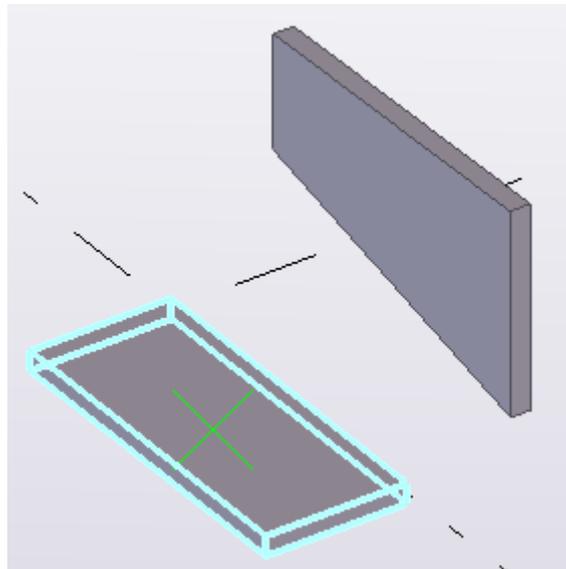
To be able to create conical bent plates, the selected parts or part faces need to be of a shape that creating a conical bent plate is possible. If the selected parts or part faces are such that creating a cylindrical bent plate is possible, a

cylindrical bent plate is created. You can create different shapes of conical bent plates: inward bend, outward bend, or with 180 degrees opening angle.

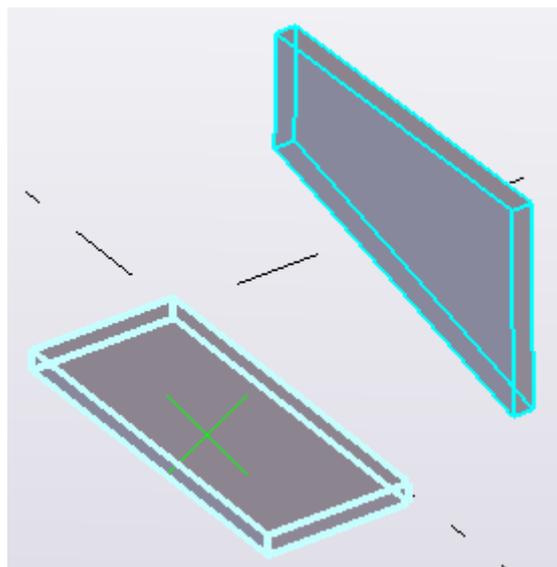
1. On the **Steel** tab, click **Plate** --> **Create conical bent plate** .
2. On the bent plate toolbar, select whether you want to create the bent plate by selecting parts or by selecting part faces.

You can enter two radiuses for the conical bent plate. If no radiuses are entered, Tekla Structures creates the bent plate using the default radiuses.

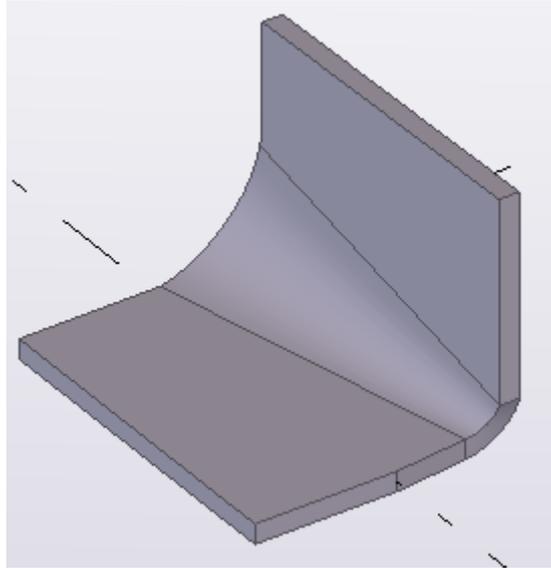
- If you selected **By parts**:
 - a. Select the first part.



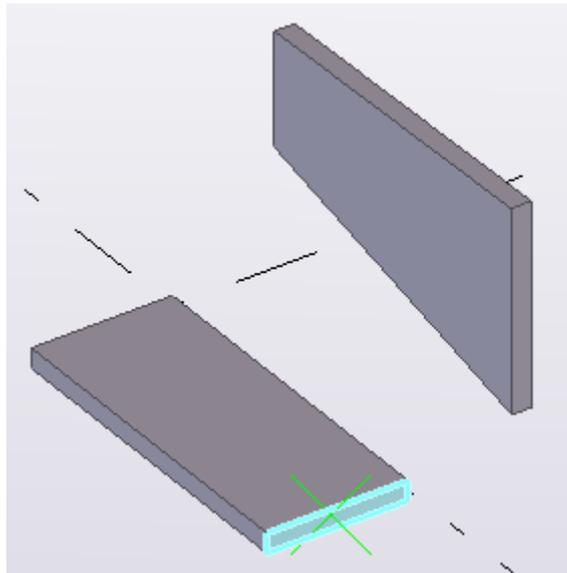
- b. Select the second part.



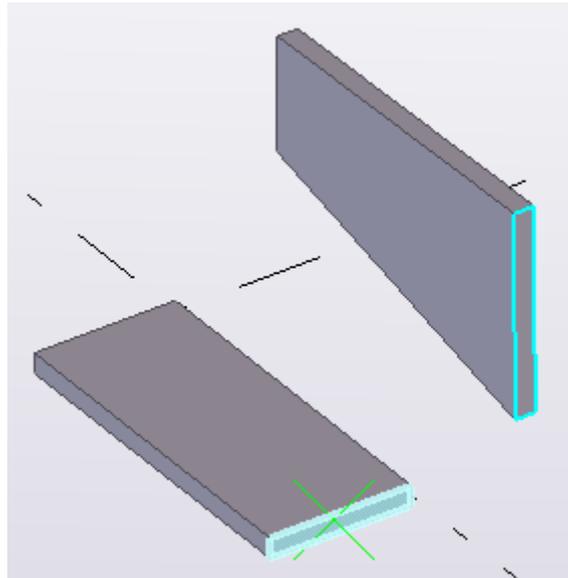
- c. Tekla Structures creates the conical bent plate.



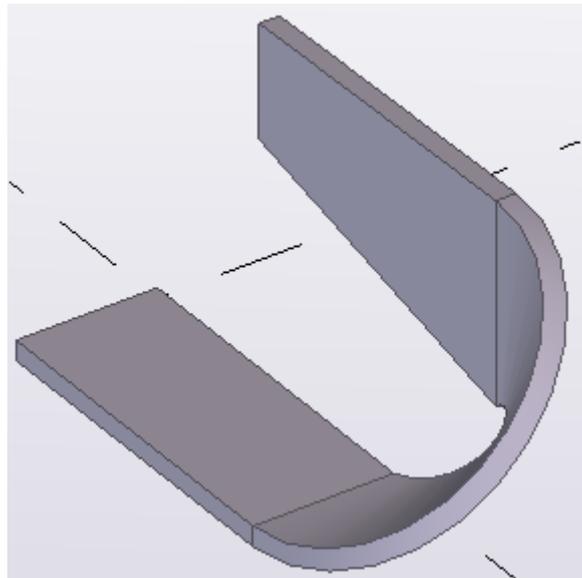
- If you selected **By faces**:
 - a. Select the first part face.



- b. Select the second part face.



- c. Tekla Structures creates the conical bent plate.

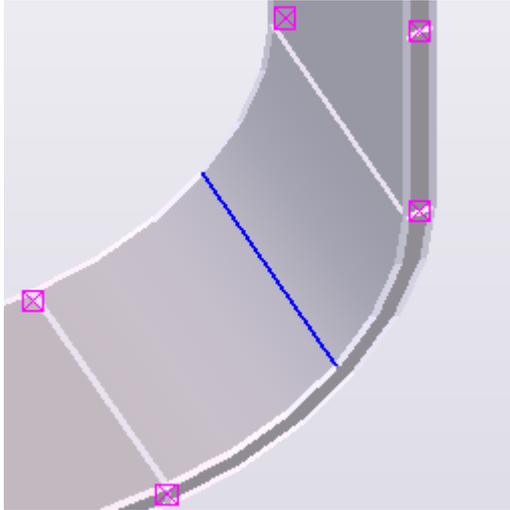


Modify the bend radius

When you create a cylindrical bent plate, you can enter a radius for the plate. For a conical bent plate you can enter two radii. If you do not enter any radius, Tekla Structures uses a default bend radius when creating bent plates. You can later change the bend radius to suit your needs.

1. Ensure that  **Direct modification** is switched on.
2. Select the bent plate.

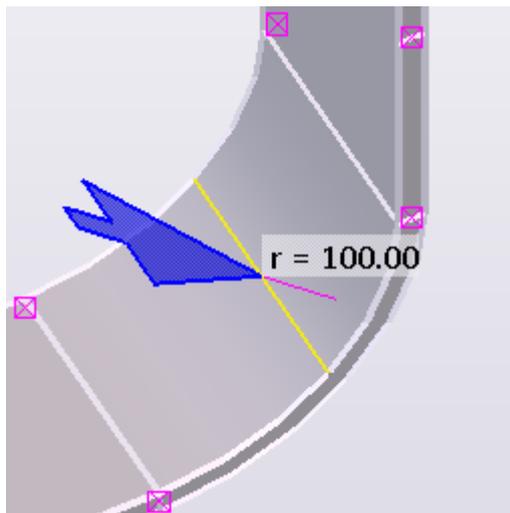
A blue line handle appears in the middle of the curved section.



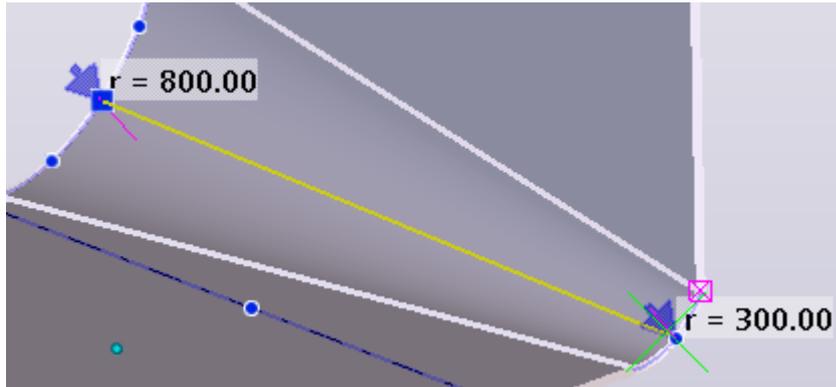
3. Select the line handle.

Depending on the type of the bent plate, one (for cylindrical bent plate) or two (for conical bent plate) blue dimension arrows appear.

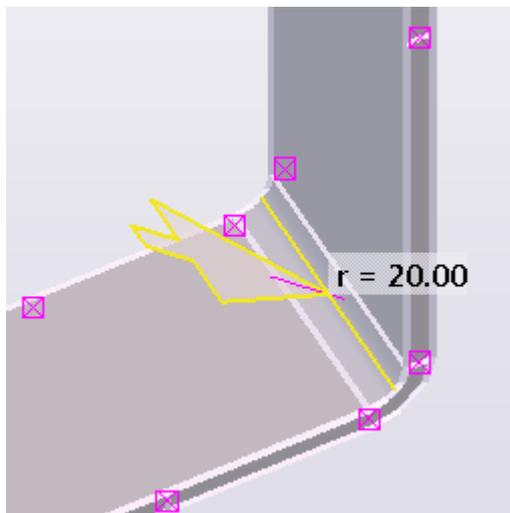
- For cylindrical bent plate:



- For conical bent plate:



4. To modify the radius, do one the following:
 - Drag the arrow or arrows forward or backward along the magenta line. The dimension "r =" changes accordingly. When you release the arrow, the radius also changes in the model.



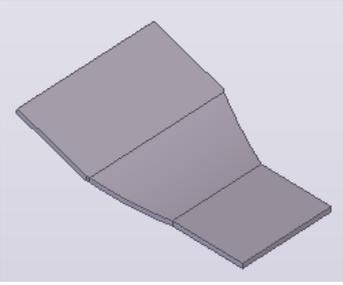
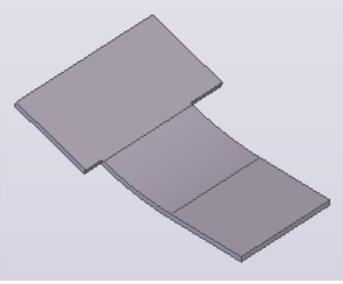
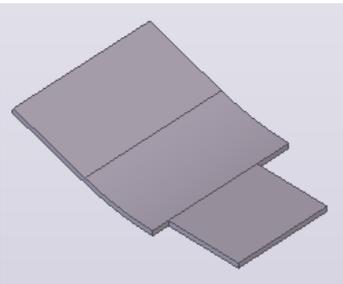
- On the contextual toolbar, enter the radius or the radiuses.
- Alternatively, you can select the arrow and type a dimension. When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box. Click **OK** to confirm the dimension.

Modify the shape of a bent plate

When you create a bent plate, Tekla Structures adds a curved section between the parts you select. You can modify the curved section by choosing one of the predefined options or by modifying the shape manually. You can also modify the flat sections, which are the original parts the bent plate was made of.

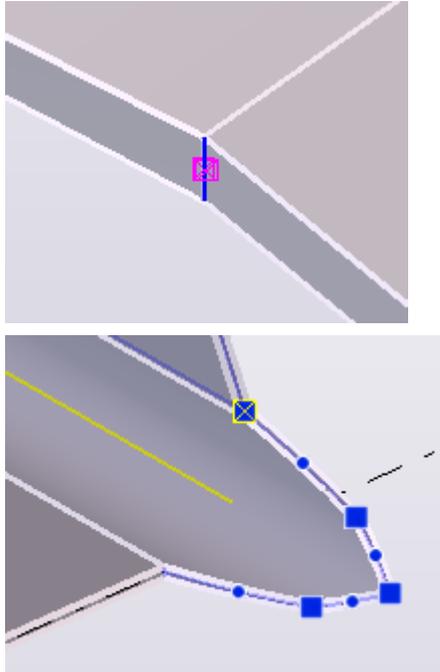
1. Ensure that  **Direct modification** is switched on.

2. Select the bent plate.
A blue line handle appears in the middle of the curved section.
3. Select the line handle.
A contextual toolbar appears.
4. On the contextual toolbar, select one of the predefined shape options:

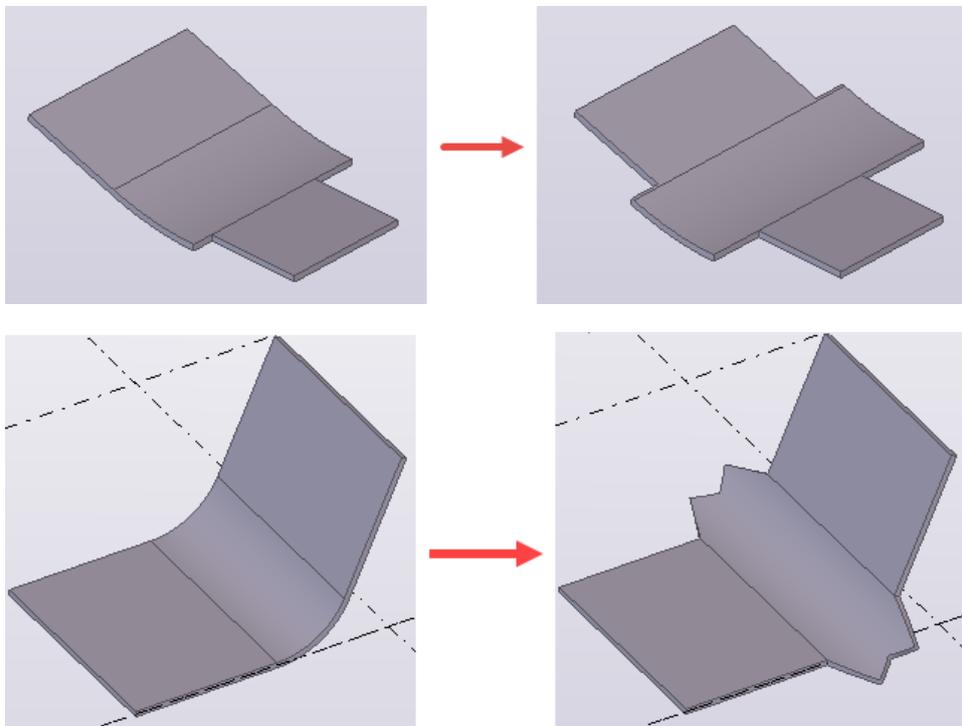
Option	Description	Example
<p>Tapered bend</p> 	<p>A gradual decrease in the width between the parts.</p> <p>This is the default shape.</p>	
<p>Narrow bend</p> 	<p>Constant width between the parts. The width is determined by the narrowest part.</p>	
<p>Wide bend</p> 	<p>Constant width between the parts. The width is determined by the widest part.</p>	

5. To modify the curved section manually:
 - a. Select the blue line handle.

You can modify the side boundaries or the lateral boundaries of the bends both in the cylindrical and in the conical bent plate. Tekla Structures displays the boundary handles in blue:



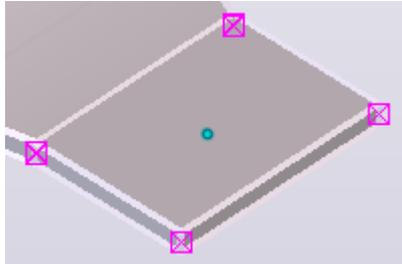
- b. Drag the handles to change the shape of the curved section.
For example:



- 6. To modify the flat sections:

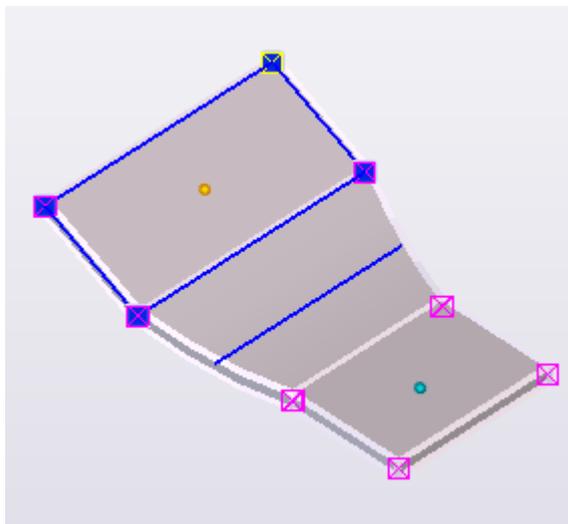
- a. Select the bent plate.

Tekla Structures displays a green selection handle in the middle of each flat section:



- b. Click the selection handle of the section you want to modify.

The direct modification handles of the selected section become visible:



- c. Use the direct modification handles to change the shape of the flat section.

7. To modify the angle of the bent plate:

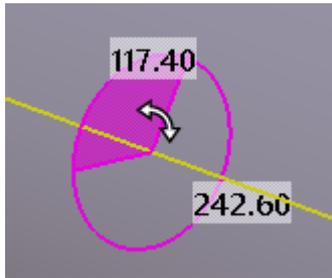
- a. Click the green selection handle in the middle of the flat section whose angle you want to modify.

- b. Select the line handle.

A contextual toolbar appears.

- c. On the contextual toolbar, click  **Enable angle manipulator.**

The angle manipulator wheel appears in the model.



- d. Modify the angle using the wheel.

If you want to modify the angle of another flat section, click the other green selection handle.

8. To change the main section of the bent plate:

- a. Click the green selection handle of the section that you want to set as the main section.

A contextual toolbar appears.

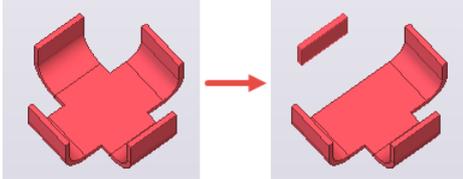
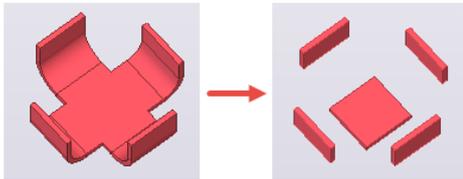
- b. On the contextual toolbar, click  **Set main section.**

The direct modification handles become active in the new main section. The main section and the coordinate system of the bent plate change accordingly, changing the bent plate orientation in an unfolded drawing.

Remove curved sections

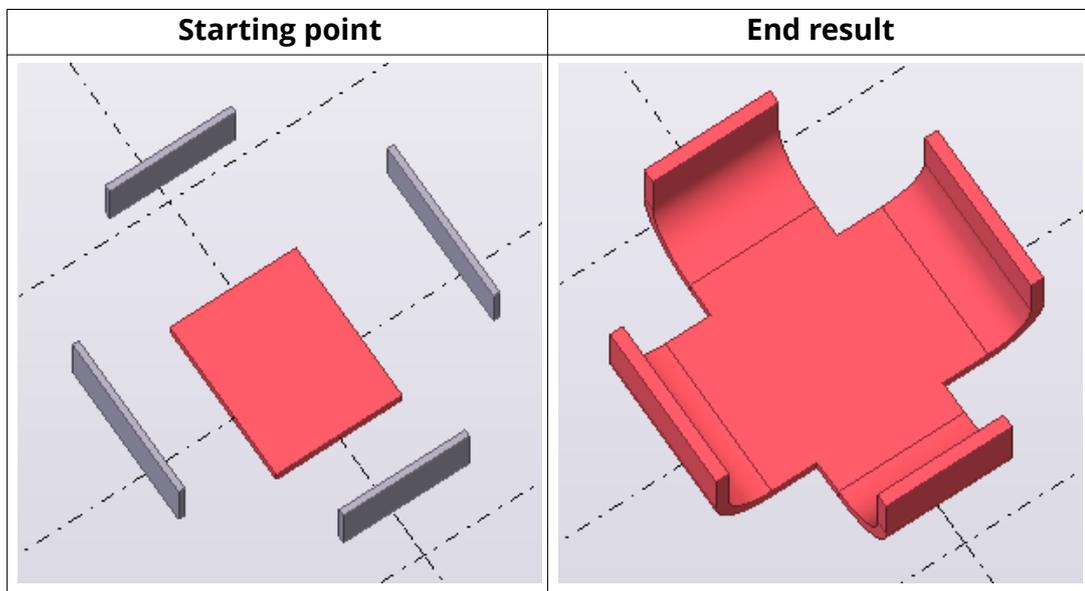
You can return bent plates into individual objects, and then edit and use them as any other model object. If the bent plate consists of several curved sections that are all joined to the same part, you can either remove each curved section separately or explode the entire bent plate all at once.

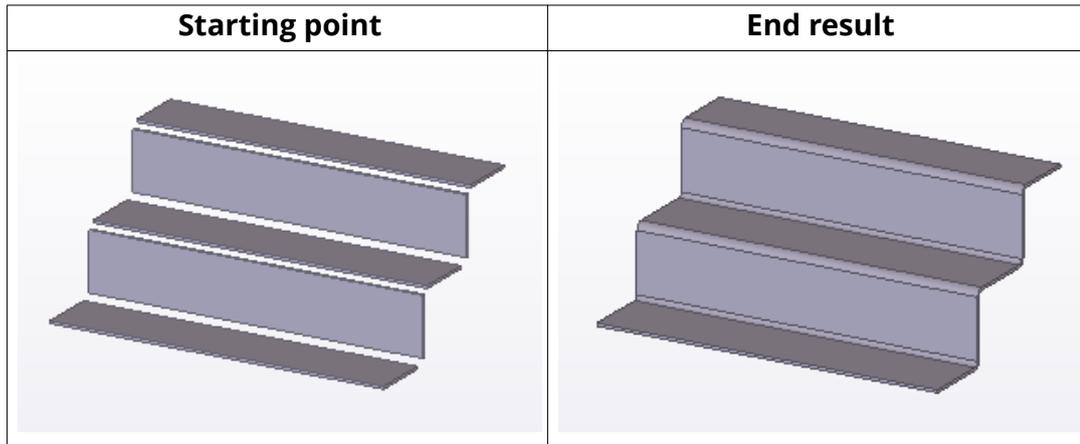
To	Do this
To remove individual curved sections	<ol style="list-style-type: none"> 1. Ensure that  Direct modification is switched on. 2. Select the curved section you want to remove. A blue line handle appears. 3. Select the line handle. A contextual toolbar appears. 4. On the contextual toolbar, click  Remove bend.

To	Do this
	<p>Tekla Structures removes the selected curved section. For example:</p> 
To explode the entire bent plate	<ol style="list-style-type: none"> 1. Select one of the curved sections. 2. Right-click and select Explode. <p>Tekla Structures explodes the entire bent plate into individual objects. For example:</p> 

Examples

Here are some examples of bent plates that you can create:





Create a stand-alone bent plate

You can create stand-alone bent plates which do not require any input parts to be selected. Use stand-alone bent plates to model cylindrical and conical parts, such as hoods, hoppers, cones, and so on.

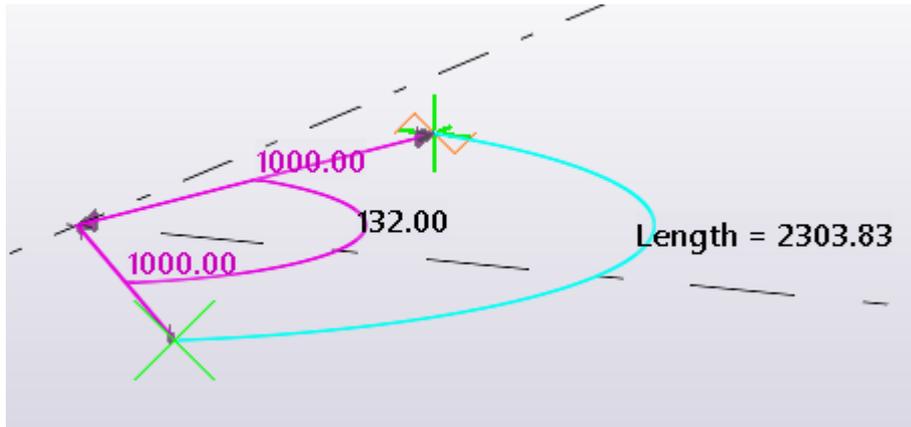
Limitations

- Tekla Structures does not support full 360 degree stand-alone bent plates. However, you can create 359 degree plates instead.
- Use the local coordinate system in drawing creation.
- Unfolded drawings of less than 180 degree conical bent plates work correctly. However, with more than 180 degree conical bent plates there might be some unexpected results in the drawing creation.

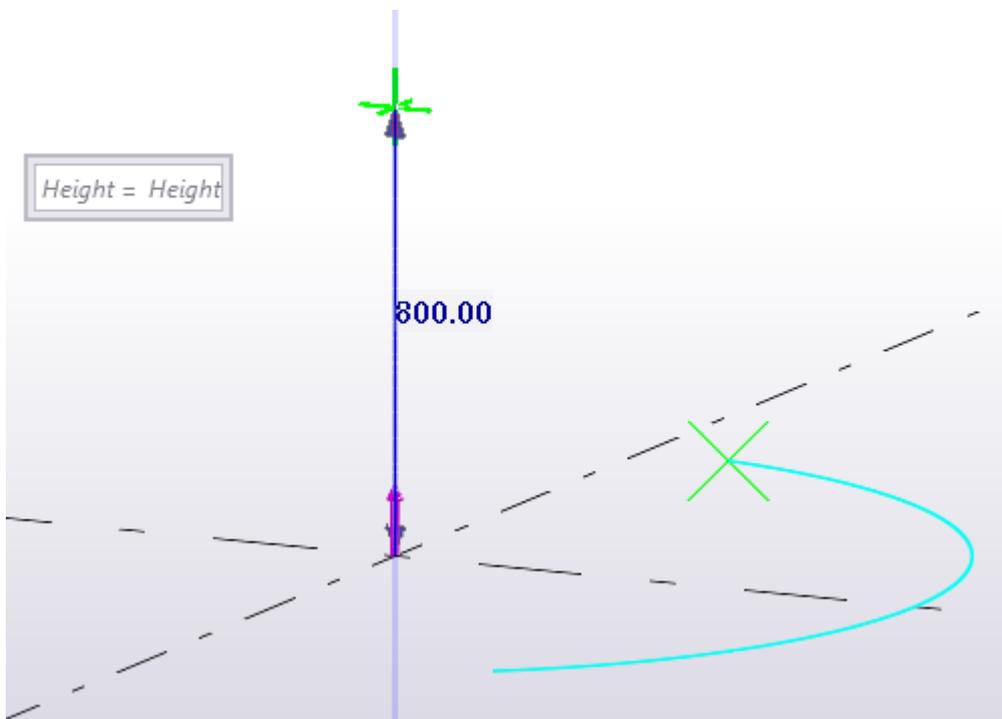
In addition to the stand-alone bent plates, you can also create [cylindrical and conical bent plates \(page 256\)](#) either by selecting two parts or two part faces. The parts that you use for creating a bent plate must be contour plates, or beams whose profile is a plate.

1. Ensure that  **Direct modification** is switched on.
2. On the **Steel** tab, click **Plate** --> **Create stand-alone bent plate** .
3. Define the first radius of the bend:
 - a. Pick the center point.
 - b. Pick the start point for the arc.
 - c. Pick the end point for the arc.

The picking order defines the up direction. For example, if you create an arc on the xy plane in the counterclockwise direction, the up direction points to the positive z axis, according to the [right-hand rule \(page 50\)](#).

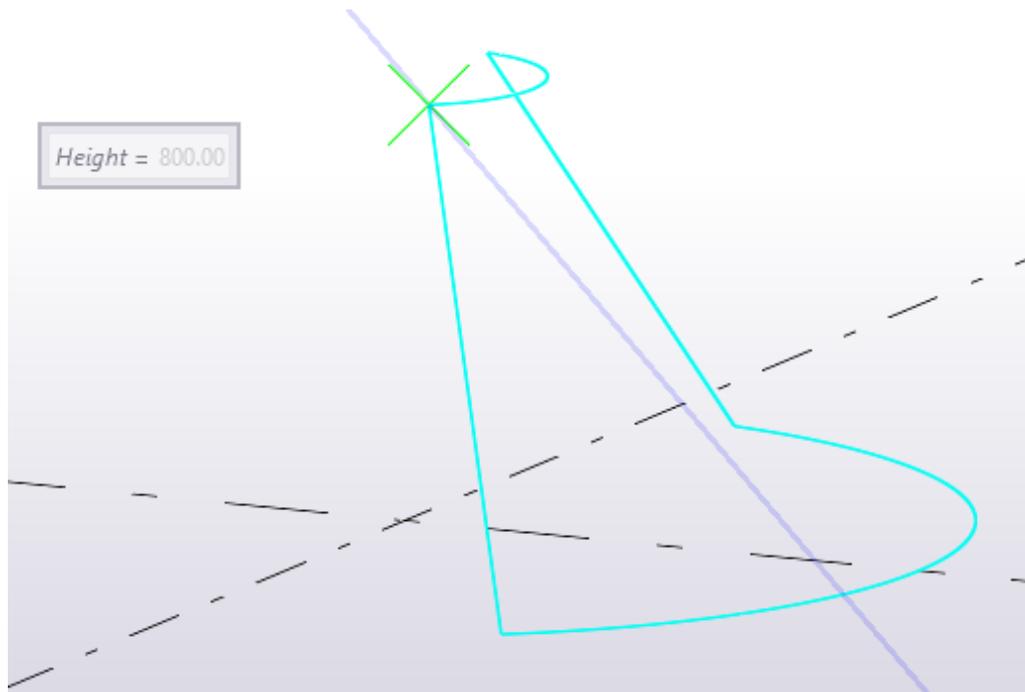


4. Pick a point to define the height of the bend.
Alternatively, you can enter the height in the bent plate contextual toolbar.

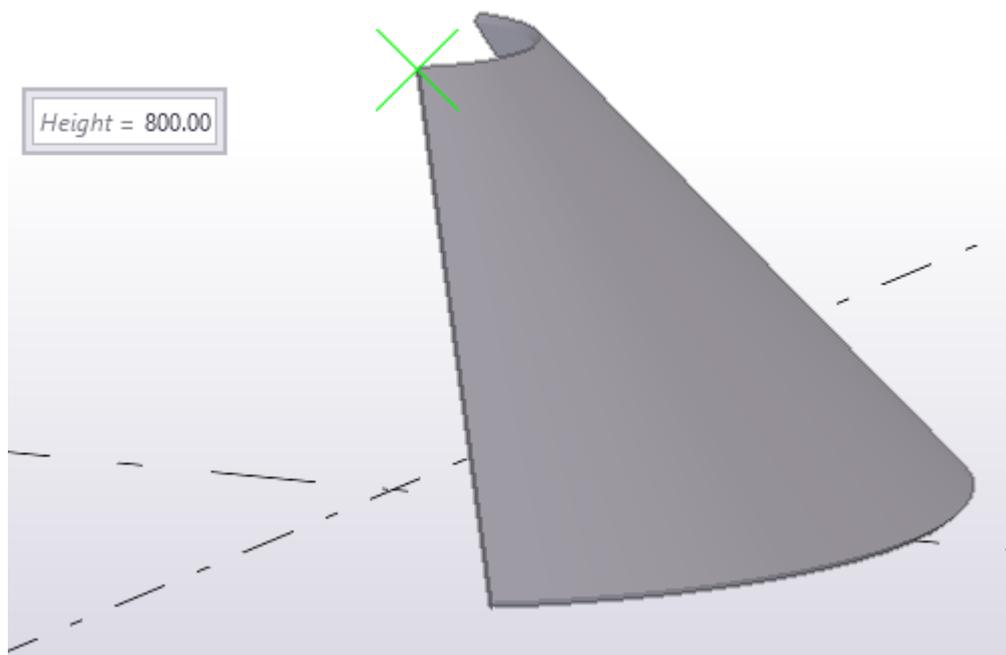


5. Define the second radius of the bend:
 - a. Pick a point based on the preview of the plate.
 - b. If you want to change the direction of the plate after you have picked a point, click the left mouse button.

Alternatively, if you want to create a cylindrical bend, click the middle mouse button. In this case the radius 2=radius 1.

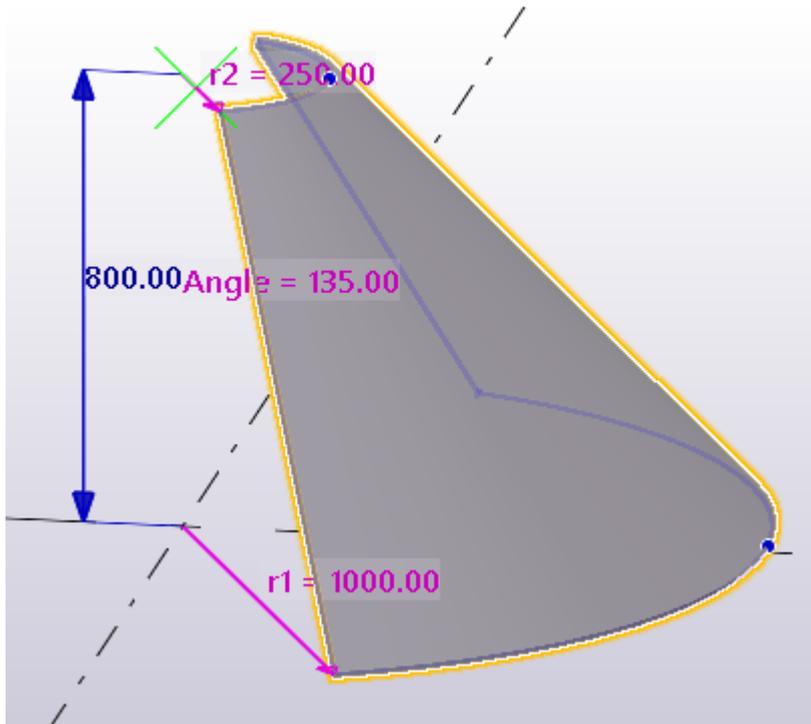


6. To complete the bent plate creation, click the middle mouse button.

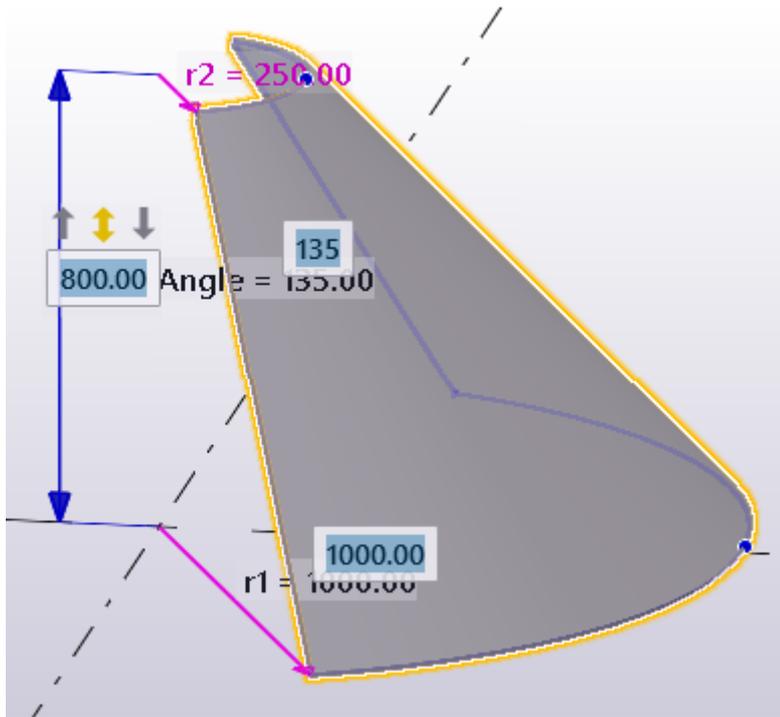


Modify the shape of a stand-alone bent plate

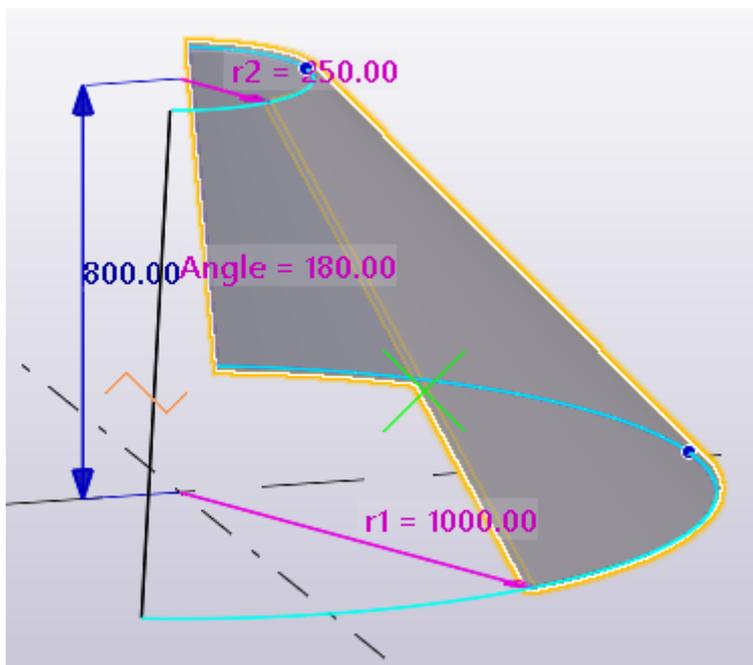
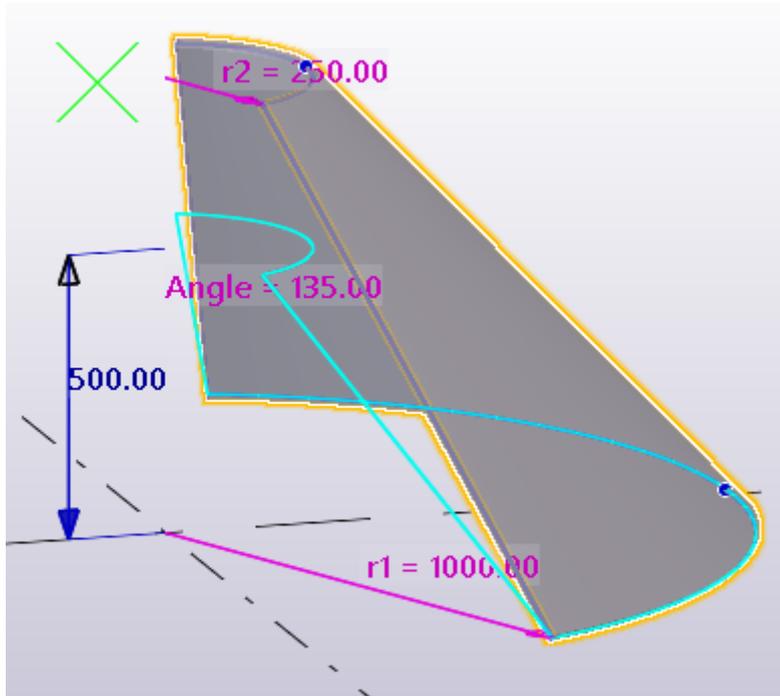
Use direct modification dimension values and handles to modify the bent plate shape.



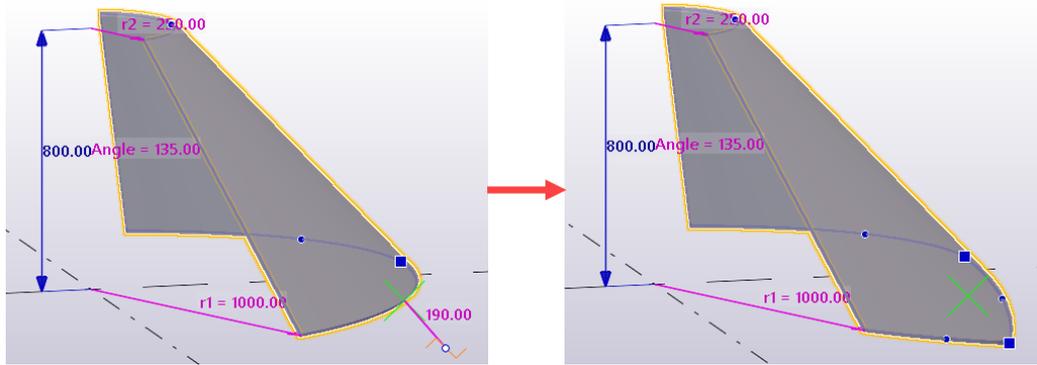
- You can change the angle, radiuses and height of the bend by entering new dimension values.



- You can drag and stretch the edges of the bent plate.



- You can add and delete intermediate points.

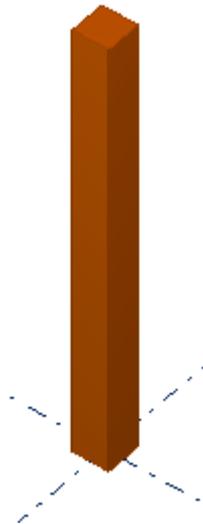


- You can create unfolded drawings of the stand-alone bent plates.

Create a concrete column

1. On the **Concrete** tab, click **Column** .
2. Pick a point.

Tekla Structures creates the column using the **Concrete column** properties in the property pane, and at the level defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete column** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete column properties

1. If the property pane is not open, double-click the column to open the **Concrete column** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Concrete column properties

Use the **Concrete column** properties in the property pane to view and modify the properties of a concrete column. To open the properties, double-click the concrete column. The file name extension of a concrete column property file is *.ccl.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the column. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the column.
Material	Material (page 318) of the column.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group columns. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 312) of the column, relative to column's reference point.
Rotation	Rotation (page 309) of the column around its axis on the work plane.
Horizontal	Horizontal position (page 313) of the column, relative to column's reference point.

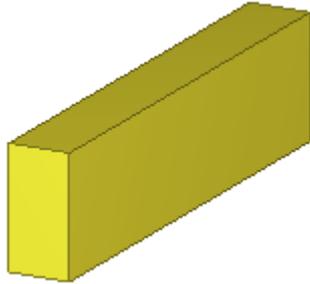
Setting	Description
Top	Position of the second end of the column in the global z direction.
Bottom	Position of the first end of the column in the global z direction.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the column is precast or cast-in-place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp columns using deformation angles.
Cambering	Use to pre-camber (page 331) the column.
Shortening	Use to shorten the column in the model. The true length of the column is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete beam

1. On the **Concrete** tab, click  .

2. Pick two points.

Tekla Structures creates the beam between the points you picked using the **Concrete beam** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete beam** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete beam properties

1. If the property pane is not open, double-click the beam to open the **Concrete beam** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the concrete beam. The file name extension of a concrete beam property file is *.cbm.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	User-definable name of a beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 308) , relative to the beam's reference line.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
At depth	Position depth (page 310) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 315) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 315) perpendicular to the beam's reference line.
Dz	Move the beam end (page 315) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .

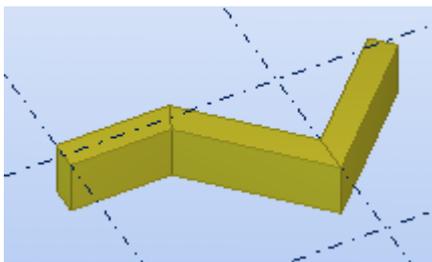
Setting	Description
Cast unit	Indicate whether the beam is precast or cast in place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 331) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete polybeam

A polybeam can contain straight and curved segments.

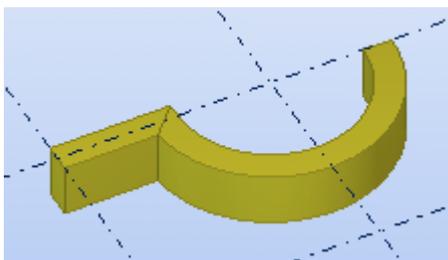
1. On the **Concrete** tab, click **Beam** --> **Polybeam**.
2. Pick the points you want the beam to go through.
3. Click the middle mouse button.

Tekla Structures creates the beam between the points you picked, using the beam's current properties.



4. If you want to create curved segments, chamfer the corners of the polybeam.

For example:



Modify concrete polybeam properties

1. If the property pane is not open, double-click the polybeam to open the **Concrete beam** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Concrete beam properties

Use the **Concrete beam** properties in the property pane to view and modify the properties of a concrete beam or polybeam. To open the properties, double-click the polybeam. The file name extension of a concrete beam property file is *.cbm.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of a beam. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the beam.
Material	Material (page 318) of the beam.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.

Setting	Description
Class	Use to group beams. For example, you can display parts of different classes in different colors.
Position	
On plane	Beam's position on the work plane (page 308) , relative to the beam's reference line.
Rotation	Rotation (page 309) of the beam around its axis on the work plane.
At depth	Position depth (page 310) of the beam. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the beam (page 315) by moving the beam end point along the beam's reference line.
Dy	Move the beam end (page 315) perpendicular to the beam's reference line.
Dz	Move the beam end (page 315) in the z direction of the work plane.
Curved beam	
Plane	Plane of curvature.
Radius	Radius of the curved beam.
Number of segments	Number of segments Tekla Structures uses when drawing the curved beam.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the column is precast or cast in place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Deforming	
Warping	Use to warp beams using deformation angles.
Cambering	Use to pre-camber (page 331) beams.
Shortening	Use to shorten beams in the model. The true length of the beam is decreased in the drawing.

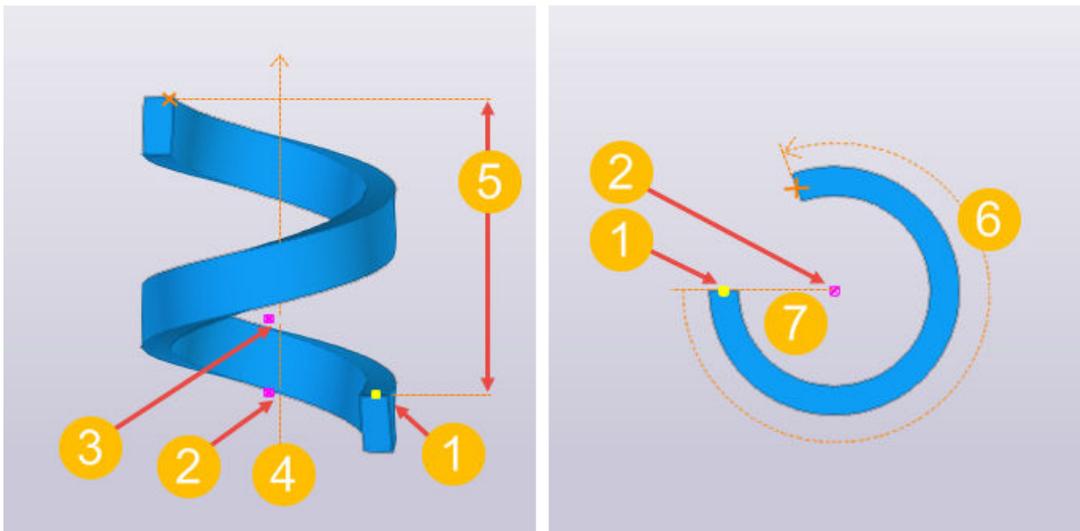
Setting	Description
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete spiral beam

Use the **Create concrete spiral beam** command when you want to model spiral staircases, parking ramps, and complex architectural shapes, for example.

Basic concepts related to spiral beams

The images below illustrate some basic concepts related to the creation of spiral beams. Note that if you change the positioning, the entire geometry of the spiral beam changes.



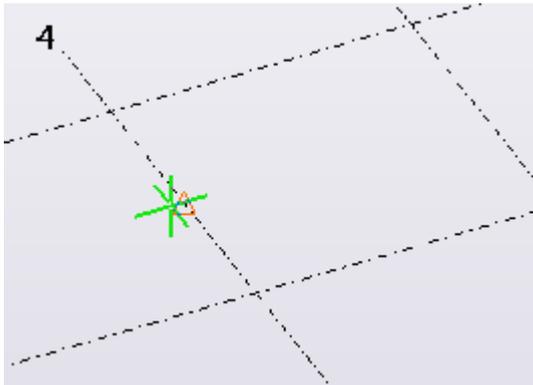
- (1) Start point (the first point picked)
- (2) Center point (the second point picked)
- (3) Direction of the rotation axis (the optional third point picked)
- (4) Center axis
- (5) Total rise: the distance from the start point to the end point, parallel to the center axis

(6) Rotation angle: the rotation angle of the spiral beam, given in degrees.
Note: positive value = counter-clockwise rotation, negative value = clockwise rotation.

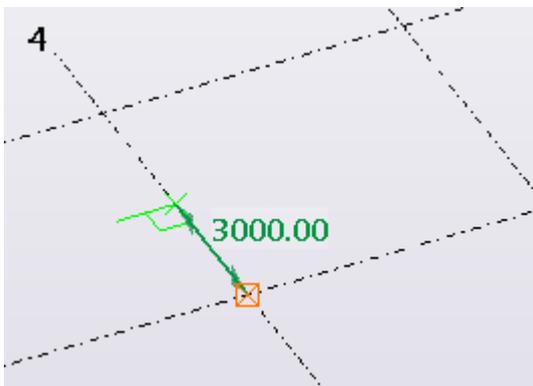
(7) Radius: the distance from the start point to the center point, perpendicular to the center axis

Create a spiral beam

1. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
2. Pick the start point.



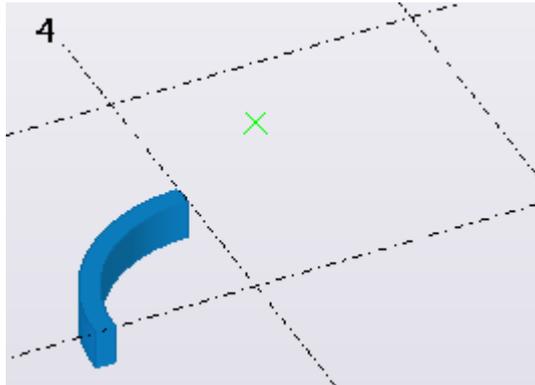
3. Pick the center point.



4. To set the rotation axis in the work plane +Z direction, click the middle mouse button to finish.

NOTE Alternatively, instead of clicking the middle mouse button, you can pick a second center axis point to define the direction of the rotation axis.

Tekla Structures creates the spiral beam. For example:



- Click the spiral beam to select it.

The contextual toolbar appears with the following options:



- Rotation angle
 - Total rise
 - Twist angle at start
 - Twist angle at end
- To add more rotation, enter a bigger value in the **Rotation angle** box.
 - To make the spiral more loose, enter a bigger value in the **Total rise** box.
 - To change the radius, move the start point or center point.

Limitations

- Spiral beam has a single, constant radius.
- Unfolding of spiral beams whose total rise is larger than 0.00 does not produce completely straight results in drawings. The amount of deviation in the part profile outlines and part length depends on several factors: the type, size, and length of the profile; the amount of total rise; and the amount of rotation angle and detailing used.
- Spiral beams are not always untwisted in unfolding. If unequal twisting is applied to the start end and end end, the unfolded drawing shows an unfolded but twisted part as a result.
- Connections and details may not work as expected with spiral beams.
- DSTV export of spiral beams may not produce a correct result.

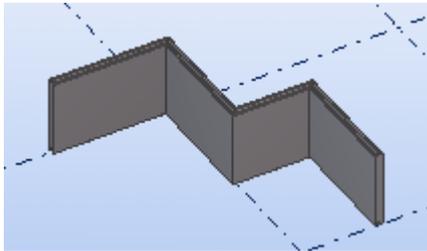
- You cannot export spiral beams as parts in IFC export. If you are modeling cast-in-place structures with spiral beam, you can export the geometry to IFC as pour objects.

Create a concrete panel or wall

You can create a concrete panel or wall that passes through the points you pick.

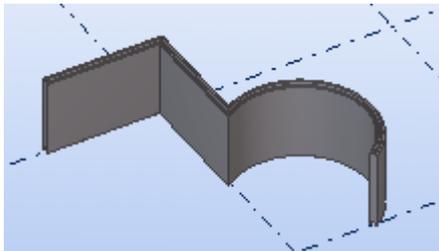
1. On the **Concrete** tab, click **Panel** .
2. Pick the points you want the panel or wall to go through.
3. Click the middle mouse button.

Tekla Structures creates the panel or wall using the **Concrete panel** properties in the property pane



4. If you want to create curved segments, chamfer the corners of the panel or wall.

For example:



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete panel** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify concrete panel or wall properties

1. If the property pane is not open, double-click the panel or the wall to open the **Concrete panel** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Concrete panel or wall properties

Use the **Concrete panel** properties in the property pane to view and modify the properties of a concrete panel or a wall. To open the properties, double-click the panel or the wall. The file name extension of a concrete panel property file is *.cpn.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the panel. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the panel (thickness × height of the wall).
Material	Material (page 318) of the panel.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group panels. For example, you can display parts of different classes in different colors.
Position	
On plane	Panel's position on the work plane (page 308) , relative to the panel's reference line.
Rotation	Rotation (page 309) of the panel around its axis on the work plane.
At depth	Position depth (page 310) of the panel. The position is always perpendicular to the work plane.

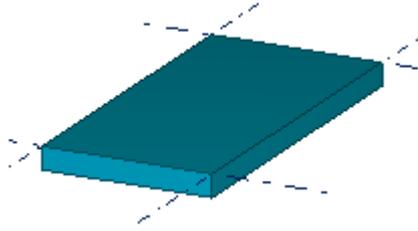
Setting	Description
End offset	
Dx	Change the length of the panel (page 315) by moving the beam end point along the panel's reference line.
Dy	Move the panel end (page 315) perpendicular to the panel's reference line.
Dz	Move the panel end (page 315) in the z direction of the work plane.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the panel or wall is precast or cast in place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved panel.
Number of segments	Number of segments Tekla Structures uses when drawing the curved panel.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a concrete slab

When you create a concrete slab, the profile you select defines the thickness of the slab and the points you pick define the shape. The corners of the slab can be chamfered.

1. On the **Concrete** tab, click **Slab** .
2. Pick the corner points of the slab.
3. Click the middle mouse button.

Tekla Structures creates the slab, using the **Concrete slab** properties in the property pane.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Concrete slab** from the list.

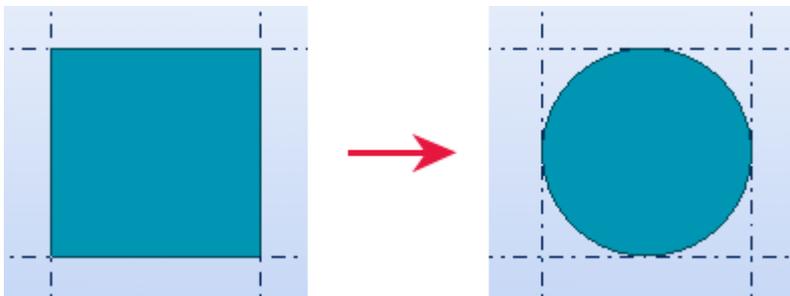
Tekla Structures starts the command and shows the properties in the property pane.

Create a round concrete slab

1. Create a square slab with four equal sides.
2. Select the slab.
3. Select the handles of the slab.

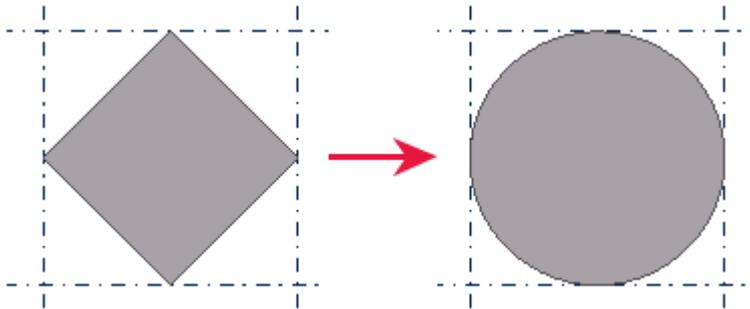
TIP To select all the handles at once, hold down the **Alt** key and drag the mouse from left to right, covering all the handles.

4. Press **Alt + Enter** to display the **Chamfer Properties** dialog box.
5. Select the round chamfer symbol  from the list.
6. Enter the chamfer radius in the **x** box. The radius must be equal to half of the side of the square.
7. Click **Modify**.



Alternative way of creating a round slab

1. Create a diamond-shaped slab with four equal sides.
2. To round the corners, use the arc point chamfer type .



Modify concrete slab properties

1. If the property pane is not open, double-click the slab to open the **Concrete slab** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.
Tekla Structures uses the new properties the next time you create an object of this type.

Concrete slab properties

Use the **Concrete slab** properties in the property pane to view and modify the properties of a concrete slab. To open the properties, double-click the concrete slab. The file name extension of a concrete slab property file is *.csl.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

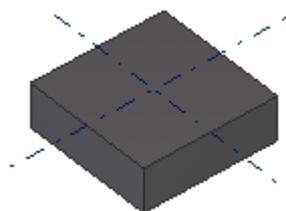
Setting	Description
General	
Name	User-definable name of the slab. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Thickness	Thickness of the slab.
Material	Material (page 318) of the slab.

Setting	Description
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group slabs. For example, you can display parts of different classes in different colors.
Position	
At depth	Position depth (page 310) of the concrete slab. The position is always perpendicular to the work plane.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the slab is precast or cast in place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a pad footing

1. On the **Concrete** tab, click .
2. Pick a point.

Tekla Structures creates the pad footing using the **Pad footing** properties in the property pane, and at the [level \(page 323\)](#) defined in the properties.



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Pad footing** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify pad footing properties

1. If the property pane is not open, double-click the pad footing to open the **Pad footing** properties.
2. [Change \(page 96\)](#) the properties as needed.

For example, to create a circular pad footing, select a circular section for **Profile**.

3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Pad footing properties

Use the **Pad footing** properties in the property pane to view and modify the properties of a pad footing. To open the properties, double-click the pad footing. The file name extension of a pad footing property file is *.cpf.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

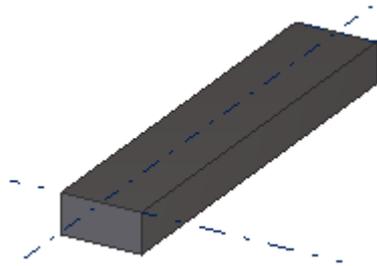
Setting	Description
General	
Name	User-definable name of the pad footing. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the pad footing.
Material	Material (page 318) of the pad footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.

Setting	Description
Class	Use to group pad footings. For example, you can display parts of different classes in different colors.
Position	
Vertical	Vertical position (page 312) of the pad footing, relative to pad footing's reference point.
Rotation	Rotation (page 309) of the pad footing around its axis on the work plane.
Horizontal	Horizontal position (page 313) of the pad footing, relative to pad footing's reference point.
Top	Position of the top surface of the pad footing in the global z direction.
Bottom	Position of the bottom surface of the pad footing in the global z direction.
Cast unit	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the pad footing is precast or cast-in-place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create a strip footing

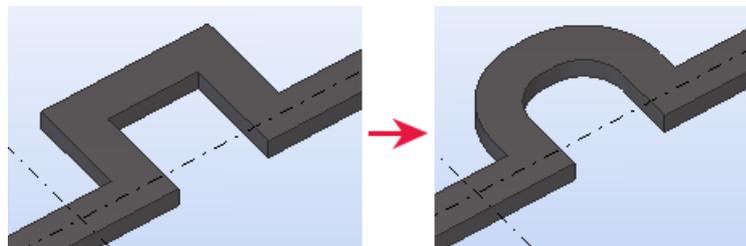
1. On the **Concrete** tab, click **Footing** --> **Strip footing**.
2. Pick the points you want the footing to go through.
3. Click the middle mouse button.

Tekla Structures creates the strip footing between the points you picked using the **Strip footing** properties in the property pane.



4. If you want to create curved segments, chamfer the corners of the footing.

For example:



Alternatively, you can start the command in the property pane.

1. Ensure that you have nothing selected in the model.
2. In the property pane, click the **Object type list** button  and select **Strip footing** from the list.

Tekla Structures starts the command and shows the properties in the property pane.

Modify strip footing properties

1. If the property pane is not open, double-click the strip footing to open the **Strip footing** properties.
2. [Change \(page 96\)](#) the properties as needed.
3. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

Strip footing properties

Use the **Strip footing** properties in the property pane to view and modify the properties of a strip footing. To open the properties, double-click the strip footing. The file name extension of a strip footing property file is *.csf.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the strip footing. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Profile	Profile (page 316) of the strip footing.
Material	Material (page 318) of the strip footing.
Finish	Type of finish. Finish is user-definable. It describes how the part surface has been treated.
Class	Use to group strip footings. For example, you can display parts of different classes in different colors.
Position	
On plane	Strip footing's position on the work plane (page 308) , relative to the footing's reference line.
Rotation	Rotation (page 309) of the strip footing around its axis on the work plane.
At depth	Position depth (page 310) of the strip footing. The position is always perpendicular to the work plane.
End offset	
Dx	Change the length of the strip footing (page 315) by moving the footing end point along the footing's reference line.
Dy	Move the strip footing end (page 315) perpendicular to the beam's reference line.
Dz	Move the strip footing (page 315) in the z direction of the work plane.
Cast unit	

Setting	Description
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the strip footing is precast or cast-in-place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Bending	
Plane	Plane of curvature.
Radius	Radius of the curved strip footing.
Number of segments	Number of segments Tekla Structures uses when drawing the curved strip footing.
Custom properties	
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

Create items

In Tekla Structures, the term *item* refers to the parts that have a 3D *shape*. Shapes are created in an external modeling software, or in Tekla Structures, and they are available in the Tekla Structures shape catalog.

Items are similar to other [parts \(page 235\)](#), such as beams and columns. The main difference between items and other types of parts is that a 3D shape defines the geometry of an item, whereas a part has a 2D profile that is extruded to create the length of the part.

You can use items to model objects that would be difficult to model using basic Tekla Structures parts and commands, such as cutting. You can also use items to model objects that use shapes modeled in an external software or by a manufacturer.

Every item has properties that define it, such as shape, material, and location. If you want to use item properties in view and selection filters or in drawing and report templates, you need to use the template attributes of parts and profiles. If you want to separate items from parts, use the IS_ITEM template attribute.

Limitations to items

- Items have a fixed geometry according to their shape, so items cannot be scaled, stretched, or fitted.
- Items cannot be mirrored.

- Items cannot be split or combined. Splitting an imported item creates a duplicate to the splitting position.
- Items can only be cut or attached to another part if they have a solid shape.
- The gross weight value of an imported item may be different from that of an identical Tekla Structures part modeled with cuts. This is because the cuts are not taken into account when calculating the gross weight of parts.
- The contextual toolbar is not available for items.

Create an item or a concrete item

1. Ensure that  **Direct modification** is switched on.
This makes it easier to adjust the location and rotation of the item in the model.
2. Depending on the material of the item you want to create, do one of the following:

- On the **Steel** tab, click **Item** .
- On the **Concrete** tab, click **Item** .

TIP Alternatively, you can start the command in the property pane.

- a. Ensure that you have nothing selected in the model.
- b. In the property pane, click the **Object type list** button  and select **Item** or **Concrete Item** from the list.

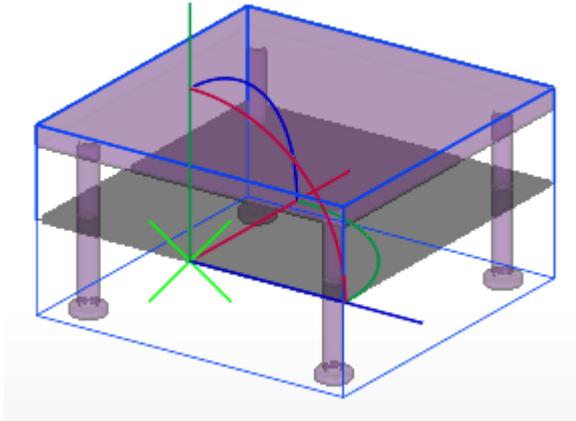
Tekla Structures starts the command and shows the properties in the property pane.

In the property pane, you can modify the item properties and select the shape, for example.

3. Move the mouse pointer over object faces and edges in the model, and see how the item turns over and adjusts to the object faces.
If you are adding the item to another object (such as part or pour object), Tekla Structures also shows location dimensions to the nearest object edges.
4. Pick the first point of the item.
5. Pick another point to indicate the direction of the item.
Tekla Structures places the item between the points you picked starting from the first point (yellow handle) towards the direction of the second

point (magenta handle), using the **Item** or **Concrete Item** properties in the property pane.

Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the item. The handles are red, green, and blue, according to the local coordinate system of the item.



6. To move the item along any of its coordinate axes, drag the relevant axis handle to a new location.
7. To rotate the item around any of its coordinate axes, drag the relevant rotation handle to a new location.

Press **Tab** to rotate the item in 90-degree steps in the direction of the selected rotation handle.

8. To move or rotate the item by specifying a distance or angle:
 - a. Select an axis handle, a rotation handle, or a dimension arrowhead.
 - b. Type the value by which you want the dimension to change.

When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.

- c. Click **OK** to confirm the new dimension.
9. If you want to add more items to the model, click the middle mouse button and repeat steps 3–8.
10. To stop adding and modifying items, press **Esc**.

Modify item or concrete item properties

1. If the property pane is not open, double-click the item or the concrete item to open the **Item** or the **Concrete Item** properties.
2. [Change \(page 96\)](#) the properties as needed.

3. Click **Modify** to apply the changes.
Tekla Structures uses the new properties the next time you create an object of this type.

Change the shape of an item

When you create or modify an item, you can select the shape from the list that contains all the shapes available in the shape catalog.

Before you start, ensure that you have the required shape imported to the shape catalog.

1. Double-click an item to open the item properties in the property pane.
2. Click the ... button next to the **Shape** box to open the **Shape Catalog** dialog box.
3. If needed, use the **Filter** box to search for a shape.
4. Select a shape from the list.
5. Click **OK** to close the **Shape Catalog** dialog box.
6. Click **Modify** to apply the changes.

Item and concrete item properties

Use the **Item** and the **Concrete Item** properties in the property pane to define, view, and modify the properties of an item. To open the properties, double-click the item or the concrete item.

The file name extension of an item property file is *.ips.

The file name extension of a concrete item property file is *.ipc.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of an item. Tekla Structures uses part names in reports and in Document manager , and to identify parts of the same type.
Shape	Shape of an item. To select a shape from the shape catalog, click the ... button next to the Shape box. To show the item shape in reports and drawing tables, use the PROFILE template attribute.

Setting	Description
Material	Material (page 318) of the item.
Finish	Type of finish. Finish is user definable. It describes how the part surface has been treated, e.g. with anti-corrosive paint, hot galvanized, fire retardant coating, etc.
Class	Use to group items. For example, you can display parts of different classes in different colors.
Numbering series (available for items)	
Part numbering	Part prefix and start number for the part position number (page 628) .
Assembly numbering	Assembly prefix and start number for the assembly position number (page 628) .
Position	
On plane	Item's position on the work plane (page 308) , relative to the item's reference line.
Rotation	Rotation (page 309) of the item around its axis on the work plane.
At depth	Position depth (page 310) of the item. The position is always perpendicular to the work plane.
End offset	
Dx	Move the item along (page 315) its reference line.
Dy	Move the item perpendicular (page 315) to its reference line.
Dz	Move the item in the z direction (page 315) of the work plane.
Cast unit (available for concrete items)	
Cast unit numbering	Part prefix and start number for the part position number (page 628) .
Cast unit	Indicate whether the item is precast or cast-in-place.
Pour phase	Pour phase (page 407) of the cast-in-place parts. Use to separate pour objects from one another.
Custom properties	

Setting	Description
More	Click the More button to open the user-defined attributes (page 319) (UDAs) of the part. UDAs provide additional information about the part.

2.2 Adjust part position and show part information

When you create a part, you position the part by picking points. If needed, you can adjust the position of a part in different ways after the part creation.

Part position

In part creation, the part handles and part reference line help you to define the part position. You can adjust the [position of parts \(page 306\)](#), such as [rotation \(page 309\)](#), by using the **Position** and **End offset** sections in the property pane, or by using the contextual toolbar.

Also, check the following tips that help you to create and position, for example, curved and horizontal parts:

- [Create curved parts \(page 321\)](#)
- [Create horizontal parts \(page 322\)](#)
- [Create beams close to each other \(page 323\)](#)
- [Position columns, pad footings, and orthogonal beams \(page 323\)](#)
- [How to model identical areas \(page 324\)](#)

Show part information

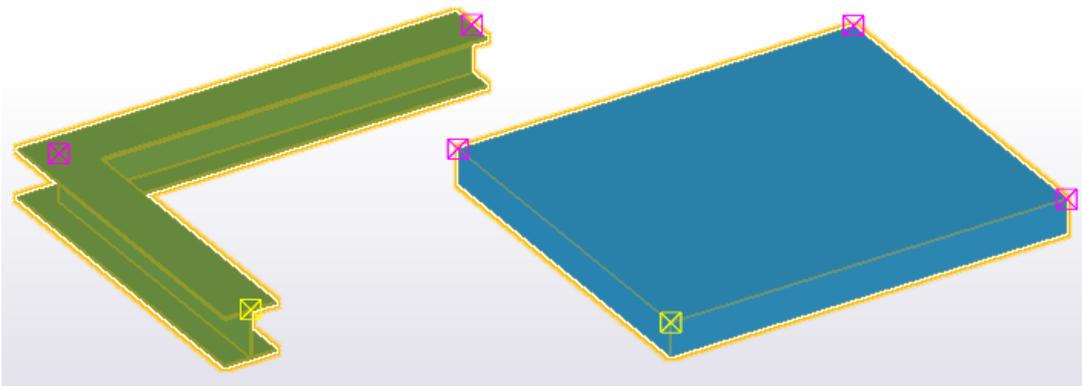
To display selected part properties in the model view, use [part labels \(page 320\)](#). Part labels are textual descriptions that are displayed next to the part whose properties they show.

Show part handles and part reference lines in a model view

Part *handles* can be used to move a model object or to modify the model object's shape or size. Part *reference line* is line between two reference points, and the reference line has handles at the line ends.

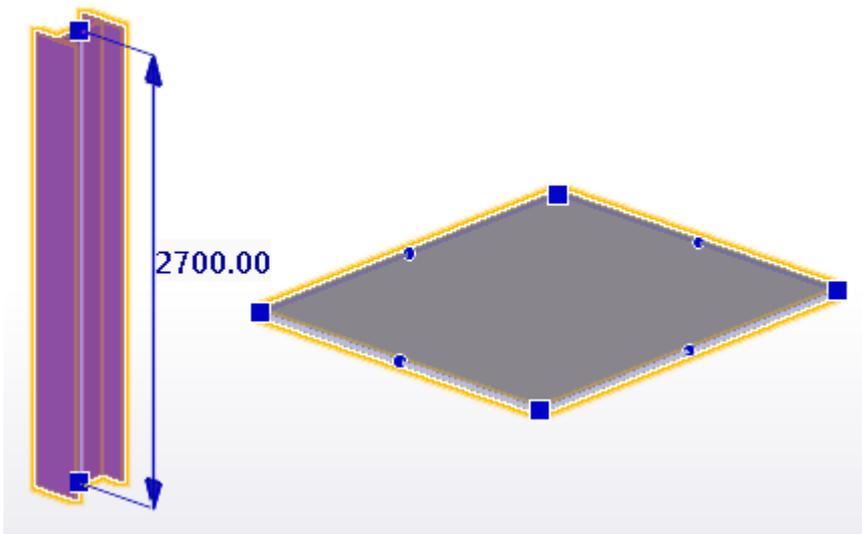
Show part handles

Tekla Structures indicates the direction of a part with handles. When you select a part, Tekla Structures shows the handles. The handle of the first end point is yellow, the rest are magenta.



For information on how to select only the handles of a part, see [Select objects \(page 116\)](#).

If [direct modification \(page 103\)](#) is on, Tekla Structures also displays direct modification handles for the reference points, corners, segments, and segment midpoints of the selected part. These handles are blue.



Modify the length of a part using handles

If you do not want to use direct modification, you can use the part handles to modify the length of a part.

1. Ensure that the **Direct modification** switch  is **not** active.
2. Select the part.
Tekla Structures highlights the handles of the part.
3. Click one of the handles to select it.
4. Move the handle like any other object in Tekla Structures.

For example, right-click and select **Move**.

If **Drag & drop** is [active \(page 147\)](#), just drag the handle to a new location.

WARNING Do not use cuts or [fittings \(page 361\)](#) to change the length of a part, for the following reasons:

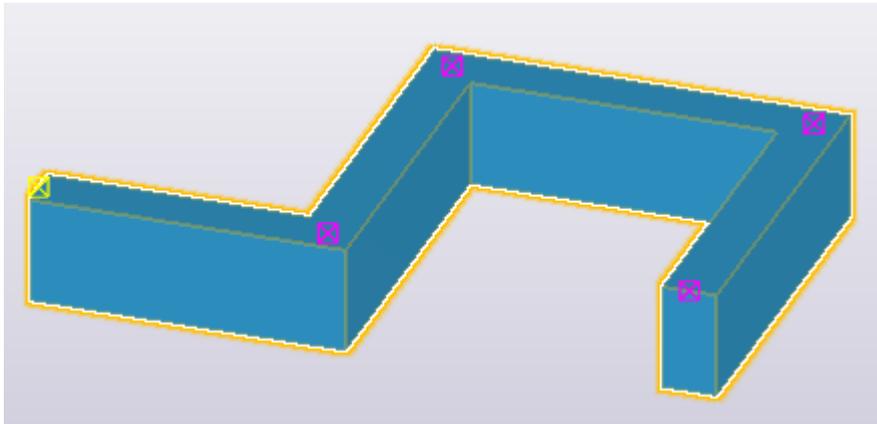
- Cuts may cause shop errors, because cuts do not always affect part length when you export information to NC files.
 - Fittings may cause problems with connections and details.
-

Swap handles

You can change the modeling direction of a part by using the **Swap handles** macro. This changes the yellow start handle to magenta, and the other way around.

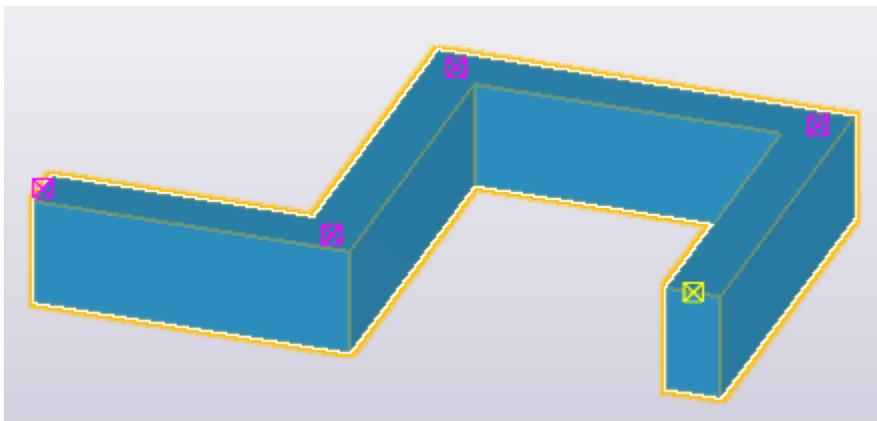
1. Select the part whose modeling direction you want to change.

Tekla Structures shows the part handles.



2. Go to **Quick Launch**, start typing `swap handles`, and select the **Macro.Swap handles** command from the list that appears.

Tekla Structures changes the modeling direction of the part, and swaps the start and end handles.

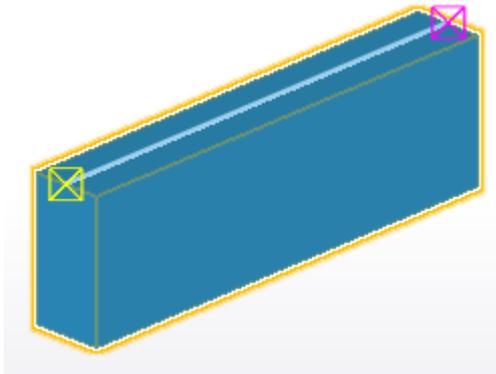


Show part reference lines in a model view

When you create a part, you position the part by [picking points \(page 77\)](#). These points are the part reference points. If you pick two points to position a part, the points form a part reference line, and handles appear at the line ends. By default, the part reference line is invisible in the model. It may be useful to display it when snapping to the middle points of parts, for example.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. On the **Advanced** tab, select the **Part reference line** check box.
4. Click **Modify**.

The part reference lines are displayed.



Modify the position of a part

Use the property pane and its **Position** and **End offset** sections to modify the part position. Alternatively, you can use the contextual toolbar to modify the part position.

To modify the position of a part, do one of the following:

To	Do this
Modify part position using the property pane (page 96)	<ol style="list-style-type: none">1. Double-click a part to open the part properties in the property pane.2. In the Position or in the End offset section, modify the desired position settings, such as

To	Do this
	<p>part rotation (page 309) or part vertical position (page 312).</p> <p>For example, you can define the part to be positioned 200 units above its handles.</p> <p>3. Click Modify.</p>
<p>Modify part position using the contextual toolbar</p>	<ol style="list-style-type: none"> 1. Click  in the contextual toolbar. 2. Modify the settings. The object moves in the model accordingly. <ul style="list-style-type: none"> • To change the overall position of a beam, column, panel, or footing, use the round selection dial. Click a sector in the dial to select a position. • To change the rotation angle, click and drag the green rotation angle knob. • To change the Angle, Plane offset, or Depth offset, enter a value in the corresponding box. <div data-bbox="959 1182 1236 1395" data-label="Image"> </div> <ul style="list-style-type: none"> • To change the position of a plate or slab, select an option and enter a value in the Depth offset box. <div data-bbox="959 1576 1161 1756" data-label="Image"> </div>

TIP The rotation angle knob snaps to every 5 degrees. Hold down **Shift** to override this.

See also

[Part position on the work plane \(page 308\)](#)

[Part rotation \(page 309\)](#)

[Part position depth \(page 310\)](#)

[Part vertical position \(page 312\)](#)

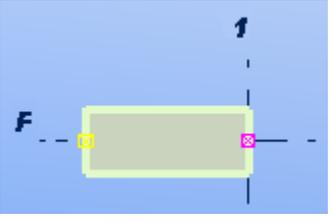
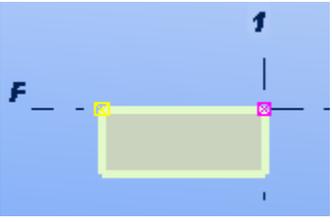
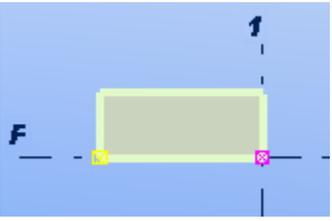
[Part horizontal position \(page 313\)](#)

[Part end offsets \(page 315\)](#)

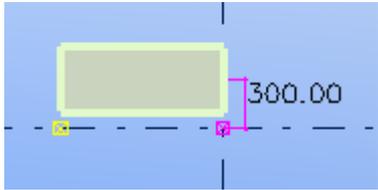
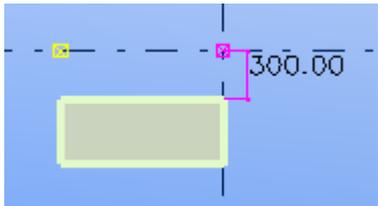
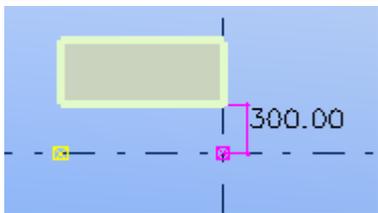
Part position on the work plane

Use the **On plane** setting in the part properties to view and change the part's position on the work plane. The position is always relative to part's reference line.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

Option	Description	Example
Middle	The reference line is in the middle of the part.	
Right	The part is positioned underneath the reference line.	
Left	The part is positioned above the reference line.	

Examples

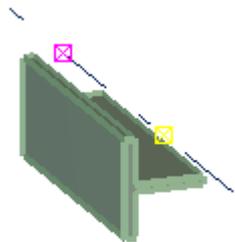
Position	Example
Middle 300	
Right 300	
Left 300	

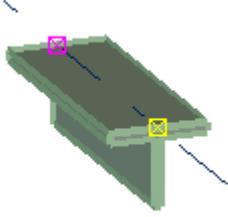
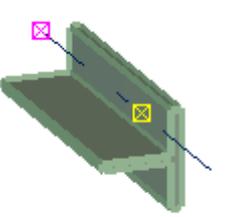
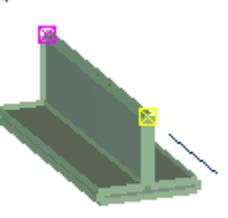
Part rotation

Use the **Rotation** setting in the part properties to view and change the rotation of a part around its axis on the work plane.

You can also define the angle of rotation. Tekla Structures measures positive values clockwise around the local x axis.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

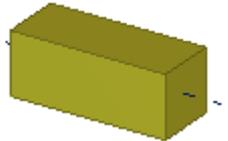
Option	Description	Example
Front	The work plane is parallel to the front plane of the part.	

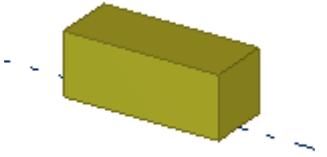
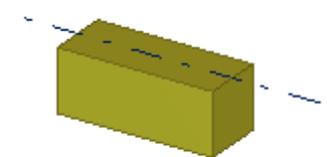
Option	Description	Example
Top	The work plane is parallel to the top plane of the part.	
Back	The work plane is parallel to the back plane of the part.	
Below	The work plane is parallel to the bottom plane of the part.	

Part position depth

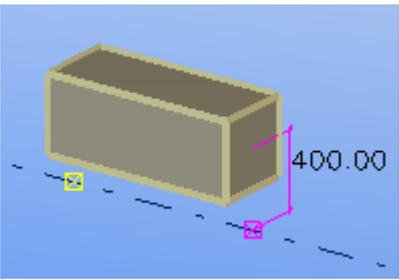
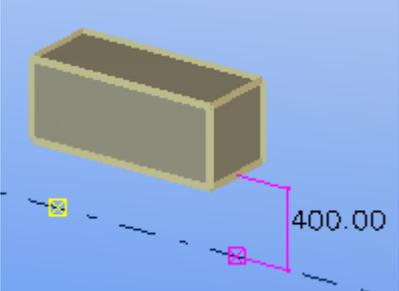
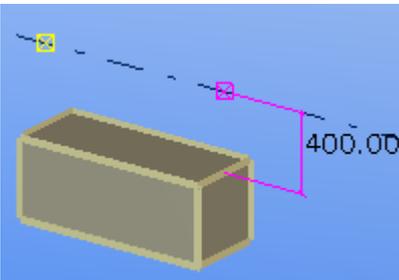
Use the **At depth** setting in the part properties to view and change the position depth of the part. The position is always relative to the part reference line between the part handles.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

Option	Description	Example
Middle	The part is positioned in the middle of the reference line.	

Option	Description	Example
Front	The part is positioned above the reference line.	
Behind	The part is positioned underneath the reference line.	

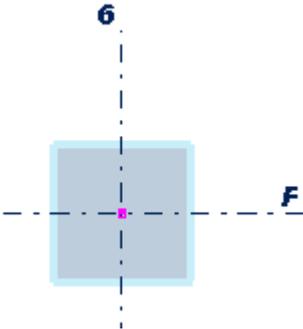
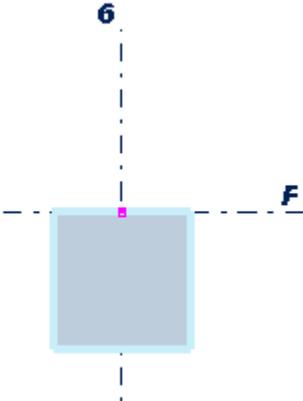
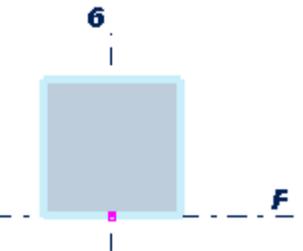
Examples

Position	Example
Middle 400	
Front 400	
Behind 400	

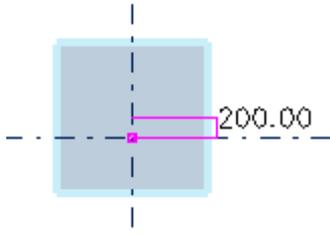
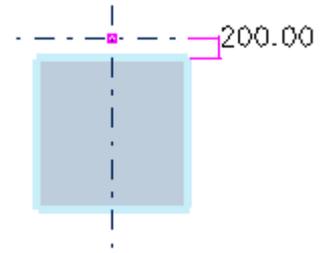
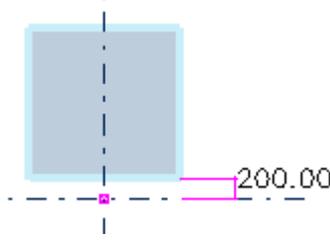
Part vertical position

Use the **Vertical** setting in the part properties to view and change the vertical position of the part. The position is always relative to the part's reference point.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

Option	Description	Example
Middle	The reference point is in the middle of the part.	
Down	The part is positioned underneath the reference point.	
Up	The part is positioned above the reference point.	

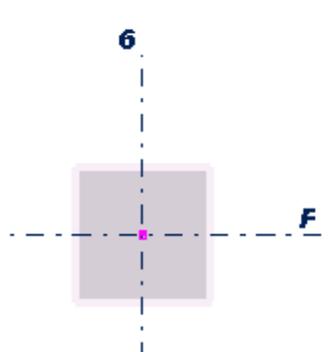
Examples

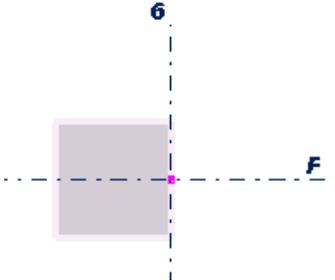
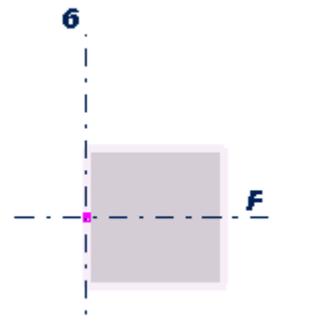
Position	Example
Middle 200	
Down 200	
Up 200	

Part horizontal position

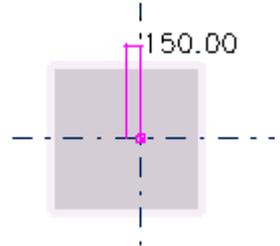
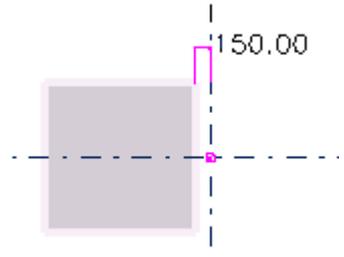
Use the **Horizontal** setting in the part properties to view and change the horizontal position of the part. The position is always relative to the part's reference point.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

Option	Description	Example
Middle	The reference point is in the middle of the part.	

Option	Description	Example
Left	The part is positioned on the left side of the reference point.	
Right	The part is positioned on the right side of the reference point.	

Examples

Position	Example
Middle 150	
Left 150	

Position	Example
Right 150	

Part end offsets

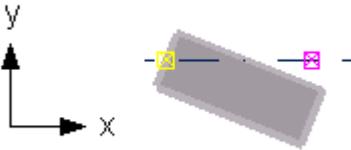
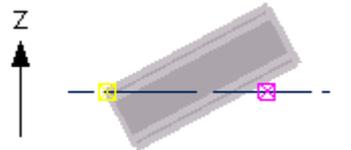
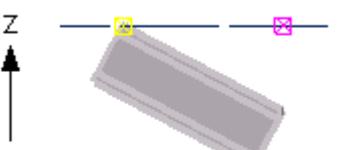
Use the **Dx**, **Dy** and **Dz** settings in the part properties to move the ends of a part, relative to its reference line. You can enter positive and negative values.

Alternatively, you can use the [contextual toolbar \(page 306\)](#) to modify the part's position.

Option	Description
Dx	Changes the length of the part by moving the part end point along the reference line.
Dy	Moves the part end perpendicular to the reference line.
Dz	Moves the part end in the z direction of the work plane.

Examples

Position	Example
Dx End point: 200	
Dx End point: -200	
Dy End point: 300	

Position	Example
Dy End point: -300	
Dz End point: 400	
Dz End point: -400	

Select and change the profile or material of a part

Each part has a profile and a material, which are selected using the profile catalog and the material catalog.

Select and change the profile of a part

There are two types of profiles available for parts in Tekla Structures:

- *Fixed profiles*



Fixed profiles are profiles that can be obtained pre-manufactured. The properties of the fixed profiles conform to industry standards, and you should not modify them unless you are an administrator. Fixed profiles are environment-specific.

- *Parametric profiles*



Parametric profiles are partly user-definable: they have a predefined shape but you can change their cross section dimensions using one or more parameters. Tekla Structures calculates the cross section shape each time you open the model.

You can use the fixed or parametric profiles available in the Tekla Structures profile catalog, or you can customize the profile catalog in several ways.

Change the profile of a part

When you [create or modify a part \(page 235\)](#), you can select the profile of a part from a list that contains all the profiles available in the profile catalog.

1. Double-click a part to open the part properties in the property pane.
2. Click the ... button next to the **Profile** box.

The **Select Profile** dialog box appears.

By default, only the profile types that are relevant to the material of the part are shown. For example, if you are changing the profile of a steel part, only the profile types that are associated with steel are shown.

3. If needed, define what profile information you want to see.
 - To display all the profiles of the profile catalog in the list, regardless of the material the profile types are associated with, select the **Show all profiles** check box.
 - To see all the properties of profiles, select the **Show details** check box.
4. Select a profile from the list.
5. If the profile is parametric, define its dimensions on the **General** tab.

Property	Symbol	Value	Unit
Height	h	300.00	mm
Web thickness	s	15.00	mm
Flange thickness	t	20.00	mm
Width	b		mm

1

(1) Click the **Value** box and replace the existing value with a new one.

6. Click **OK** to close the **Select Profile** dialog box.
7. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

Use standardized values for profile dimensions

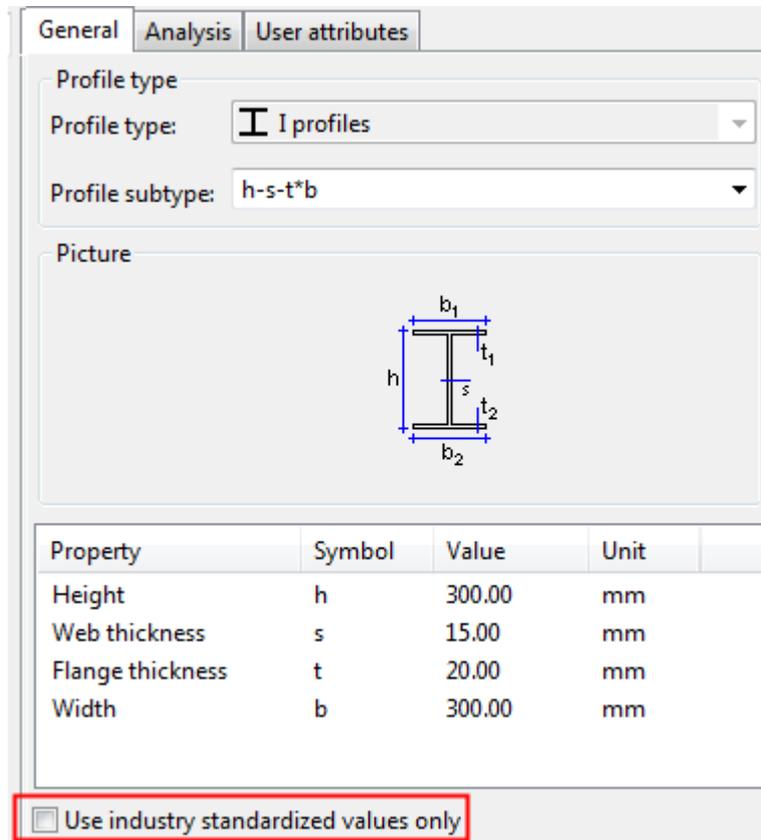
You can use standardized values for the dimensions of parametric profiles.

1. Double-click a part to open the part properties in the property pane.
2. Click the ... button next to the **Profile** box.

The **Select Profile** dialog box appears.

3. Select a parametric profile.

If standardized values have been defined for this profile, the **Use industry standardized values only** check box appears on the **General** tab under the profile properties:



4. Select the **Use industry standardized values only** check box.
5. Select the profile dimensions from a list in the **Value** column.

Select and change the material of a part

When you [create or modify a part \(page 235\)](#), you can select the material and grade of a part from a list that contains all the materials available in the material catalog.

1. Double-click a part to open the part properties in the property pane.
2. Click the ... button next to the **Material** box.
The **Select Material** dialog box appears.
3. If needed, define what material information you want to see.

- To include aliases for material grades in the list, select the **Show aliases** check box.

Aliases are alternative names. For example, they can be former names, or names used in different countries or standards. Tekla Structures

automatically changes aliases into the standard name when you select a material grade.

- To see all the properties of materials, select the **Show details** check box.
4. Select a material from the list.
 5. Click **OK** to close the **Select Material** dialog box.
 6. Click **Modify** in the property pane.

Alternatively, if you know the name of the profile, you can enter it directly in the **Profile** box in the property pane or in the contextual toolbar.

TIP If needed, you can customize the material catalog.

Examples of user-defined attributes (UDAs) for parts

User-defined attributes (UDAs) provide additional information about a part. UDAs can consist of numbers, text, or lists. If needed, you can define new user-defined attributes.

The following table gives some examples of what you can use part UDAs for:

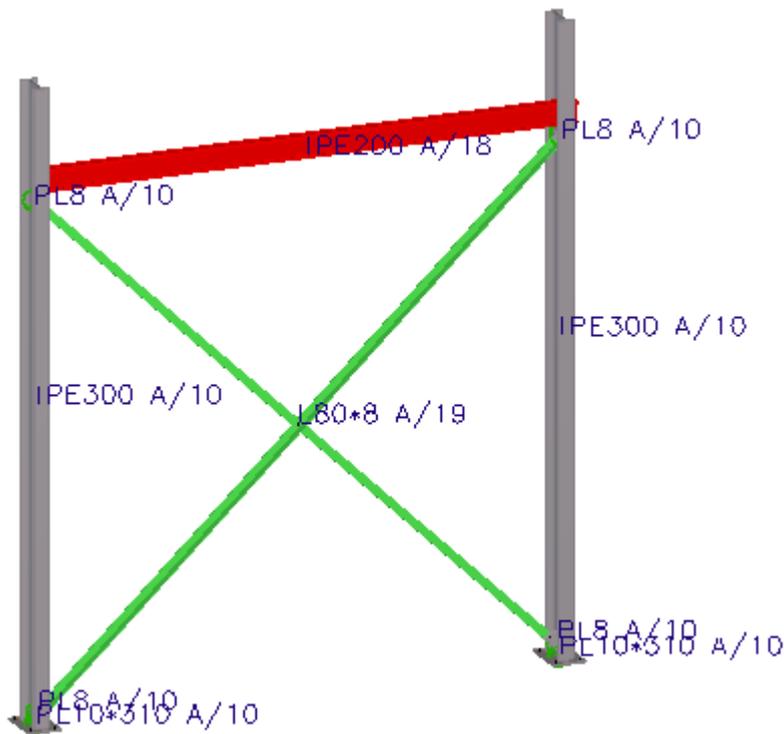
Attribute	Can be used...
Comment	In part and weld marks in Tekla Structures drawings, or in projects.
Shorten	When drawings of the parts are created, Tekla Structures decreases the true length of the part by this value. This is useful when creating assembly drawings of bracing bars that should always be under tension.
Camber	In part marks in Tekla Structures drawings.
Preliminary mark	To obtain preliminary numbers for parts in reports.
Locked	To protect objects from being accidentally changed.
Shear, Tension, and Moment	To save reaction forces for AutoDefaults. You can enter forces separately for each end of a part.
User field 1...4	User-defined fields. You can change the names of these fields and add new user-defined fields.
Connection code	When importing information on connection types into Tekla Structures. You can then use the connection codes as rules in AutoConnection and AutoDefaults. Each end of a part can have a different connection code.

Attribute	Can be used...
Moment connection	For selecting whether to show moment connection symbols in drawings or not.

Show part information by using part labels

You can display selected part properties, user-defined attributes and template attributes in a model view by using *part labels*.

Part labels are textual descriptions that are displayed next to the part they represent. You can define what information to display in the labels, such as the name, profile, and position number of the part.



1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Go to the **Advanced** tab.
4. Next to the **Part label**, in the **Properties** list, select a property.
5. Click **Add >** to add the property to the **Part label** list. The **Part label** check box is automatically selected when you add properties.

6. To remove a property from the **Part label** list, select a property and click **Remove**. The **Part label** check box is automatically cleared if you remove all the properties.
7. If needed, define which user-defined attribute or template attribute you want to display in part labels.
 - a. Select **User-defined attribute** in the **Properties** list.
 - b. Click **Add >**. The **Part label** dialog box appears.
 - c. Enter the user-defined attribute name exactly as it appears in the objects.inp file, or the template attribute name. For example, PRELIM_MARK.
 - d. Click **OK**.
8. In the **Show for** list, select for which parts you want to show the part labels.
 - **All**: Part labels are shown for all parts in the view.
 - **Selected**: Part labels are only shown for the parts you select.
 - **Main part for selected**: Part labels are only shown for the main parts of the assemblies you select.
 - **Main part for all**: Part labels are shown for all main parts of all assemblies.

Note that if you select the **Selected** or the **Main part for selected** option, you need to first apply the changes to the view when the view is selected. Then continue to select the objects for which you want to show the part labels.
9. Click **Modify**.

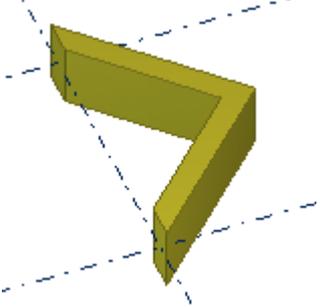
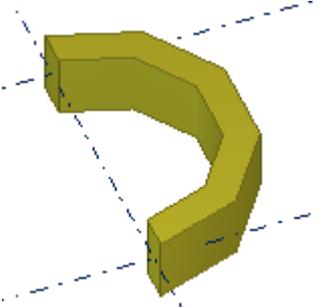
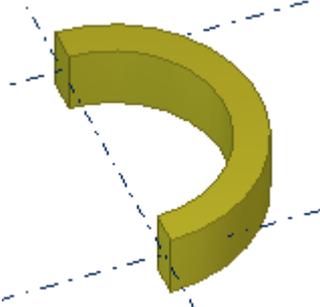
Create curved parts

You can create curved parts by defining a radius and the number of segments for a part. The number of segments determines how realistic the curved part looks: the more segments, the less angular the part appears.

1. Create a part that can be bent: a beam, panel, or strip footing.
2. Double-click the part to open the part properties.
3. Go to the **Curved beam** or **Bending** settings area, depending on the part type.
4. In the **Radius** box, enter the radius.
5. In the **Number of segments** box, enter the number of segments you want to use.

6. If needed, define the plane of curvature, which is relative to the current work plane.
7. Click **Modify** to bend the part.

Examples

Number of segments: 2	
Number of segments: 5	
Number of segments: 15	

See also

[Modify the position of a part \(page 306\)](#)

Create horizontal parts

When creating horizontal parts, such as beams, always pick points in the same direction. For example, pick positions from left to right, and from bottom to top (in positive x, y directions). This ensures that Tekla Structures places and

dimensions the parts in the same way in drawings, and that part marks automatically appear at the same part end.

To ensure that beam rotation is correct in drawings, set part **Rotation** to **Top** in the part properties.

Create beams close to each other

When you create beams so that they are located very close to each other, Tekla Structures may consider them as a twin profile. To prevent this from happening, use the user attribute `MAX_TWIN_SEARCH_DIST` in the profile catalog.

1. On the **File** menu, click **Catalogs** --> **Profile catalog** to open the **Modify Profile Catalog** dialog box.
2. Select the desired profile in the profile tree.
3. Go to the **User attributes** tab and set the property **Twin profile detection distance** to a larger value than 0, for example to 0.1.
4. Click **OK**.
5. Create the beams using the profile.

See also

[Create a steel beam \(page 239\)](#)

[Create a twin profile \(page 246\)](#)

Position columns, pad footings, and orthogonal beams

For parts that you create by picking only one point (such as columns), you can define the top and bottom level of the part in the global z direction. The part is created at the defined level, **not** at the level you picked in the model. This can be useful when creating multi-story structures, as you can define exact levels for each part you create.

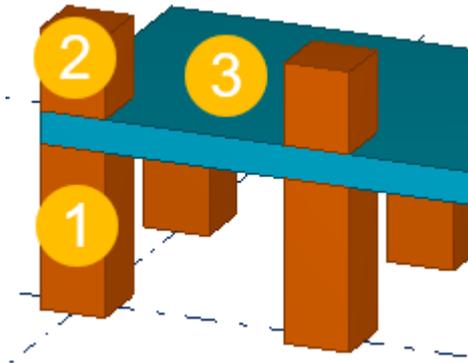
To define the top and bottom levels of a part:

1. Create a part that requires you to pick only one point.
For example, a column.
2. Double-click the part to open the part properties.
3. Go to the **Position** settings area.
4. Modify the top and bottom levels of the part.
 - **Top:** Use to define the top level of the part.
 - **Bottom:** Use to define the bottom level of the part.

5. Click **Modify**.

Example

In this example, the concrete columns form a two-story structure. To position the upper columns correctly, you must modify their bottom level position.



(1) Top level = 1000, Bottom level = 0

(2) Top level = 1700, Bottom level = 1200

(3) Slab thickness = 200

See also

[Modify the position of a part \(page 306\)](#)

How to model identical areas

Most structures contain identical areas, from simple frames to entire floors. Save time by modeling these areas once, then copying them throughout the model. For example, create a column with a base plate and cap plate, then copy the column to all the locations where it occurs in the model.

You can use this technique to create and reproduce any identical area. Depending on the project, you may even be able to add connections before copying the area of the building.

TIP For a project that has several identical floors, try modeling an entire floor, then copying it to several levels.

See also

[Copy and move objects \(page 131\)](#)

2.3 Modify parts

This section explains how to modify different part properties, such as shape, position and length of a part. It also explains how to split and combine parts, and how to use the deforming options to warp and camber parts.

Click the links below to find out more:

[Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts \(page 325\)](#)

[Split parts \(page 326\)](#)

[Combine parts \(page 327\)](#)

[Attach parts to each other \(page 327\)](#)

[Warp a part \(page 329\)](#)

[Camber a part \(page 331\)](#)

Modify the adaptivity of reinforcement, surface treatment, or edge chamfers in parts

Reinforcement, surface treatment and edge chamfers adapt to the parts they are linked to. For example, reinforcement, surface treatment and edge chamfers automatically adapt to changes in part geometry and size. You can modify the adaptivity settings either for the entire model or for each model object separately. If you modify the adaptivity of individual model objects, these modifications override the default settings that you may have defined for the entire model.

The options are:

- **Off:** adaptivity is not defined
- **Relative:** handles retain their relative distances to the nearest part faces in relation to the part's overall size
- **Fixed:** handles retain their absolute distances to the nearest part faces

Define default adaptivity settings

You can define default adaptivity settings that affect the entire model.

1. On the **File** menu, click **Settings** --> **Options** , and go to the **General** settings.
2. Under **Default adaptivity**, select one of the options.
3. Click **OK** to save the changes.

Modify the adaptivity of an individual model object

You can modify the adaptivity settings for each reinforcement or surface treatment separately. These modifications override the default settings that you may have defined for the entire model.

1. In the model, select the [reinforcement \(page 512\)](#) or [surface treatment \(page 373\)](#) whose adaptivity settings you want to change.
2. Right-click, select **Adaptivity**, and then select one of the options.

Split parts

Use splitting to split a part into two. You can use splitting with straight parts, polybeams and curved beams without offsets, and normal and tapered reinforcing bar groups. You can also split plates and slabs by using a polygon.

Split a straight or curved part or polybeam

1. On the **Edit** tab, click **Split**.
2. Select the part you want to split.
3. Pick a point for the dividing line.
4. If you split a polybeam, check that the following are correct:
 - The position and orientation settings of the split polybeams
 - The components related to the split polybeams

Split a plate or slab using a polygon

1. Ensure that the Z axis is perpendicular to the plate or slab you want to split.
2. On the **Edit** tab, click **Split**.
3. Select the part you want to split.
4. Pick positions to outline the polygon to be used for splitting.
5. Click the middle mouse button to close the polygon and to split the part.

NOTE When you pick the corner points of the polygon to be used for splitting, make sure the starting and end points are:

- outside of the part, and
- on the same side of the part.

NOTE If you split contour plates that have bolts, welds or surface treatments, check the result after splitting.

Combine parts

You can combine two parts into one. This can be useful when you want to model complex parts (such as folded plates) that are otherwise difficult to model, or when you want to model prefabricated parts that are delivered to the workshop already attached to profiles.

1. On the **Edit** tab, click **Combine**.

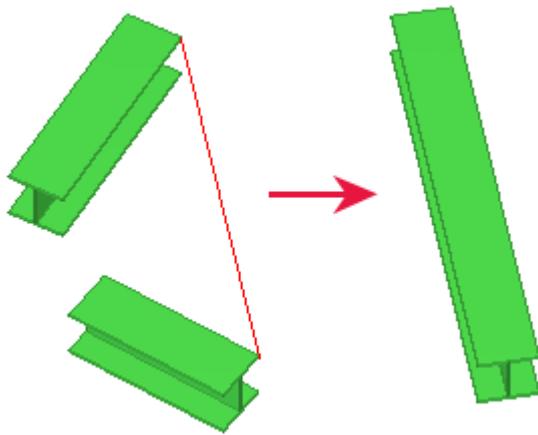
2. Select the first part.

The properties of the first selected part will be used for the combined part.

3. Select the second part.

The parts are combined into one.

If the center lines of the parts are not in line with each other, Tekla Structures combines them by taking the largest distance between the start and end points from both parts. For example:



Limitations

- Combining does not work for contour plates, polybeams or slabs.
- When you combine parts, Tekla Structures retains the attached objects and connections. Tekla Structures does not recreate connections in the part that was selected first.

See also

[Attach parts to each other \(page 327\)](#)

Attach parts to each other

You can attach one or more parts to another part, or detach or explode the attached parts using the **Added material** commands.

When you modify the properties of attached parts, note that some of the part properties are taken from the main part. These properties are not shown in the properties of the attached part. You can inquire the properties of the whole part and the properties of each attached part separately. The attached parts are taken into account when calculating area, volume, and weight:

- **Weight (Gross)** compares the weight with fittings and without fittings, and shows the biggest weight result without cuts and with attached parts.
- **Weight (Net)** shows the weight with cuts and attached parts based on the geometry volume of the modeled part.
- **Weight** shows the net weight.

Limitations

- Connections must be added to the part to which other parts have been attached. You cannot add connections to an attached part.
- All reinforcement components may not work correctly with parts that have been attached to each other using the **Added material** commands. The geometry of the parts is not always kept suitable for adding a component. For example, the reference points of the attached part may be lost and therefore the orientation information needed for adding the reinforcement is not known anymore.

Attach a part to another part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material --> Attach to part** .
3. Select the part to attach to.
4. Select the part you want to attach.
You can attach more than one part at a time.
5. Click the middle mouse button to attach the part.

Detach an attached part

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material --> Detach from part** .
3. Select the attached part you want to detach.
You can detach more than one part from several different parts at a time. Select the parts either by clicking them or by using area selection.

4. Click the middle mouse button to detach the part.
The detached part keeps the color it had as an attached part.

Explode attached parts

You can explode a part that has attached parts.

1. Double-click the view to open the view properties, click the **Display...** button, and ensure that the **Cuts and added material** option is selected in the display settings.
2. On the **Edit** tab, click **Added material --> Explode part** .
3. Select the part you want to explode.
4. Click the middle mouse button to explode the part.

Warp a part

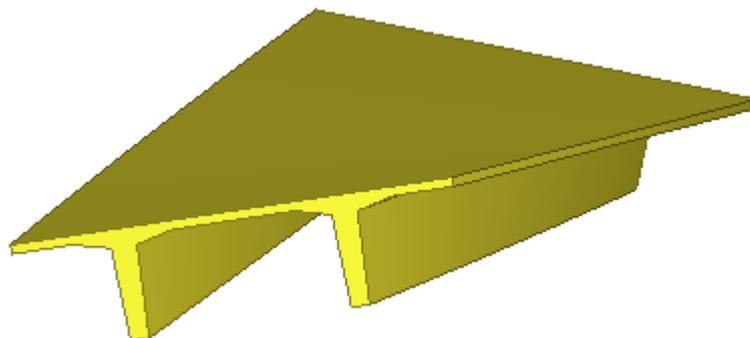
You can warp steel and concrete beams and columns, and concrete slabs. The warping functionality is available only in the **Full, Precast Concrete Detailing**, and **Steel Detailing** configurations.

Warp a beam or a column using deformation angles

1. Double-click a beam or a column to open the properties.
2. Go to the **Deforming** section.
3. In the **Warping Start** box, enter the angle of the beam at its start point, relative to the part handles.
4. In the **Warping End** box, enter the angle of the beam at its end point, relative to the part handles.

For example, to warp the beam 10 degrees at the end point, enter 0 in the **Start** angle box and 10 in the **End** angle box.

5. Click **Modify** to warp the beam.



Warp a concrete slab by moving chamfers

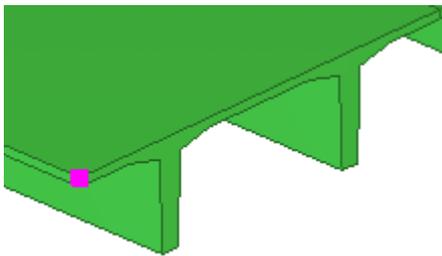
Before you start, create a concrete slab by using the **Slab** command on the **Concrete** tab.

1. Double-click a chamfer to open the **Corner chamfer** properties.
2. Modify the chamfer properties.
Do not modify the chamfers so that the slab faces are no longer planar.
 - To move the upper corner of the chamfer, modify the **Dz1** value.
 - To move the lower corner of the chamfer, modify the **Dz2** value.
3. Click **Modify** to warp the slab.

Warp a Floor Bay (66) slab

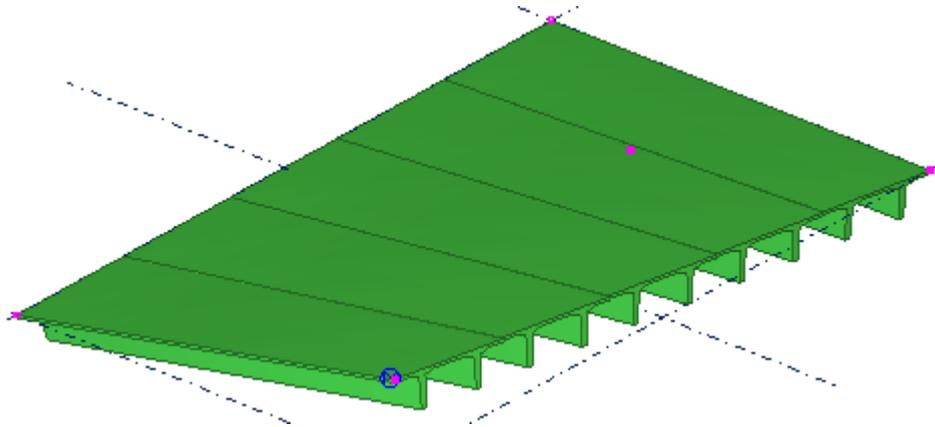
Before you start, create a concrete slab by using the Modeling of floor bay (66) component.

1. Ensure that the **Select components** selection switch is on.
2. Select the chamfer you want to move.
For example, select the corner point of a slab component to warp that end of the slab:



3. Right-click and select **Move special --> Linear...**
4. In the **Move - Linear** dialog box, enter a value in the appropriate direction box.
For example, enter 100 in the **dZ** box to lift that corner up 100 mm.
5. Click **Move**.

Tekla Structures moves the point in the direction you selected, which warps the slabs.



6. Right-click and select **Interrupt**.
7. Ensure that the **Select objects in components** selection switch is on.
8. To see the warping angle of a single slab, double-click a slab to open the **Concrete beam** properties, and go to the **Deforming** section.

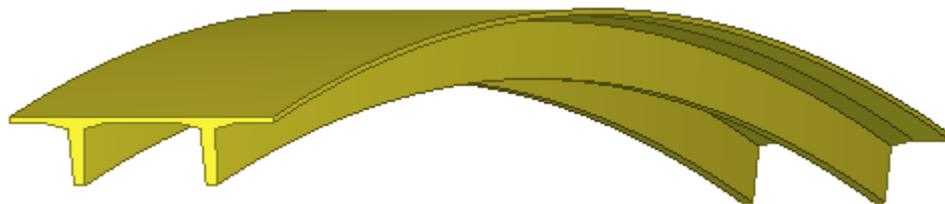
The **Warping** start and end values show the warping angle at the start point and at the end point of the part.

Camber a part

You can use cambering to pre-camber parts, in other words, to curve long heavy sections that will settle on site and become flat. Use cambering to show the natural camber of a prestressed part in a model. Cambering affects the position of cuts, skews, and embeds in the model.

1. Double-click the part to open the part properties.
2. Go to the **Deforming** section.
3. In the **Cambering** box, define the degree of camber.
4. Click **Modify**.

Tekla Structures cambers the part in the local z direction.



2.4 Add details to parts

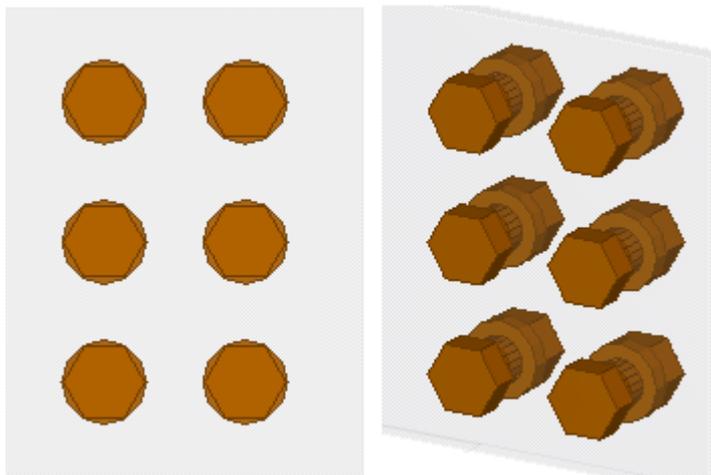
This section explains how to create details using Tekla Structures. It also gives you some techniques for fine-tuning the part shape.

You can create details for example the following ways:

- create bolts, [studs \(page 342\)](#) and holes
- create [welds \(page 346\)](#) and define how they [appear in the model \(page 358\)](#)
- create [fittings \(page 361\)](#), cuts and part chamfers to refine the shape of a part
- [add surface treatment to parts \(page 373\)](#) and [surfaces to part faces \(page 386\)](#)

Create bolts

To create bolts, you can either create a single bolt group or apply a component that automatically creates bolt groups.



Tekla Structures uses the same command for creating bolts, [studs \(page 342\)](#) and holes. If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

You can create different marks for bolts and holes in drawings.

Create a bolt group

1. On the **Steel** tab, click **Bolt** .
The **Bolt** properties open.

2. Modify the **Bolt** properties, if needed.
For example, the **Bolt group** settings affect the end result.
3. Select the main part, to which the secondary parts will be bolted.
4. Select the secondary parts.
5. Click the middle mouse button to finish selecting parts.
6. Pick a point to indicate the bolt group origin.
7. Pick a second point to indicate the direction of the bolt group x axis.

NOTE Tekla Structures determines the location of the bolt group using the following values: the bolt group x axis and the work plane. Dimensions are relative to the bolt group origin, which is the first point picked. Tekla Structures sets the x direction of the bolt group using the second point picked. It is important that the points you pick to create the bolt group are close enough to the parts you want to connect.

Create a single bolt

1. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
2. Under **Bolt group**, select **Array** from the **Shape** list.
3. In the **Bolt dist X** and **Bolt dist Y** boxes, enter 0.
4. Create the bolt the same way you would create a bolt group:
 - a. Select the main part, to which the secondary parts will be bolted.
 - b. Select the secondary parts.
 - c. Click the middle mouse button to finish selecting parts.
 - d. Pick a point to indicate the bolt origin.
 - e. Pick a second point to indicate the direction of the x axis.

Create bolts using the Auto bolt component

Use the **Auto bolt** component to bolt parts and nearby parts, shim plates, splice plates, or other plates. **Auto bolt** follows the part rotation and finds the best rotation so that you do not need to set the work plane. With **Auto bolt** one bolt group can span many parts, for example, manage a splice as a single group

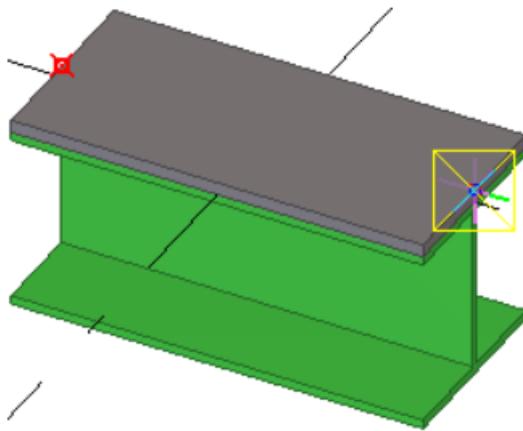
1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

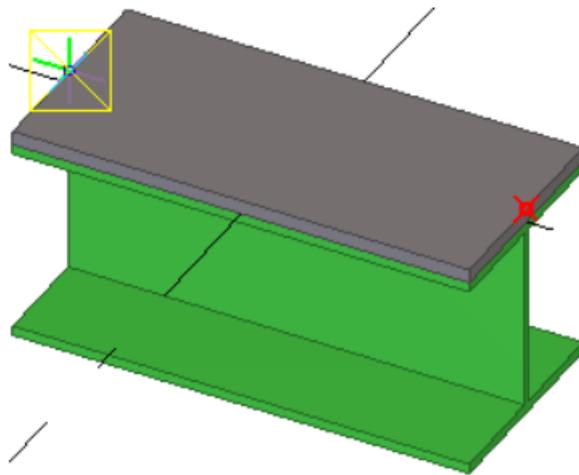
2. Start typing `auto bolt` in the search box.
3. Double-click **Auto bolt** in the catalog to open the **Auto bolt** dialog box.
4. Define the bolt properties.
5. If needed, you can show cut length as temporary lines to view where the bolts should be placed even if they are not created.
 - Select  in the list at the bottom of the dialog box to not show the temporary lines.
 - Select  in the list at the bottom of the dialog box to show the temporary lines.

To delete the temporary lines, right-click the view and select **Redraw View**.

6. Click **Apply**.
7. Select the main part.

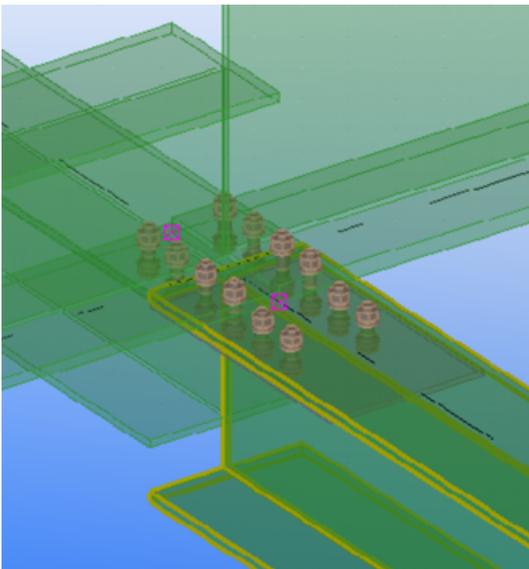
Auto bolt uses this part to identify the best rotation. This part will be the main part of the assembly.
8. Select the secondary part.
9. Click the middle mouse button.
10. Pick the first and the second position to define the bolt group direction.

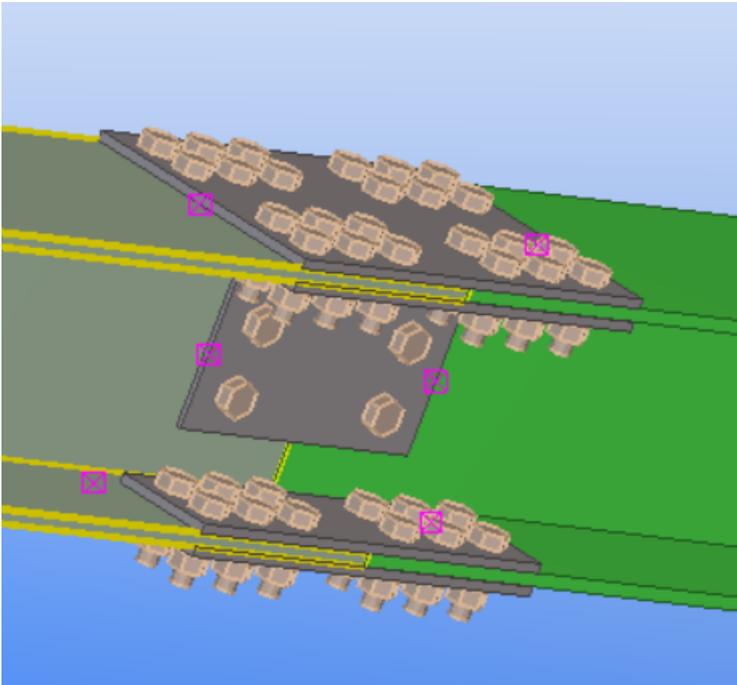
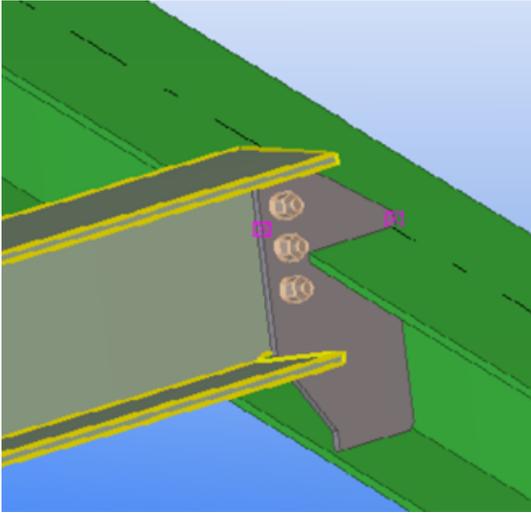




Examples

Examples of parts bolted using the **Auto bolt** component are shown below. The main parts and the selected points are highlighted.





Create a bolt group by exploding a component

An alternative way to create bolts is to first apply a component that includes bolt groups, and then explode the component.

1. Apply a component that includes bolt groups.
For example, connect two beams, or a beam to a column, using a bolted end plate.
2. [Explode \(page 750\)](#) the component.
 - a. Select the component to explode.

- b. Right-click and select **Explode Component**.
Tekla Structures separates the objects in the component.
3. Modify the bolt group.
 - a. Select the bolt group and double-click it to open the properties.
 - b. Modify the properties.
 - c. Click **Modify**.

Change or add bolted parts

You can change the parts a bolt group connects to.

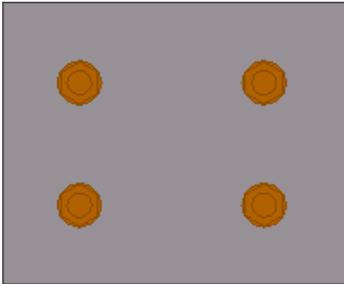
1. On the **Steel** tab, click **Bolted parts**.
2. Select the bolt group.
3. Reselect the main and secondary parts.
4. Click the middle mouse button to finish selecting the parts.

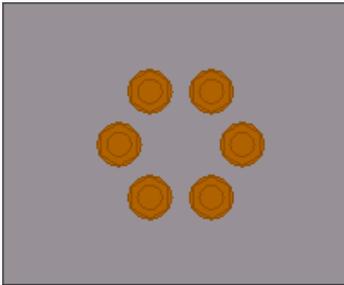
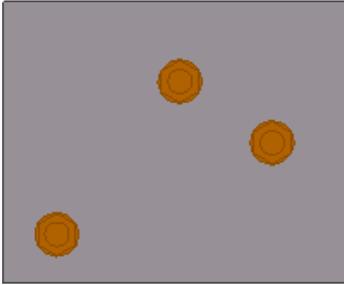
Bolt group shape

Tekla Structures uses the values of the **Bolt dist X** and **Bolt dist Y** boxes in the **Bolt** properties to determine how many bolts the bolt group contains, as shown in the table below:

Shape	Bolt dist X	Bolt dist Y
Array	Spacing between bolts, in the x direction of the bolt group.	Spacing between bolts, in the y direction of the bolt group.
Circle	Number of bolts.	Diameter of the bolt group.
List	x coordinate of each bolt, from the bolt group point of origin.	y coordinate of each bolt, from the bolt group point of origin.

Examples

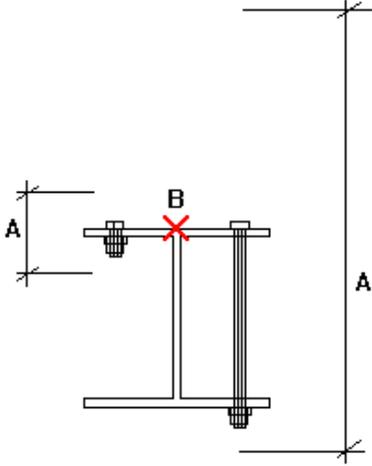
Bolt group shape	Dimensions	Result
Array	Bolt dist X: 150 Bolt dist Y: 100	

Bolt group shape	Dimensions	Result
Circle	Number of bolts: 6 Diameter: 100	
List	Bolt dist X: 75 175 250 Bolt dist Y: 75 -50 0	

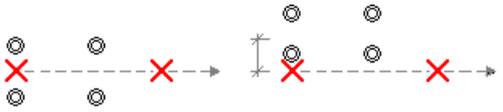
Bolt properties

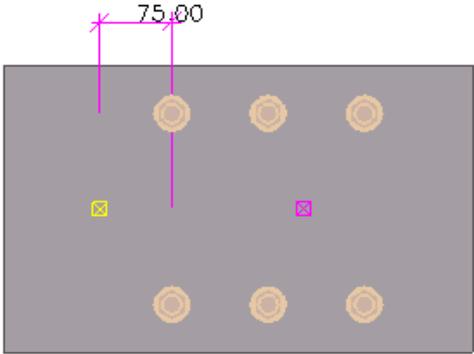
Use the **Bolt** properties to view or modify the properties of a bolt group. The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

Setting	Description
Bolt	
Size	Bolt diameter.
Standard	Bolt assembly standard/grade.
Bolt type	Define whether the bolts are assembled on-site or in the shop.
Connect as	Indicate whether you are bolting a secondary part or a sub-assembly.
Thread in material	Indicate if the thread of the bolt can be inside the bolted parts. Tekla Structures does not use this value when calculating the length of full-threaded bolts.
Cut length	Indicate which parts the bolt connects. The value defines the area Tekla Structures should search for parts that belong to the bolt group. Using cut length you can determine

Setting	Description
	<p>whether the bolt will go through one flange or two.</p> <p>Tekla Structures searches for parts using half the cut length value, in both directions from the bolt group plane. In the illustration below, A is the cut length and B is the bolt origin. Tekla Structures calculates the search area as $A/2$ in both directions from point B.</p>  <p>Tekla Structures warns you if the cut length is too small (i.e. the bolt group contains no parts) and makes the bolt length 100 mm.</p> <p>If there are large gaps between the connected parts, the gap is added to the length of the bolt. Tekla Structures calculates bolt length using the total distance between the first and last surfaces.</p> <p>NOTE: If you want to force a bolt to be a certain length, enter a negative value for cut length (e.g. -150).</p>
Extra length	<p>Additional bolt length.</p> <p>Increases the material thickness that Tekla Structures uses when calculating bolt length. For example, you might need extra bolt length to allow for painting. You can also build</p>

Setting	Description
	additional lengths into bolt assemblies.
Assembly	Select whether to create washers and nuts with the bolt. If you want to create only holes without any bolts, clear all the check boxes.
Bolt group	
Shape	Bolt group shape. You have the following options: <ul style="list-style-type: none"> • Array for rectangular • Circle for circular • xy list for any shape
Bolt dist X	Bolt spacing, number of bolts, or coordinate, depending on the bolt group shape.
Bolt dist Y	Bolt spacing, group diameter, or coordinate, depending on the bolt group shape.
Number of bolts	Number of bolts in a circular bolt group.
Diameter	Diameter of bolts in a circular bolt group.
Holes	
Tolerance	Tolerance = Hole diameter - Bolt diameter
Parts with slotted holes	If you want to create oversized or slotted holes, select the desired check boxes to indicate which plies of the connection get special holes.
Special hole type	Oversized, slotted, or no holes. This option becomes active when you select the Special hole check boxes next to Parts with slotted holes .
Slotted hole X	x allowance of a slotted hole. Zero for a round hole.
Slotted hole Y	y allowance of a slotted hole. Zero for a round hole.
Rotate slots	If the bolt connects several parts, you may want to rotate alternate holes by

Setting	Description
	90 degrees. This allows the bolt to move in different directions.
Oversized	Allowance of an oversized hole.
Position	
On plane	<p>Move the bolt group perpendicular to the bolt group x axis.</p> 
Rotation	<p>Define how far the bolt group is rotated around the x axis, relative to the current work plane.</p> <p>For example, you can use this box to indicate on which side of the connected parts you want the bolt head to be.</p> 
At depth	Move the bolt group perpendicular to the current work plane.
Offset from	
Dx, Dy, Dz	<p>Offsets that move the bolt group by moving the bolt group x axis. Use to change the position of the bolt group.</p> <p>The start point values Dx, Dy and Dz move the first end of the bolt group, relative to the bolt group x axis. The end point values move the second end of the bolt group.</p> <ul style="list-style-type: none"> • A positive Dx value moves the start point towards the end point. • Dy moves the end point perpendicular to the bolt group x axis on the current work plane. • Dz moves the end point perpendicular to the current work plane. <p>An example bolt group with the Dx start point set to 75:</p>

Setting	Description
	
Custom properties	
More	Click the More button to open the user-defined attributes (UDAs) of the bolt. UDAs provide more information about the bolts.
Show cut length as temporary lines	<p>This option is available in the Auto bolt component.</p> <p>Shows where the bolts should be placed even if they are not created.</p> <ul style="list-style-type: none"> • Select  to not show the temporary lines. • Select  to show the temporary lines.

Create studs

A stud is special type of bolt that is welded to steel parts to transfer loads between steel and concrete.

Tekla Structures uses the same command for creating [bolts \(page 332\)](#), studs and holes. When you create studs, select the stud assembly standard in the **Bolt** properties. You can create a group of studs or a single stud.

You can also create studs by using the **Shear Stud (1010)** component.

1. Ensure that the needed studs are added to the bolt catalog and bolt assembly catalog.
2. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
3. In the **Standard** list, select the bolt assembly standard for the studs.
4. Under **Bolt group**, do one of the following:

- To create a group of studs, define **Shape** and the related properties as desired.
 - To create a single stud, select **Array** from the **Shape** list and type 0 in the **Bolt dist X** and **Bolt dist Y** boxes.
5. Modify the other properties as desired.
 6. Select the main part.
 7. Click the middle mouse button to finish selecting parts.
 8. Pick a point to indicate the origin of the stud or stud group.
 9. Pick a second point to indicate the direction of the stud group x axis.

Create holes

Tekla Structures uses the same command for creating bolts, studs and holes. Before creating holes, you need to change some of the properties in the **Bolt** properties. If you want to create only holes, do not use any bolt elements (such as bolts, washers, and nuts).

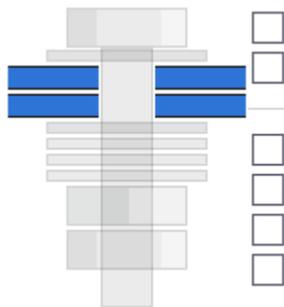
You can create the following types of holes:

- Round
- Oversized
- Slotted
- Tapped

Create round holes

You can create a group of round holes, or a single round hole. Tekla Structures calculates the diameter of a round hole as the sum of **Size** and **Tolerance**.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. If you do not want to create any bolts, clear all the **Assembly** check boxes.



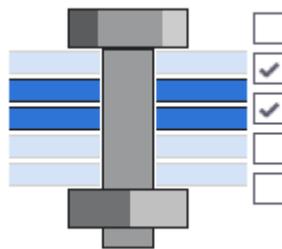
3. If needed, modify the hole properties.
4. Create the holes the same way you would create a [bolt group \(page 332\)](#):
 - a. Select the main part, to which the secondary parts will be bolted.
 - b. Select the secondary parts.
 - c. Click the middle mouse button to finish selecting parts.
 - d. Pick a point to indicate the hole group origin.
 - e. Pick a second point to indicate the direction of the hole group x axis.

Create oversized holes

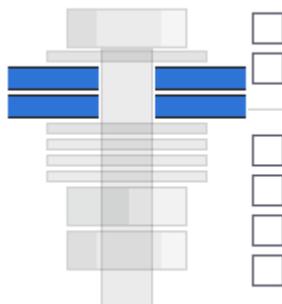
You can create a group of oversized holes.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. Next to **Parts with slotted holes**, indicate which plies of the connection get oversized holes by selecting the desired **Special hole** check boxes.

For example:



3. If you do not want to create any bolts, clear all the **Assembly** check boxes.



4. In the **Special hole type** list, select **Oversized**.
5. In the **Oversized** box, enter the allowance for the oversized hole.
You can also use a negative value to create smaller (tapped) holes.
6. Create the holes the same way you would create a [bolt group \(page 332\)](#):
 - a. Select the main part, to which the secondary parts will be bolted.

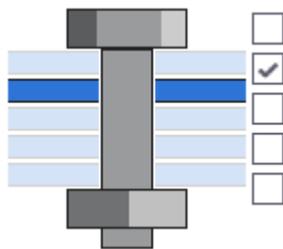
- b. Select the secondary parts.
- c. Click the middle mouse button to finish selecting parts.
- d. Pick a point to indicate the hole group origin.
- e. Pick a second point to indicate the direction of the hole group x axis.

Create slotted holes

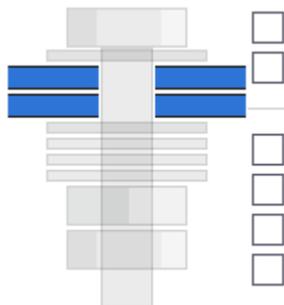
You can create a group of slotted holes.

1. On the **Steel** tab, hold down **Shift** and click  **Bolt** to open the **Bolt** properties.
2. Next to **Parts with slotted holes**, indicate which parts should be slotted by selecting the desired **Special hole** check boxes.

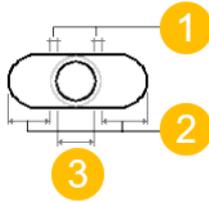
Tekla Structures counts the pieces of steel from the head of the bolt down. For example, if you select the second check box from the head of the bolt, Tekla Structures slots the second piece of steel from the head of the bolt.



3. If you do not want to create any bolts, clear all the **Assembly** check boxes.



4. In the **Special hole type** list, select **Slotted**.
5. Enter the allowance for the slotted hole in the x and y directions of the hole group using the **Slotted hole X** or **Slotted hole Y** boxes.



- (1) Tolerance
- (2) Slotted hole X or Y
- (3) Bolt size

6. If you want to rotate alternate holes by 90 degrees, select **Even** or **Odd** in the **Rotate Slots** list.



- (1) Crossing slotted holes to odd or even parts
- (2) Parallel slotted holes

7. Create the holes the same way you would create a [bolt group \(page 332\)](#):

- a. Select the main part, to which the secondary parts will be bolted.
- b. Select the secondary parts.
- c. Click the middle mouse button to finish selecting parts.
- d. Pick a point to indicate the hole group origin.
- e. Pick a second point to indicate the direction of the hole group x axis.

Create welds

You can either create a weld manually, or use a component that automatically creates welds.

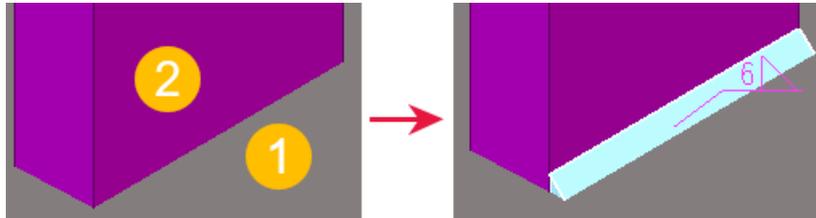
By default, Tekla Structures places the welds to the arrow side using the **Above line** properties according to the ISO standard. You can change this to **Below line** to comply with the AISC standard with the advanced option XS_AISC_WELD_MARK.

Create a weld between parts

Weld two parts together using the weld position defined in the **Weld** properties. The length of the weld depends on the length of the connection between the welded parts.

1. On the **Steel** tab, click **Weld** --> **Create weld between parts** .

2. Select the part to weld to.
If you are creating a workshop weld, this is the main part of the assembly.
3. Select the part to be welded.
If you are creating a workshop weld, this is the secondary part of the assembly.



(1) Main part

(2) Secondary part

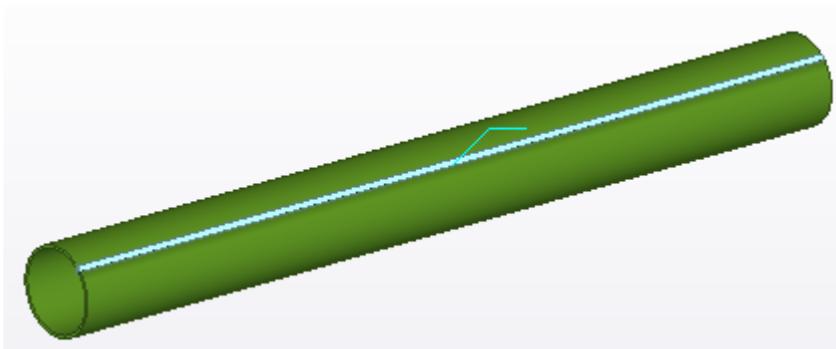
Create a weld to a part

You can create a weld to a single part, without connecting any other parts.

1. On the **Steel** tab, click **Weld** --> **Create weld to part**.
2. Select the part that you want to weld.
3. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.
4. Click the middle mouse button to create the weld.

Example

Use the **Create weld to part** command to weld seams in tubular sections:



TIP To model tubular sections with visible seams, use the SPD profile.

Create a polygon weld

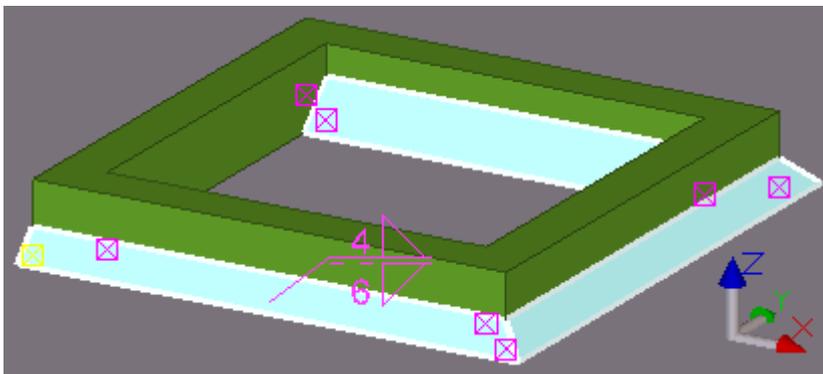
Create polygon welds when you want to define the exact position of the weld by picking the points you want the weld to traverse.

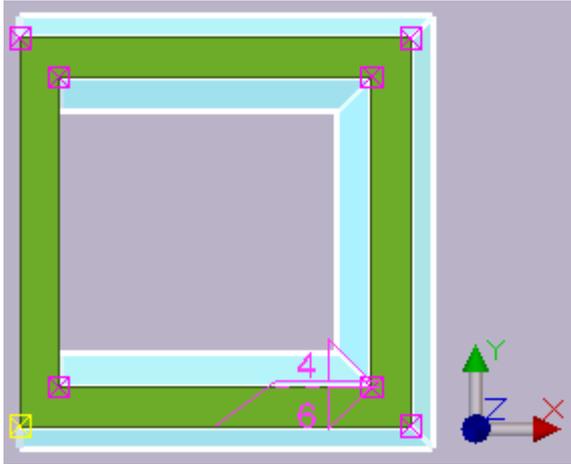
If you want to create double-sided polygon welds, define both the **Above line** and **Below line** properties.

1. On the **Steel** tab, click **Weld** --> **Create polygon weld** .
2. Select the part to weld to.
If you are creating a workshop weld, this is the main part of the assembly.
3. Select the part to be welded.
If you are creating a workshop weld, this is the secondary part of the assembly.
4. Pick the starting and end point, or alternatively, pick the points you want the weld to go through.
To create a double-sided polygon weld, pick the polygon points on one side of the part to be welded. Tekla Structures automatically searches for the corresponding points on the other side of the part.
5. Click the middle mouse button to create the weld.
6. If needed, modify the weld by dragging the handles.

Example

This example shows a double-sided polygon weld along three (outer and inner) edges of a rectangular hollow profile:



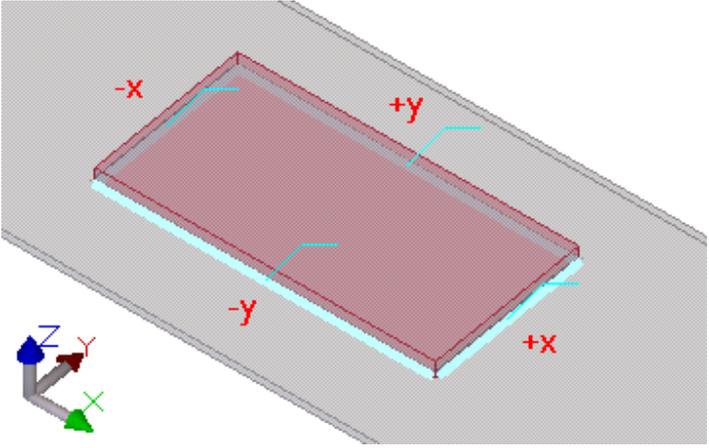


Weld properties

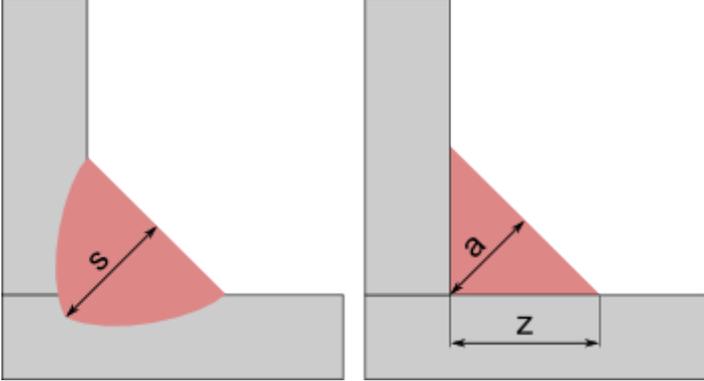
Use the **Weld** properties to view or modify the properties of a weld. The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

NOTE Some of the properties are only displayed in reports, not in drawings.

Setting	Description
Common attributes	
Edge/Around	<p>Indicates whether only one edge or the entire perimeter of a face should be welded.</p> <p>Edge: </p> <p>Around: </p>
Workshop/Site	<p>Indicates where the weld should be made. This setting affects assemblies and drawings.</p> <p>Workshop: </p> <p>Site: </p>
Position	<p>Not available for polygon welds.</p> <p>Defines the position of a weld relative to the work plane. The type and position of the parts to be welded affect the position of the weld.</p> <p>The options for weld position are:</p> <ul style="list-style-type: none"> • + x • - x

Setting	Description
	<ul style="list-style-type: none"> • + y • - y • + z • - z <p>In most cases, Tekla Structures creates the weld on the face or side of the part that faces in the selected direction (x, y, or z). Also the following factors may affect the position of the weld:</p> <ul style="list-style-type: none"> • perpendicularity of the part edge towards the selected direction (x, y or z) • length of the part edge • distance of the part edge in the selected direction (x, y or z) <p>The following image shows welds in different positions:</p> 
Shape	<p>The shape of the weld can be:</p> <ul style="list-style-type: none"> •  (Regular, continuous weld) •  (Intermittent weld) •  (Staggered, intermittent weld)
Connect as	See Use welds to create assemblies (page 389) .
Placement	<p>Defines how the weld is placed in relation to the assembly parts.</p> <p>The options are:</p>

Setting	Description
	<ul style="list-style-type: none"> <li data-bbox="671 271 1364 595"> <p>• Auto</p> <p>The weld placement adapts to the typical situation of the weld type.</p> <p>Square-, V-, and U-groove welds are located in the middle of the main and secondary parts. Single-bevel and J-groove welds are located on the secondary part side.</p> <p>This is the default option.</p> <li data-bbox="671 607 1364 752"> <p>• Main part</p> <p>The weld is located fully on the main part side.</p> <p>Does not affect V- or U-groove welds.</p> <li data-bbox="671 763 1364 938"> <p>• Secondary part</p> <p>The weld is located fully on the secondary part side.</p> <p>Does not affect V- or U-groove welds.</p>
Preparation	<p>Defines which assembly parts, if any, are automatically prepared for welding.</p> <p>The options are:</p> <ul style="list-style-type: none"> <li data-bbox="671 1077 1364 1223"> <p>• None</p> <p>Parts are not prepared for welding.</p> <p>This is the default option.</p> <li data-bbox="671 1234 1364 1357"> <p>• Auto</p> <p>Parts are prepared for welding according to the weld type.</p> <li data-bbox="671 1368 1364 1447"> <p>• Main part</p> <p>The main part is prepared for welding.</p> <li data-bbox="671 1458 1364 1559"> <p>• Secondary part</p> <p>The secondary part is prepared for welding.</p>
Weld	
Prefix	<p>The weld size prefix. Shown in drawings, but only if the weld size is also defined.</p> <p>The standard ISO 2553 prefixes are:</p> <ul style="list-style-type: none"> <li data-bbox="671 1738 1364 1783">• a - Design throat thickness <li data-bbox="671 1794 1364 1839">• s - Penetration throat thickness <li data-bbox="671 1850 1364 1874">• z - Leg length

Setting	Description
	 <p>Note that if the last character of the prefix is <i>s</i>, Tekla Structures creates the solid weld object according to the right-hand image so that <i>a</i> equals weld size.</p>
Type	See the list of weld types (page 354) below.
Size	<p>The size of the weld.</p> <p>If you enter a zero or negative weld size, Tekla Structures creates the weld, but does not display it in drawings.</p> <p>For compound weld types $V+\Delta$ and $II+\Delta$, you can enter two size values.</p>
Angle	<p>The angle of weld preparation, bevels, or groove.</p> <p>Enter a positive value for bevel and groove welds.</p> <p>Tekla Structures displays the angle between the weld type symbol and the fill type contour symbol.</p>
Contour	<p>The fill type contour of a weld can be:</p> <ul style="list-style-type: none"> • None • Flush  • Convex  • Concave  <p>This setting does not affect solid weld objects.</p>
Finish	<p>Tekla Structures displays the finish symbol above the weld type symbol in drawings. The options are:</p> <ul style="list-style-type: none"> • G (Grind) • M (Machine) • C (Chip) •  (Flush finished weld)

Setting	Description
	<ul style="list-style-type: none"> •  (Smooth blended weld face) <p>This setting does not affect solid weld objects.</p>
Root face	<p>Root face thickness is the height of the narrowest part inside the root opening.</p> <p>Root face values do not appear in drawings, but you can use the <code>WELD_ROOT_FACE_THICKNESS</code> template attribute in reports to show the root face dimension in the weld list.</p>
Effective throat	<p>The weld size used in weld strength calculation.</p>
Root opening	<p>The space between the welded parts.</p> <p>Enter a positive value for square-groove welds.</p>
# of incr.	<p>The amount of increments in an intermittent weld.</p> <p>Only use with the ISO standard.</p>
Length	<p>Defines the length value that is shown in the weld mark.</p> <p>For intermittent welds, defines the length of an increment.</p> <p>Does not affect continuous solid weld objects.</p>
Pitch	<p>If the advanced option <code>XS_AISC_WELD_MARK</code> is set to <code>TRUE</code>, the center-to-center spacing of weld increments in an intermittent weld.</p> <p>If the advanced option <code>XS_AISC_WELD_MARK</code> is set to <code>FALSE</code>, the space between the weld increments in an intermittent weld.</p> <p>Tekla Structures uses the – character by default to separate weld length and pitch, for example 50–100. To change the separator to @, for example, set the advanced option <code>XS_WELD_LENGTH_CC_SEPARATOR_CHAR</code> to @.</p>
	<p>Use these buttons to copy and to link together the Above line and Below line property values.</p> <p>Click the  and  buttons to copy values between the Below line and Above line columns.</p> <p>Click the  button to switch the linking on or off.</p> <p>The middle button is yellow  when the values are linked. This means that if you change a value in either of the columns, also the corresponding value in the other column changes.</p>

Setting	Description
Tail information	
NDT inspection level	Defines the non-destructive testing and inspection level.
Electrode classification	Defines the weld electrode classification.
Electrode strength	Defines the electrode strength.
Electrode coefficient	Defines the electrode strength coefficient.
Process type	Defines the process type.
Reference text	Additional information to appear in the weld mark. For example, information about the weld specification or process. Note that the special characters are shown in weld marks in model views only if the Arial font supports the special characters.
Custom properties	
More	Click the More button to open the user-defined attributes (UDAs) of the weld. UDAs provide additional information about the weld.

List of weld types

Use the weld properties to define the weld type. Some weld types also automatically prepare the parts to be welded. The following table shows the available weld types:

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
0		None	No	No
10		Fillet weld	No	Yes
3		Bevel-groove (single-V butt weld)	Yes	Yes
4		Bevel-groove (single-bevel butt weld)	Yes	Yes
2		Square-groove (square butt weld)	Yes	Yes
5		Single-V butt weld with	Yes	Yes

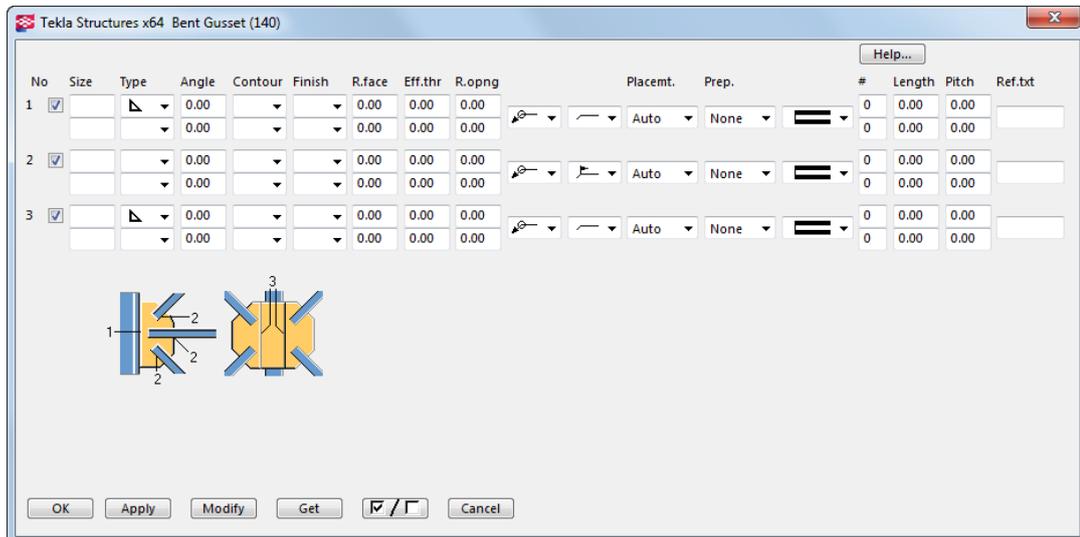
Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
		broad root face		
6		Single-bevel butt weld with broad root face	Yes	Yes
7		U-groove weld (single-U butt weld)	Yes	Yes
8		J-groove weld (single-J butt weld)	Yes	Yes
16		Flare V-groove weld	No	No
15		Flare-bevel-groove weld	No	No
1		Edge-flange weld	No	No
17		Corner-flange weld	No	No
11		Plug weld	No	No
9		Bevel backing weld	No	No
12		Spot weld	No	No
13		Seam weld	No	No
14		Slot weld	No	No
18		Partial penetration weld (single-bevel butt + fillet)	No	Yes
19		Partial penetration weld (square groove + fillet)	No	Yes
20		Melt-through weld	No	No

Number	Type	Name	Optional automatic weld preparation	Supported solid weld object
21		Steep-flanked single-V butt weld	Yes	Yes
22		Steep-flanked single-bevel butt weld	Yes	Yes
23		Edge weld	No	No
24		Surfacing weld	No	No
25		Fold joint	No	No
26		Inclined joint	No	No

Welds in components

You can define the properties of the welds used in components. Tekla Structures displays the appropriate weld dialog box when you click the **Welds** button in the component properties dialog box.

The example image shows each weld definition using a number for the **Bent gusset (140)** connection. For each weld definition, use the upper row to define the above-line properties of the weld, and the lower row for the below-line properties.



See also

[Create welds \(page 346\)](#)

Weld preparation

When parts are prepared for welding, their edges can be beveled to produce a groove for the weld. You can define the angle of bevels and grooves.

You can either prepare a part for welding manually, or apply a component that does it automatically, or you can use the **Preparation** options in the **Weld** properties or in the component weld properties.

NOTE When you use components' weld preparation options, the [supported weld types \(page 354\)](#) are placed correctly in the model. If you use cuts for preparing the part edges, the welds may not be placed correctly.

Prepare a part for welding with a polygon

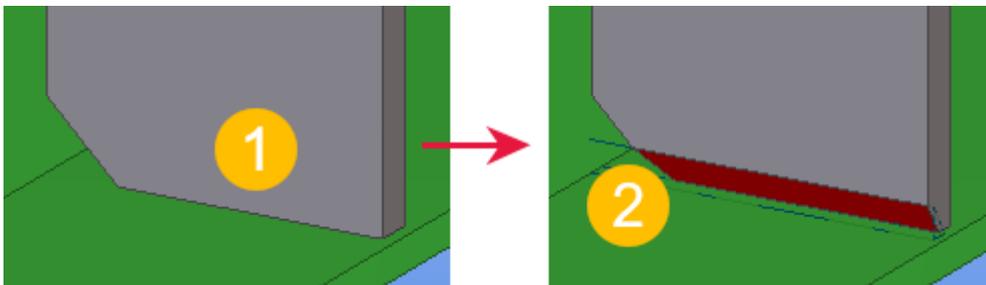
You can manually prepare a part for welding by cutting it with a polygonal shape.

Before you start, ensure that the [work plane \(page 51\)](#) is on the plane you are cutting on.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with polygon** .
2. Select the part that you want to cut.
3. Pick positions to outline the polygon to be used for cutting.

Extend the polygon outside the part, so that it is clear that the part edge should be cut away.

4. Click the middle mouse button to close the polygon and to cut the part.



(1) The part to be cut

(2) Cuts are displayed using dash-and-dot lines

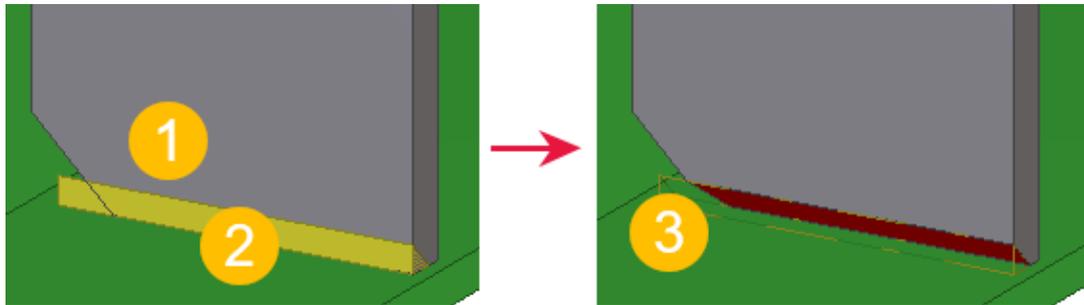
Prepare a part for welding with another part

You can manually prepare a part for welding by cutting it with another part. The cutting part will subsequently be deleted.

Before you start, create a cutting part and position it through the part you want to cut.

1. On the **Steel** tab, click **Weld** --> **Prepare part for welding with another part** .

2. Select the part that you want to cut.
3. Select the cutting part.



- (1) The part to be cut
 (2) The cutting part
 (3) Cuts are displayed using dash-and-dot lines

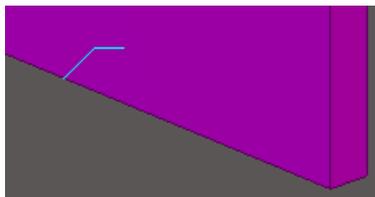
Set the visibility and appearance of welds

Modify the display settings to define how welds appear in the model.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Ensure that the **Welds** check box is selected.
4. Select a representation option for welds:

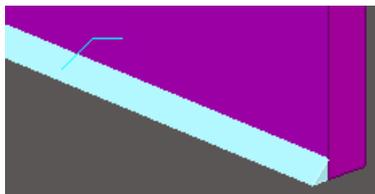
- **Fast**

Use this option to show the weld symbols only.



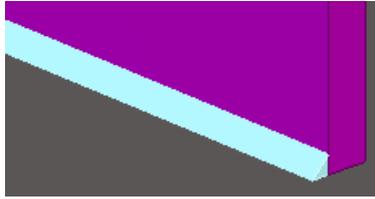
- **Exact**

Use this option to show welds as solid objects with weld symbols, and to display weld marks when you select welds.



- **Exact - no weld mark**

Use this option to show welds as solid objects without weld symbols. Weld marks will not be displayed when you select welds.



5. Ensure that the view is being selected.
6. Click **Modify** to apply the changes.

NOTE If the representation option is **Exact** and you still cannot see the weld object in the model, check that the following properties have been defined for the weld in question:

- **Size**
 - **Type**
 - **Angle**
 - **Root opening**
-

See also

[Create welds \(page 346\)](#)

[Display settings \(page 918\)](#)

Change a weld to a polygon weld

You can change existing welds to polygon welds if the existing welds have been created by using the **Create weld between parts** command or by a component. The new polygon welds will traverse the same points as the original welds did.

When converting double-sided weld segments to a polygon weld, Tekla Structures may not be able to create the polygon weld. If the welds to be converted consist of more than one polygon, or if there is a different number of weld segments on the sides of the welded part, Tekla Structures does not create the double-sided polygon weld but separate single-sided polygon welds.

1. Select the weld you want to change.
To select multiple welds, hold down the **Ctrl** or **Shift** key.
2. On the **Steel** tab, click **Weld** --> **Convert to polygon weld** .

See also

[Create welds \(page 346\)](#)

Split a polygon weld

You can split a double-sided polygon weld into two single-sided polygon welds.

1. Select the double-sided polygon weld that you want to split.
2. Right-click and select **Split**.

See also

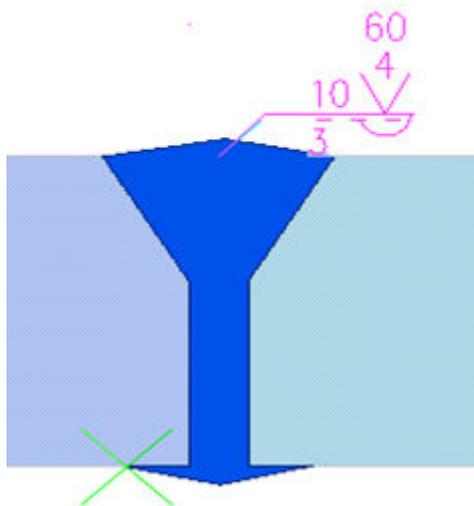
[Create welds \(page 346\)](#)

[Change a weld to a polygon weld \(page 359\)](#)

Create user-defined cross sections for welds

You can define special cross sections for model welds. This is useful when you need weld cross sections that are not predefined in Tekla Structures.

For example, you can create bevel backing welds:



To find the welds in the model that have a user-defined cross section, set **Category** to **Weld** and **Property** to **User-defined cross section** in the selection or view filter, or in the color and transparency settings.

Define a user-defined cross section for a weld

1. Select the weld you want to modify.
2. Right-click and select **Define Cross Section**.
3. In the weld cross section editor view:
 - a. Pick points to indicate the weld cross section corners.

- b. Click the middle mouse button to finish picking.

Remove a user-defined cross section from a weld

You can remove user-defined cross sections from model welds and revert to the previous standard cross sections.

1. Select a weld that has a user-defined cross section.
2. Right-click and select **Remove Cross Section**.

Tekla Structures removes the user-defined cross section and uses the previous standard cross section and properties for the weld.

Limitations

- User-defined weld cross sections are reported using only the above-line properties.
- User-defined weld cross sections do not create automatic weld preparation.

Create fittings

You can fit the end of a part by creating a straight cutting line between two points you pick. Use fittings to shorten beams. Do not use fittings for making considerable beam extensions.

When you create fittings, Tekla Structures fits the part end to the cutting line and automatically deletes the shortest side of the part. Use [plane view \(page 29\)](#) when you create fittings.

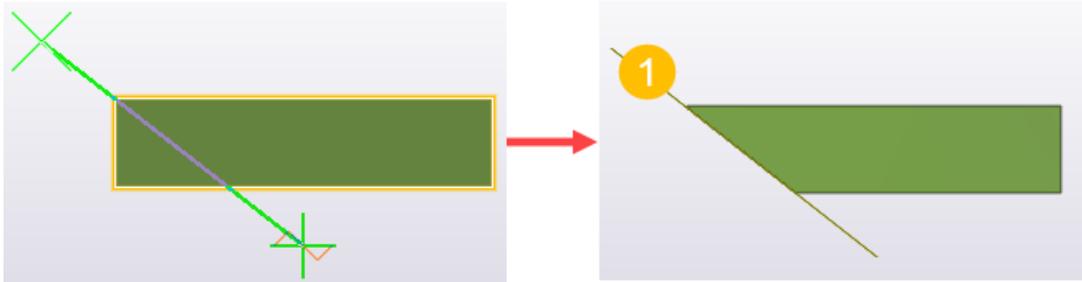
TIP Ensure that the **Snap to nearest points (point on line)** snap switch is active when you create fittings.

Limitations:

- Fittings cannot be used on contour plates.
- If you apply a second fitting on the same part end, Tekla Structures will ignore the first fitting. This happens if you use the **Fit part end** command for cutting and try to make two cuts on the same part end. In situations like this, use, for example, the **Line cut** command instead.

1. On the **Edit** tab, click **Fit part end**.
2. Select the part you want to cut with a fitting.
3. Pick the first point of the cutting line.
4. Pick the second point of the cutting line.

Tekla Structures creates a fitting between the two points you picked. The fitting adjusts the end of a beam on a plane, perpendicular to the view plane.



(1) Fitting symbol

See also

[Show part handles and part reference lines in a model view \(page 303\)](#)

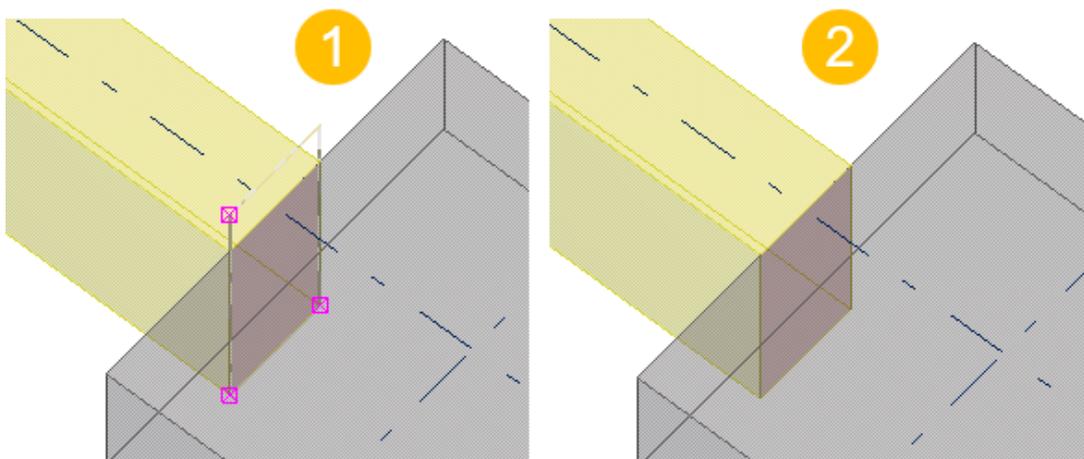
Create cuts

You can use cuts to shape a part. Do not use cuts to otherwise change the length of a part in the model.

Cut parts with a line

Use line cuts to shape the end of a beam or column. A line cut cuts the end of a beam on a plane that passes through the points you pick. Tekla Structures displays the cut line using dash-and-dot lines.

1. On the **Edit** tab, click **Line cut**.
2. Select the part you want to cut.
3. Pick the first point of the cutting line.
4. Pick the second point of the cutting line.
5. Pick the side you want to remove.
6. If you want to modify the cut, use [direct modification \(page 103\)](#).



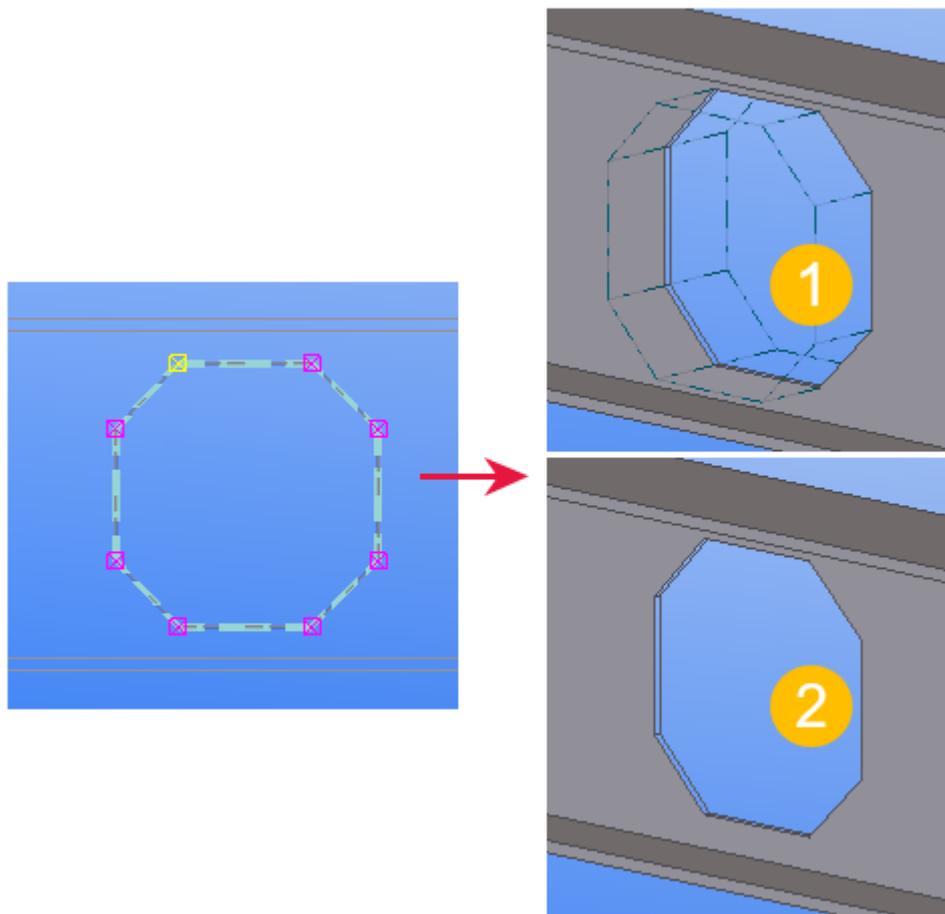
(1) Cuts are displayed using dash-and-dot lines

(2) Cut lines can be hidden

Cut parts with a polygon

A polygon cut cuts a part using a polygonal shape. Tekla Structures displays the cut using dash-and-dot lines.

1. Press **Ctrl+P** to switch to the [plane view \(page 29\)](#).
2. Ensure that the [work plane \(page 51\)](#) is on the plane you are cutting on.
For example, if you are creating a polygonal cut on the yz plane, you should temporarily set your work plane to the yz plane as well.
3. On the **Edit** tab, click **Polygon cut**.
4. Select the part you want to cut.
5. Pick positions to outline the polygon to be used for cutting.
Define the polygon so that there is some tolerance between the edges. If the edge of a cutting polygon is in exactly the same position as the edge of the part to be cut, it can be unclear whether the edge should be cut away.
6. Click the middle mouse button to close the polygon and to cut the part.
7. To modify the cut shape, use [direct modification \(page 103\)](#).
8. If needed, you can modify the cut properties.
 - a. Double-click the cut to open the **Polygon cut** properties.
 - b. [Change \(page 96\)](#) the properties as needed.
 - c. Click **Modify** to apply the changes.



(1) Polygon-shaped cut

(2) Cut lines can be hidden

NOTE Tekla Structures uses the parametric profile BL to create polygonal cuts.

If you are unable to create polygonal cuts, ensure that the BL profile is defined in the `profitab.inp` file in `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\profil` as follows:

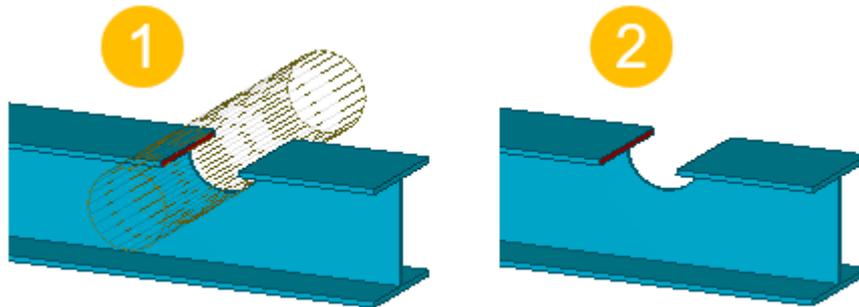
```
BL ! PL ! -1 ! ! 1 ! 2 ! ! !
```

Cut parts with another part

You can cut a part using another part. Tekla Structures displays the cut using dash-and-dot lines. Note that you can cut parts that already have cuts. That can be useful, for example, when you want to create more sophisticated cut shapes.

1. Create a cutting part and position it through the part you want to cut.
2. On the **Edit** tab, click **Part cut**.

3. Select the part you want to cut.
4. Select the cutting part.
Tekla Structures cuts the selected main part. The part cut does not affect other parts.
5. Delete the cutting part.
 - a. Ensure that the  **Select cuts and added materials selection switch** (page 122) is off.
 - b. Select the cutting part and press **Delete**.
6. If needed, you can modify the cut properties.
 - a. Double-click the cut to open the part cut properties.
 - b. [Change](#) (page 96) the properties as needed.
 - c. Click **Modify** to apply the changes.



(1) Cuts are displayed using dash-and-dot lines

(2) Cut lines can be hidden

NOTE Do not create cuts with the same planes or vertices. Otherwise, it may be unclear what should be cut away

Hide cut lines in a model view

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Ensure that the **Cuts and added material** option is **not** selected in the display settings.
4. Click **Modify**.

Tips on how to cut efficiently

- **Avoid part faces**

Avoid creating cuts that are exactly on the part planes or go through vertices. Try to position the cut at least 0.3 mm outside of the part planes.

- **Use polygon cuts**

Whenever possible, use polygon cuts. The **Polygon cut** command automatically extends the cut slightly outside of the part face. Note that after creating the polygon, you may have to adjust the position of the handles manually.

- **Use edge chamfers**

Whenever possible, use [edge chamfers \(page 369\)](#) instead of small cuts, especially in components.

- **Tips for flange cuts**

When cutting a flange, if the cutting part cuts very slightly the web as well (at least 0.3 mm), the cut is more likely to succeed. For example, if you are cutting a beam that has roundings, it may be useful to cut even further onto the web than just the flange thickness.

- **Tips for round tube cuts**

Use the Round tube (23) component for round tube cuts. The component automatically rotates the cutting part until a successful cut position is found. If the component fails, rotate the cutting part slightly until you find a successful cut position.

NOTE If a cut fails, Tekla Structures displays the cutting part using dash-and-dot lines. An error notification is printed in the session history log stating which part and which cut caused the failure.

To locate the failure in the model, click a row that contains an ID number in the session history log. Tekla Structures selects the corresponding part and cut in the model.

Polygon cut properties

Use the **Polygon cut** properties in the property pane to view and modify the properties of a polygon cut.

Note that the polygon cut properties are available in the property pane only after a polygon cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	

Setting	Description
Name	Name of the polygon cut.
Profile	Profile of the polygon cut, by default parametric profile BL .
Material	Material of the polygon cut, by default ANTIMATERIAL . The cut material cannot be changed.
Class	Use to group polygon cuts. For example, you can display cuts of different classes in different colors.
Position	
At depth	Position depth of the polygon cut.
Custom properties	
More	Click the More button to open the user-defined attributes (UDAs) of the cut. UDAs provide more information about the cuts.

Part cut properties

Part cut uses the properties of the cutting part. For example, if the cutting part is steel beam, part cut uses the **Steel beam cut** properties. The default part cut properties depend on the used cutting part.

Note that the part cut properties are available in the property pane only after a part cut has been created and selected. You cannot access or modify the cut properties before the cut is created.

The following part cut properties are available:

- Steel beam cut
- Steel column cut
- Steel spiral beam cut
- Steel item cut
- Concrete beam cut
- Concrete column cut
- Concrete panel cut
- Concrete slab cut
- Concrete spiral beam cut
- Bent plate cut
- Pad footing cut

- Strip footing cut
- Concrete item cut

If needed, you can [customize \(page 213\)](#) the property pane layouts of part cuts.

Create part chamfers

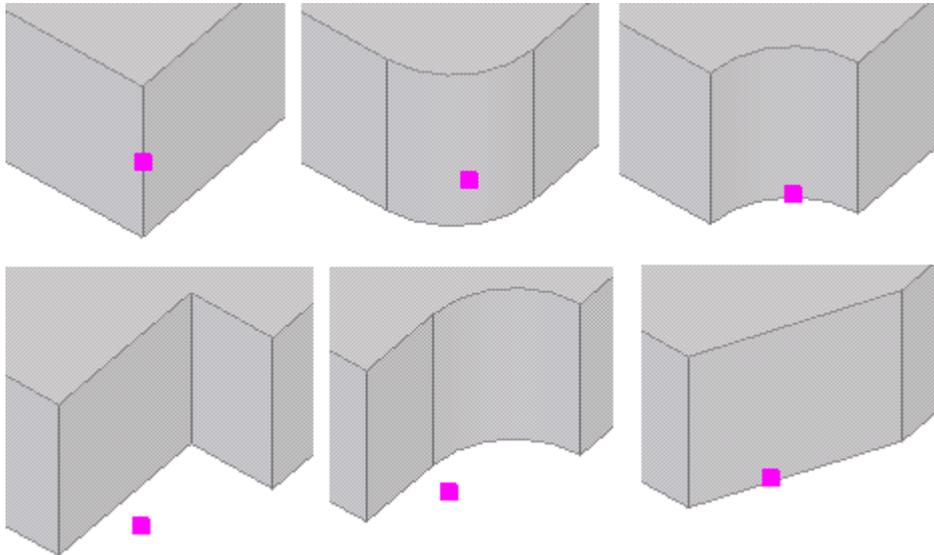
Chamfers are modeling details that can be used to refine the shape of parts for aesthetic, practical, and manufacturing reasons. In Tekla Structures, you can chamfer part corners and part edges.

Limitations:

- Only the following parts have corner chamfers: contour plates, concrete slabs, strip footings, steel and concrete polybeams, and concrete panels.
- The end points of a part do not have corner chamfers. The handles that you select must be at corner points or between two segments of a part.

Chamfer part corners

When Tekla Structures creates a part, by default it has a rectangular chamfer at each corner, which does not change the geometry of the part. You can modify the default chamfers.



TIP To make it easier to select the handles of part corners, ensure that the

Direct modification switch  is **not** active.

1. Select the part.

2. Double-click the handle of a part corner.
The **Corner chamfer** properties open.
3. Modify the chamfer properties.
4. Select the handles of the part corners you want to modify.
5. Click **Modify** to apply the changes.

Chamfer part edges

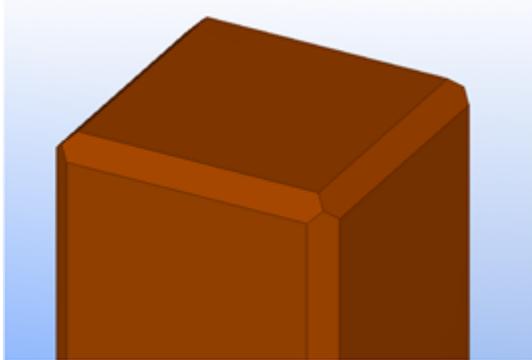
1. Double-click the view to open the **View Properties** dialog, click the **Display...** button, and ensure that the **Cuts and added material** option is **not** selected in the display settings.
2. On the **Edit** tab, click **Chamfer edge**.
Alternatively, you can start the command in the **Object type list** in the property pane.
3. Select the part you want to chamfer.
4. Pick a point where you want the chamfer to start on the part edge.
5. Pick a second point where you want the chamfer to end on the part edge.
Tekla Structures displays the chamfer in light blue.
6. If needed, you can modify the chamfer.
 - a. Double-click the chamfer to open the **Edge chamfer** properties.
 - b. [Change \(page 96\)](#) the chamfer properties.
 - c. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

TIP Alternatively, use the contextual toolbar to modify the edge chamfer.

7. Right-click the view and select **Redraw View**.

Tekla Structures removes the chamfered edge.



Corner chamfer properties

Use the **Corner chamfer** properties in the property pane to view and modify the properties of a corner chamfer. To open the properties, double-click the handle of chamfered corner.

The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

Setting	Description
Shape	
Type	Shape of the chamfer. See Corner chamfer types and dimensions for more information.
X / Distance X / Radius	Dimensions of the chamfer. The dimension depends on the chamfer type.
Y / Distance Y / Radius	
Dz1	Only used for contour plates and concrete slabs. Moves the top or bottom surface of the part corner in the part's local z direction. Use these options, for example, to give plates varying thicknesses.
Dz2	

Corner chamfer types and dimensions

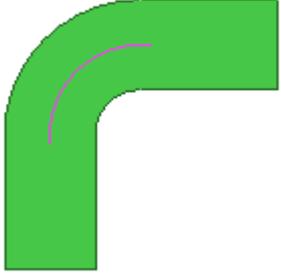
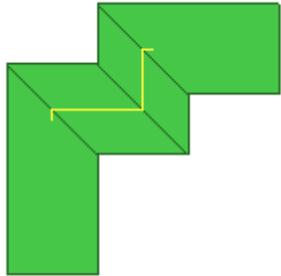
The table below describes the available corner chamfer types and dimensions. The chamfer type numbers can be used in sketches and custom components. Straight chamfers can have different dimensions in two directions. Curved chamfers only use one dimension.

Number	Type	Symbol	Dimensions
0	None		x: not used y: not used
1	Line		x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
2	Rounding		x: the radius y: not used
3	Arc		x: the radius y: not used
4	Arc point		x: not used y: not used
5	Square		The chamfer is perpendicular to the edges. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
6	Square parallel		The chamfer is parallel to the opposite edge. x: the distance in the x coordinate direction from the corner y: the distance in the y coordinate direction from the corner
7	Line and arc		x (if smaller than y): the arc radius x (if bigger than y): the distance in the x coordinate direction from the corner y (if smaller than x): the arc radius y (if bigger than x): the distance in the y coordinate direction from the corner

Status of corner chamfers in polybeams

To see the corner chamfer lines of polybeams, set the advanced option XS_DRAW_CHAMFERS_HANDLES to CHAMFERS or to CHAMFERS_AND_HANDLES.

Tekla Structures shows the status of polybeam chamfers using the following colors:

Color	Description	Example
Magenta	Correct chamfer	
Yellow	Correct chamfer that cannot be unfolded	
Red	Incorrect chamfer	

Edge chamfer properties

Use the **Edge chamfer** properties in the property pane to view and modify the properties of an edge chamfer. To open the properties, double-click the edge chamfer when the chamfer is visible in the model. The file name extension of an edge chamfer property file is *.cha.

The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

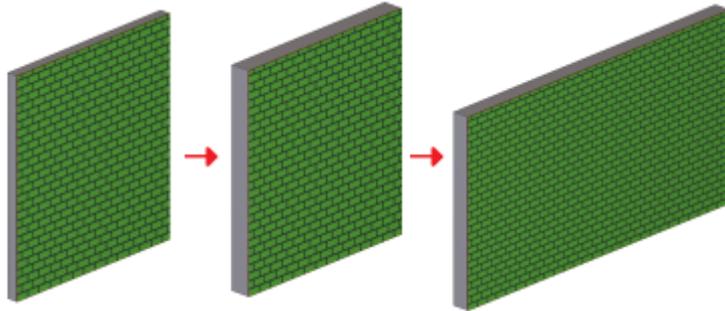
Setting	Description	More information
Shape on edge		
Type	Shape of the chamfer.	
Distance X	Defines how far away from the chamfered	

Setting	Description	More information
	edge the chamfer will end in the x direction.	
Distance Y	Defines how far away from the chamfered edge the chamfer will end in the y direction.	
Shape at end		
First end type	The shape and position of the first end point.	The options are: <ul style="list-style-type: none"> • Full: The end point is positioned at the end of the part (moving along the nearest edge), and the shape is straight. • Straight: The end point is positioned at the point you pick, and the shape is straight. • Bevelled: The end point is positioned at the point you pick, and the shape is angled.
Second end type	The shape and position of the second end point.	
Distance	The distance between the (picked) end point and the bevelled points.	
General		
Name	Name of the chamfer.	

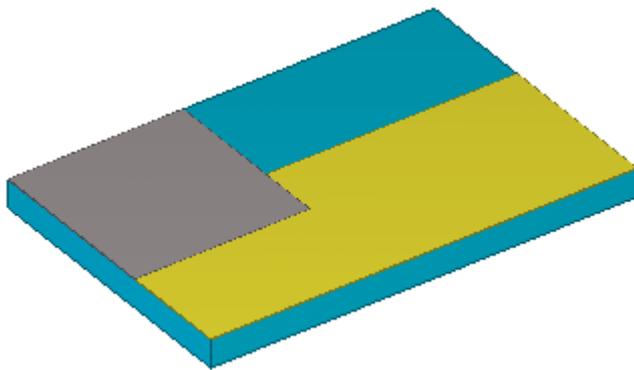
Add surface treatment to parts

Use the surface treatment tools to add surface treatment to parts. Surface treatment for concrete parts include flat finishes, surface mixes, and tiles. Surface treatment for steel parts include fire-proofing and unpainted areas, for example.

When you modify the shape or size of a part, Tekla Structures automatically modifies [the surface treatment to fit the part \(page 325\)](#).



When you create overlapping surface treatments, the smaller surface treatment overrides the larger one. The overlapping area is recognized in reports: only the topmost (visible) surface treatment is calculated.



Add surface treatment to an entire part face

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to part face** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.
4. Select the part to apply the surface treatment to.
 - a. Move the mouse cursor over a part. Tekla Structures highlights the faces that you can select.
 - b. Select the part face.

Add surface treatment to a selected area on a part face

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to selected area** .
2. Pick the origin of the surface treatment.
3. Pick a point to indicate the direction of the surface treatment.

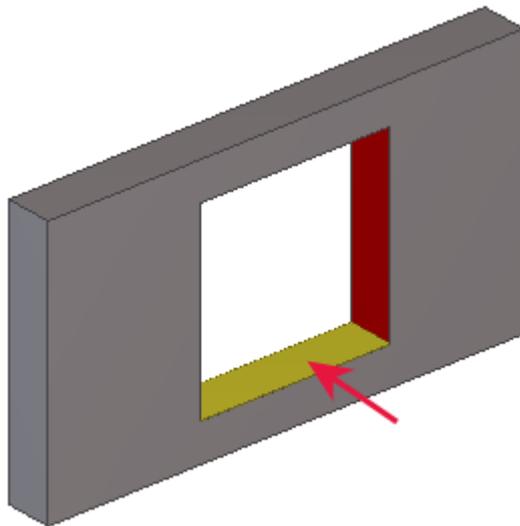
4. Select an area of the part face to apply the surface treatment to.
 - a. Move the mouse pointer over a part. Tekla Structures highlights the faces that you can select.
 - b. Select the part face.
 - c. Pick three or more points on the part face to define a polygonal area.

Add surface treatment to all faces of a part

1. On the **Edit** tab, click **Surfaces** --> **Surface treatment to all faces of part**.
2. Select the part to apply the surface treatment to.

Add surface treatment to cut faces

1. On the **Edit** tab, click **Surfaces** and then either **Surface treatment to part face** or **Surface treatment to selected area**.
2. Pick the origin of the surface treatment.
3. Pick the direction.
4. Select the cut face to apply the surface treatment to:



5. If you are using the **Surface treatment to selected area** command, pick the points to define the area of the surface treatment.

Surface treatment on chamfered parts

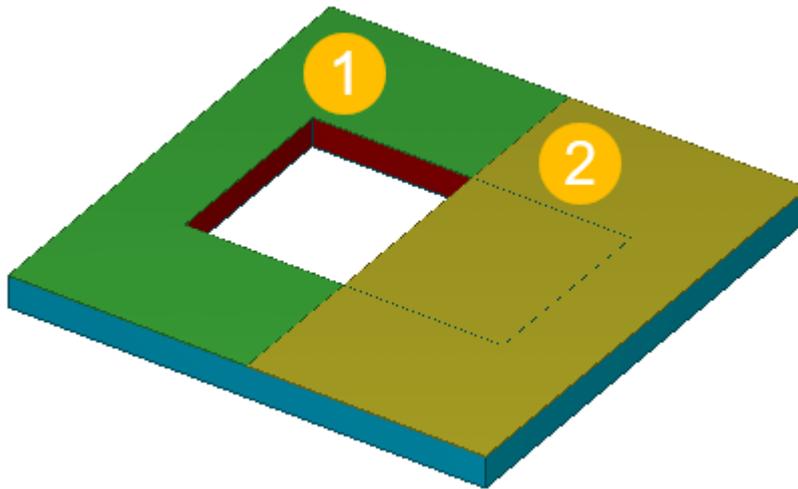
Take these things into account when adding surface treatment to chamfered parts:

- Surface treatment does not work on sketched profiles with chamfers.

- Add surface treatment before chamfering the part. If surface treatment is applied to a chamfered part, the surface treatment chamfer cannot be modified later on.
- The chamfers for the main part and surface treatment are separate. Modifying the main part chamfer does not affect the surface treatment chamfer.
- The orientation of unsymmetric chamfers depends on the face where it was created (such as top, bottom, left, or right). To change the orientation of an unsymmetric chamfer, you must swap the chamfer's x and y values.

Surface treatment on parts with openings and recesses

To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set the **Cut by father part cuts** to **Yes** in the **Surface treatment** properties.



- (1) The green surface treatment has the **Cut by father part cuts** set to **Yes**.
 (2) The tiled surface treatment is not cut by the cut in the part: **Cut by father part cuts** is set to **No**.

NOTE If you use the **Surface treatment to all faces of part** command and set the **Cut by father part cuts** to **Yes**, Tekla Structures automatically adds surface treatment also to the cut faces.

Modify surface treatment properties

1. If the property pane is not open, double-click the surface treatment to open the **Surface treatment** properties.
2. [Change \(page 96\)](#) the properties as needed.

- Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of the same type.

Note that if you modify the **Pattern** properties, you need to first click **Modify** in the property pane and then redraw the view to make the changes visible.

Surface treatment properties

Use the **Surface treatment** properties in the property pane to view and modify the properties of a surface treatment. To open the properties, double-click the surface treatment. The file name extension of a surface treatment property file is *.srf.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Name	User-definable name of the surface treatment.
Type	Select the type of surface treatment.
Subtype	Select the subtype of a specific surface treatment.
Material	Depending on the type of the surface treatment, select the material of the surface treatment.
Color	Depending on the type of the surface treatment, set the color of the surface treatment.
Thickness	Depending on the type of the surface treatment, enter the thickness of the surface treatment.
Cut by father part cuts	To force Tekla Structures to consider openings and recesses in parts when adding surface treatment, set this to Yes .
Position	
In depth	Select the location of the surface treatment, and set the Depth offset value.
Pattern (for tiled surface treatment)	
Pattern	If you have set Tile surface as the Type of the surface treatment, select

Setting	Description
	the pattern of the tiled surface treatment.
Tile width Tile height	Define the tile width and height.
Mortar height Mortar width	Define the mortar height and width.
Tile color Mortar color	If needed, use the color picker to pick the tile and the mortar color.
Custom properties	
More	Click the More button to open the user-defined attributes (UDAs) of the surface treatment. UDAs provide additional information about the surface treatment.

Define new surface treatment subtypes

You can add new options to the **Subtype** list in the **Surface treatment** properties by editing the `product_finishes.dat` file.

NOTE This section is for advanced users.

1. Copy the `product_finishes.dat` file to your firm, project, or model folder. The file is located under the `\ProgramData\Trimble\Tekla Structures\<version>\environments` folder. The exact file location may vary depending on the folder structure of your environment files.

2. Open the copied file using any text editor.

The first section of the file explains the available types of surface treatment. The surface treatment types are hard-coded, so do not modify this section:

```
// Product finishes
// -----
//
// Type          : Type of surfacing
//                1 = concrete finish
//                2 = special mix
//                3 = tile surface
//                4 = steel finish
```

3. Go to the sections that define the options for each type of surface treatment:

```
// =====
// *** Concrete Finish
// =====
// WET FINISH
// -----
1          MF          "Magnesium Float"
1          SMF         "Smooth Magnesium Float"
1          WT          "Wet Trowel"
```

4. Add rows to define new options.
 - a. Define the surface treatment type. For example, 1 for concrete finish.
 - b. Define a code for the surface treatment option. For example, MF for Magnesium Float.
 - c. Define the full name of the surface treatment option. For example, Magnesium Float. Remember to enclose the name in double quotes " ".
5. Save the file.

See also

[Add surface treatment to parts \(page 373\)](#)

Tiled surface treatment

Tekla Structures includes complex tile and brick surface treatment options, such as basketweave and herringbone patterns. Tiled surface treatment options are based on repeating tile patterns that are stored in XML format.

The tile pattern surface treatment are available in **Surface treatment** properties if you have set the **Type** to **Tile surface**.

NOTE This section is for advanced users.

Define a new tile pattern

1. Copy the `TilePatternCatalog.xml` file to your firm, project, or model folder. The file is located under the `\ProgramData\Trimble\Tekla Structures\<<version>\environments` folder. The exact file location may vary depending on the folder structure of your environment files.
2. Open the copied file using any text editor.
3. Add a new `<TilePattern>` element to the file.

The `<TilePattern>` element must have `<HOffset>` and `<VOffset>` elements and at least one `<Tile>` element. Other elements are optional.

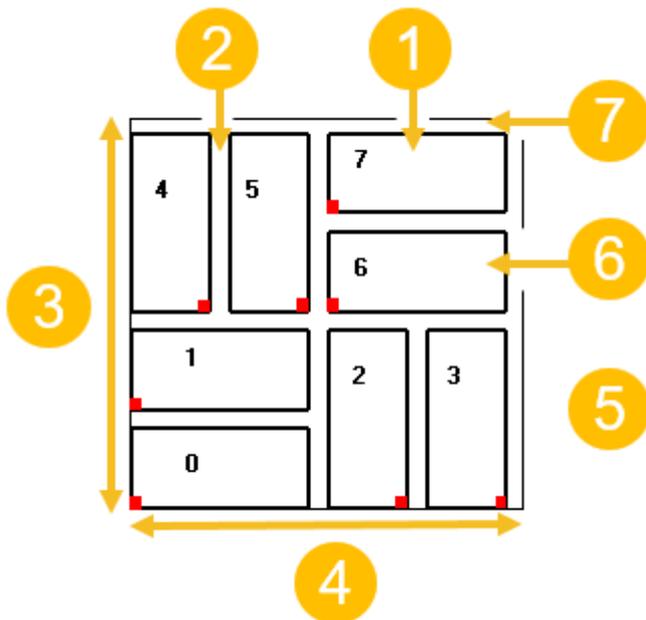
TIP You may find it easier to copy one of the existing elements, and then modify it to suit your needs.

- Repeat adding `<TilePattern>` elements for all the patterns you want to define.
- Save the `TilePatternCatalog.xml` file.

Example tile pattern definition

This example explains how the **Basketweave** tile pattern is defined in the `TilePatternCatalog.xml` file.

The **Basketweave** pattern block is made up of eight tiles:



(1) Tile width

(2) Mortar width

(3) `VOffset`

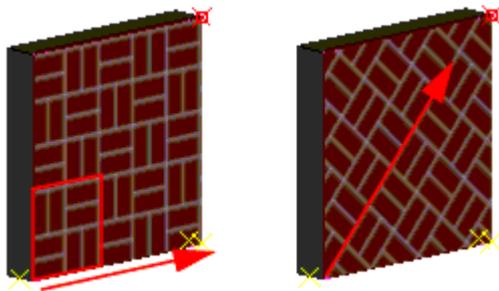
(4) `HOffset`

(5) Red marks indicate `TileOrigin`. Angle value for vertical tiles is 90.

(6) Tile height

(7) Mortar height

The pattern is repeated in the x and y direction of the surface treatment, starting from the origin of the surface treatment. You can run the pattern in different x directions:



In the `TilePatternCatalog.xml` file, the pattern is defined as follows:

```

<TilePattern Name="Basketweave">
  <Parameter Name="W" DefaultValue="220">
    <Label> _Tile_Width </Label>
  </Parameter>
  <Parameter Name="H" DefaultValue="100">
    <Label> _Tile_Height </Label>
  </Parameter>
  <Parameter Name="TH" DefaultValue="100">
    <Label> _Tile_Thickness </Label>
  </Parameter>
  <Parameter Name="MH" DefaultValue="20">
    <Label> _Mortar_Height </Label>
  </Parameter>
  <Parameter Name="MW" DefaultValue="20">
    <Label> _Mortar_Width </Label>
  </Parameter>
  <HOffset>
    <Vector2D X="W+2*H+3*MW" Y="0" />
  </HOffset>
  <VOffset>
    <Vector2D X="0" Y="W+2*H+3*MH" />
  </VOffset>
  <Tile Angle="0" Width="W" Height="H" Thickness="TH">
    <TileOrigin>
      <Vector2D X="0" Y="0" />
    </TileOrigin>
  </Tile>

```

1

2

3

- (1) Name of the pattern
- (2) Size of the pattern block in the x direction, after which the pattern repeats
- (3) Size of the pattern block in the y direction, after which the pattern repeats

The definition file uses the same symbols as the **Surface treatment** properties:



Tile pattern definitions

The predefined tile patterns that are available in the **Surface treatment** properties are stored in the following files:

File	Description
TilePatternCatalog.xml	<ul style="list-style-type: none"> Contains the tile pattern definitions. Located under the \ProgramData\Trimble\Tekla Structures\<version>\environments folder.
TilePatternCatalog.dtd	<ul style="list-style-type: none"> Document Type Declaration (DTD) file that defines the elements allowed in the TilePatternCatalog.xml file. Located in the same folder as the TilePatternCatalog.xml file.
Thumbnail images	<ul style="list-style-type: none"> The images that appear in the Pattern section in the Surface treatment properties. Located in the ..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps folder. File names identify the pattern types. For example, herringbone.bmp illustrates the herringbone pattern type.

Tile pattern elements

The `TilePatternCatalog.xml` file can contain the following elements:

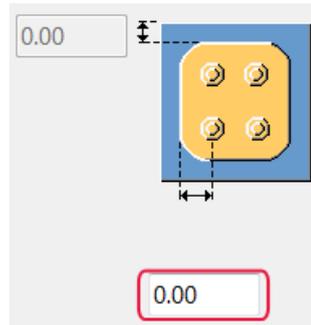
Element	Description
<code>TilePatternCatalog</code>	The container for tile patterns. Required.
<code>TilePattern</code>	Tile pattern element. Required. This element can contain the following elements listed in this table.
<code>HOffset</code>	Horizontal offset of the tile pattern. Required.
<code>VOffset</code>	Vertical offset of the tile pattern. Required.
<code>Tile</code>	The individual tiles used in a tile pattern. At least one required.
<code>Color</code>	Color of the tile or mortar, defined by the RGB values (0–255). Optional.
<code>Parameter</code>	Creates an attribute for any element in the <code>TilePattern</code> . Optional.
<code>Label</code>	The label that identifies a parameter in the Surface treatment properties. Optional.
<code>TileOrigin</code>	The origin of an individual tile, defined from the origin of the pattern. Optional.

Create an unpainted area using the No paint area component

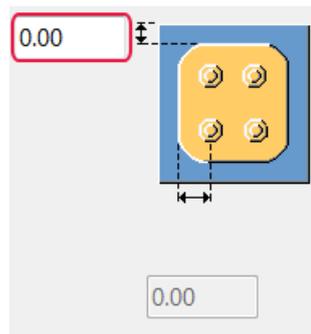
You can create an unpainted area between bolted steel parts using the **No paint area** component.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Search for **No paint area**.
3. On the **General** tab:
 - a. Click the **Load bolt standards** button to display the available bolt standards, and select the relevant standards.
 - b. Select the clearance location from the **Create for** list.

- Define the hole tolerance.



- Define the contact area offset.



- c. In the **Allowable gap** box, enter the maximum distance that can exist between two plates so that the surface treatment can be created.

4. On the **Surfacing attributes** tab:

- a. Select one of the following **Surfacing attributes**.

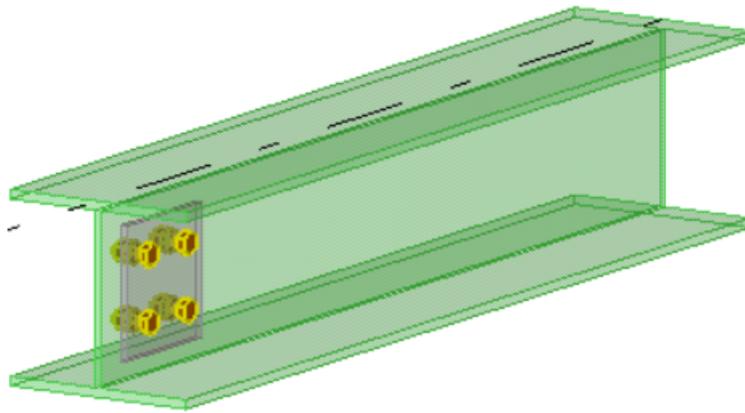
- standard surface treatment properties file
- A custom surface treatment properties file

You can create your own property files in the **Surface treatment** properties. The **Type** has to be **Steel finish** and the **Subtype** has to be **UP - Unpainted**.

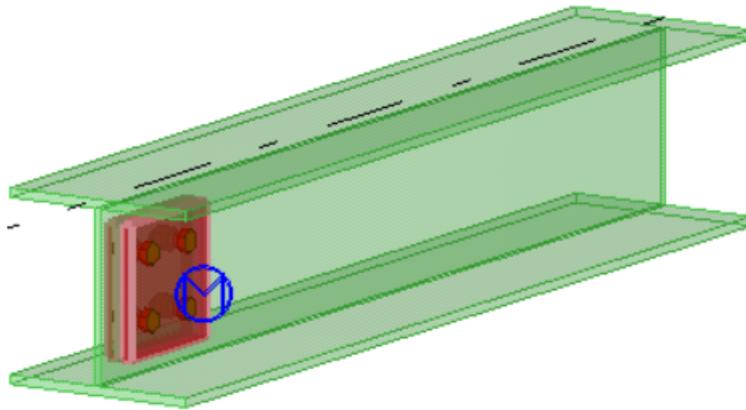
- ...

Define the custom attributes and the position for the surface treatment.

5. Click **OK**.
6. Select a bolt group in the model.



The unpainted area is created between the bolted parts.



See also

[Add surface treatment to parts \(page 373\)](#)

Add surfaces to part faces and pour object faces

You can add surfaces to the faces of parts and pour objects in the model. You can add surfaces to any face geometry, for example curved faces. You can use surfaces for calculating surface areas, such as formwork areas.

Surfaces are associated with the objects they are attached to. Surfaces cannot exist as standalone objects. A surface can be associated either with a cast-in-place part or a pour object, but not with both. Surfaces associated with parts or pours can be shown in reports as belonging to those objects.

If you modify the geometry of the associated part, the surface will adapt to the changes. If you delete or move the associated part, the surface will follow. If you copy an object that has a surface, the surface will not be copied. If a surface is added to a pour object, it will not automatically adapt to changes that only affect the pour object, such as inserting pour breaks.

Limitations:

- Surfaces do not recognize faces created by displaying objects with high accuracy, such as curved profile fillets.
- Surfaces do not have handles, so their geometry cannot be modified separately from the associated object.
- Surfaces cannot be copied.
- Surfaces cannot be moved or rotated without the associated object.
- Surfaces are not shown in drawings.

Add a surface to a face

1. On the **Edit** tab, click **Surfaces** --> **Add surface to face**.
2. Depending on whether you want to create the surface on a part or on a pour object, [use a part view or a pour view \(page 404\)](#).

To switch between part and pour views, click **Pour view** on the **Concrete** tab.

3. Select the part face or pour object face to which you want to add the surface.

Tekla Structures adds the surface using the **Surface** properties in the property pane.

If you modify the properties, Tekla Structures uses the new properties the next time you create an object of the same type.

Modify surface properties

1. If the property pane is not open, double-click the surface to open the **Surface** properties.
2. [Change \(page 96\)](#) the properties as needed.

For example, you can define the surface type and whether you want the surface to be cut by the holes in the part or pour object.

3. Click **Modify** to apply the changes.

Tekla Structures uses the properties the next time you create an object of the same type.

2.5 Create assemblies

This section explains how to turn steel parts into assemblies.

Tekla Structures creates assemblies of steel parts when you use a workshop weld or bolt to join parts together. Assemblies and their main parts are

automatically defined when you create single workshop welds or bolts, or when you apply automatic connections that create workshop welds or bolts.

Click the links below to find out more:

[Create an assembly \(page 387\)](#)

[Add objects to assemblies \(page 390\)](#)

[Change the assembly main part \(page 392\)](#)

[Change the main assembly \(page 393\)](#)

[Remove objects from an assembly \(page 393\)](#)

[Check and highlight objects in an assembly \(page 393\)](#)

[Explode an assembly \(page 394\)](#)

[Assembly examples \(page 394\)](#)

Create an assembly

1. Ensure that the  **Select assemblies** selection switch (page 122) is active.
2. Select the parts and/or assemblies that you want to join together.
3. Right-click and select **Assembly --> Make into Assembly** .

See also

[Create assemblies \(page 386\)](#)

[Create a sub-assembly \(page 387\)](#)

[Use bolts to create assemblies \(page 388\)](#)

[Use welds to create assemblies \(page 389\)](#)

Create a sub-assembly

You can create a sub-assembly of parts that are already in an assembly.

1. Ensure that the  **Select objects in assemblies** selection switch (page 122) is active.
2. Select the parts you want to include in the sub-assembly.
3. Right-click and select **Make into Sub-Assembly**.

See also

[Create an assembly \(page 387\)](#)

Use bolts to create assemblies

You can use bolts to create and connect assemblies. You can create nested assemblies by connecting sub-assemblies to an existing assembly, or you can just connect more parts to assemblies using bolts.

To control how Tekla Structures creates assemblies, use the **Connect as** and **Bolt type** lists in the **Bolt** properties. The order in which you select parts when creating the connection determines the main and secondary parts of the assembly or the assembly hierarchy.

Connect as	Bolt type	Result
As sub-assembly	Workshop or Site	Nested assembly with the assembly you are bolting as a sub-assembly. The first part you pick determines the assembly to which you are bolting.
As secondary part	Workshop	Basic assembly with the part you are bolting as a secondary part. The first part you pick usually becomes the main part in the assembly.
As secondary part	Site	No assembly created.

See also

[Create assemblies \(page 386\)](#)

[Bolt sub-assemblies to an existing assembly \(page 388\)](#)

Bolt sub-assemblies to an existing assembly

1. On the **Steel** tab, hold down **Shift** and click **Bolt**  to open the **Bolt** properties.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to bolt to.
4. Select a part in the sub-assembly to be bolted.
5. Pick the bolt group origin.
6. Pick a point to indicate the bolt group x direction.

See also

[Use bolts to create assemblies \(page 388\)](#)

Use welds to create assemblies

Tekla Structures forms assemblies based on where the weld should be made. You can create workshop welds and site welds.

The order in which you select parts when creating the connection determines the main and secondary parts of the assembly or the assembly hierarchy. The first part you select becomes the main part of the assembly. Tekla Structures dimensions secondary parts relative to the main part in assembly drawings. The largest main part in the weld becomes the main part of the assembly.

When you connect assemblies, the first part you select determines the assembly to which you weld sub-assemblies.

To control how Tekla Structures creates assemblies, use the **Connect as** and **Workshop/Site** lists in the **Weld** properties.

Connect as	Workshop/Site	Result
As sub-assembly	Workshop:  or Site: 	Nested assembly with the assembly you are welding as a sub-assembly. The first part you pick determines the assembly to which you are welding.
As secondary part	Workshop: 	Basic assembly with the part you are welding as a secondary part. The first part you pick usually becomes the main part in the assembly.
As secondary part	Site: 	No assembly created.

See also

[Create assemblies \(page 386\)](#)

[Weld sub-assemblies to an existing assembly \(page 389\)](#)

Weld sub-assemblies to an existing assembly

1. On the **Steel** tab, hold down **Shift** and click  to open the **Weld** properties.
2. In the **Connect as** list, select **As sub-assembly**.
3. Select a part in the assembly to weld to.
4. Select a part in the sub-assembly to be welded.
5. To check that the weld marks look correct, create a drawing.

See also

[Use welds to create assemblies \(page 389\)](#)

Add objects to assemblies

You can add objects to assemblies in the following ways:

To	Do one of the following
Create a basic assembly	<ul style="list-style-type: none">• Add parts to an existing assembly as secondary parts.• Bolt or weld parts to an existing assembly as secondary parts.
Create a nested assembly	<ul style="list-style-type: none">• Add parts to an existing assembly as secondary parts.• Bolt or weld assemblies to an existing assembly as sub-assemblies.• Add assemblies to an existing assembly as sub-assemblies.• Join existing assemblies together without adding any loose parts.

NOTE Sub-assemblies in a nested assembly retain their own assembly information and main part. You can also define properties separately for the sub-assemblies and the nested assembly in the part properties.

See also

[Create assemblies \(page 386\)](#)

[Assembly hierarchy \(page 390\)](#)

[Add parts to an assembly \(page 391\)](#)

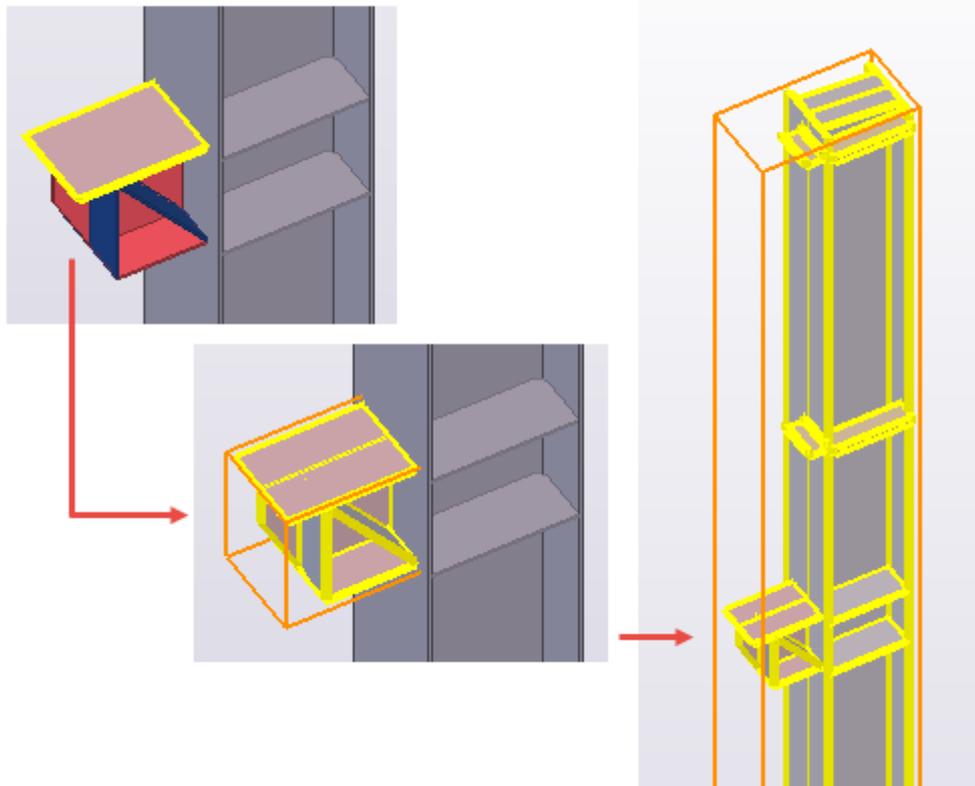
[Create a nested assembly \(page 392\)](#)

[Join assemblies \(page 392\)](#)

Assembly hierarchy

You can work on any level of a nested assembly, from single parts and bolts, through the basic and sub-assemblies, up to the highest level of the nested assembly. Hold down the **Shift** key and scroll with the mouse wheel to select

objects on different levels in the assembly hierarchy. For more information, see [Select nested objects \(page 127\)](#).



The assembly hierarchy in nested assemblies affects drawings and reports. You can create separate drawings and reports of the sub-assemblies and the nested assembly, and still produce dimensions, marks, fabrication information, etc. for all assembly levels.

See also

[Add objects to assemblies \(page 390\)](#)

Add parts to an assembly

You can add secondary parts to a basic assembly or to any level of a nested assembly.

1. Ensure that the  **Select objects in assemblies** [selection switch \(page 122\)](#) is active.
2. Select the part you want to add.
3. Right-click and select **Assembly --> Add to Assembly**.
4. Select the assembly to add to.

See also

[Add objects to assemblies \(page 390\)](#)

Create a nested assembly

1. Ensure that the  **Select assemblies** selection switch (page 122) is active.
2. Select the assemblies you want to add to another assembly. They will become sub-assemblies in the nested assembly.
3. Right-click and select **Assembly --> Add as Sub-Assembly**.
4. Select the assembly to add to.

See also

[Add objects to assemblies \(page 390\)](#)

Join assemblies

You can join existing assemblies without adding any loose parts.

1. Ensure that the  **Select assemblies** selection switch (page 122) is active.
2. Select the assemblies you want to join.
3. Right-click and select **Assembly --> Make into Assembly**.
The assembly with the largest volume becomes the main assembly.

See also

[Change the main assembly \(page 393\)](#)

[Add objects to assemblies \(page 390\)](#)

Change the assembly main part

The *main part* in a steel assembly has other parts welded or bolted to it. By default, the main part is not welded or bolted to any other parts. You can change the main part in an assembly.

1. If needed, [check \(page 393\)](#) what is currently the main part of the assembly.
2. Ensure that the  **Select objects in assemblies** selection switch (page 122) is active.
3. On the **Steel** tab, click **Assembly --> Set as main object**.

4. Select the new main part.
Tekla Structures changes the main part.

See also

[Add objects to assemblies \(page 390\)](#)

Change the main assembly

When you join two or more assemblies together, the assembly with the largest volume becomes the main assembly. You can change the main assembly in a nested assembly at any time.

1. Select the new main assembly.
2. Right-click and select **Assembly --> Set as New Main Sub-Assembly** .

See also

[Add objects to assemblies \(page 390\)](#)

Remove objects from an assembly

1. Select the part or sub-assembly you want to remove.
2. Right-click and select **Assembly --> Remove from Assembly**.

See also

[Create assemblies \(page 386\)](#)

Check and highlight objects in an assembly

Use the **Inquire** tool to check which objects belong to a particular assembly.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects**.
2. Select a part that belongs to an assembly.
Tekla Structures highlights the other parts that belong to the same assembly. The following colors are used:

Object type	Highlight color
Concrete - main part	magenta
Concrete - secondary part	cyan
Reinforcement	blue

Object type	Highlight color
Steel part - main part	orange
Steel part - secondary part	yellow

See also

[Create assemblies \(page 386\)](#)

Explode an assembly

When you explode a nested assembly, Tekla Structures breaks the assembly hierarchy level by level, always starting from the highest level. You need to use the **Explode** command several times to break a nested assembly back to single parts.

You can also explode sub-assemblies to single parts without breaking the entire assembly hierarchy.

1. Select the assembly or sub-assembly you want to explode.
2. Do one of the following:
 - To explode the entire assembly, right-click and select **Assembly --> Explode**.
 - To only explode the sub-assembly, right-click and select **Assembly --> Explode Sub-Assembly**.

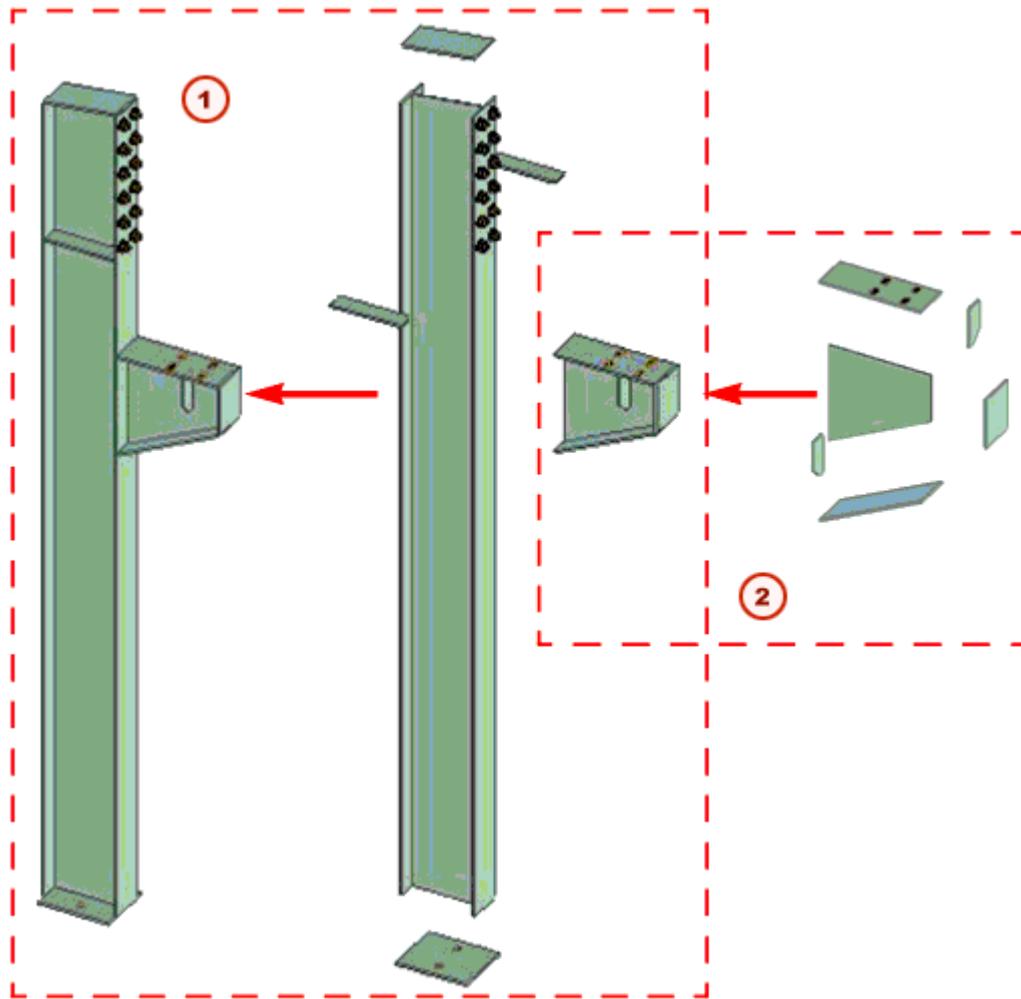
See also

[Create assemblies \(page 386\)](#)

Assembly examples

Column corbel

A column corbel is fabricated in one workshop, and then attached to the column in another workshop. Model the corbel as a sub-assembly of the column. Then create an assembly drawing for each workshop: one assembly drawing showing how the corbel is welded together, another assembly drawing showing how the corbel and the other part are welded to the column.



① Drawing 2, Workshop 2

② Drawing 1, Workshop 1

Complex truss

Model the halves of a complex truss as assemblies. Create assembly drawings for the workshop to fabricate the truss halves. Then create another assembly drawing showing how the halves should be joined on site.

Built-up profile

In a frame of built-up columns and beams, each built-up profile can be a sub-assembly. You can create an assembly drawing showing the entire frame, and separate drawings showing how the columns and beams are constructed.

See also

[Create assemblies \(page 386\)](#)

2.6 Create cast units

This section explains how to create cast units.

By default, each concrete part is considered a separate cast unit. For construction purposes, you may need to merge several concrete parts into one cast unit. For example, a single cast unit could consist of a column with corbels.

Click the links below to find out more:

[Define the cast unit type of a part \(page 396\)](#)

[Create a cast unit \(page 397\)](#)

[Add objects to a cast unit \(page 397\)](#)

[Change the cast unit main part \(page 398\)](#)

[Remove objects from a cast unit \(page 398\)](#)

[Check and highlight objects in a cast unit \(page 398\)](#)

[Explode a cast unit \(page 399\)](#)

[Casting direction \(page 399\)](#)

Define the cast unit type of a part

You must define the cast unit type of concrete parts. Tekla Structures checks the cast unit type of the main part each time you create or modify a cast unit. Precast and cast-in-place parts cannot be mixed within a cast unit.

1. Double-click a concrete part to open the part properties in the property pane.
2. Go to the **Cast unit** section.
3. In the **Cast unit** list, select one of the following:
 - **Cast in place**
Cast units that are constructed fully in their final location.
 - **Precast**
Cast units that are built at another location and transported to their final location for placement in the full structure.
4. Click **Modify** to save the changes.

NOTE It is important to use the correct cast unit type, because some functionalities, for example numbering, are based on the cast unit type.

See also

[Create cast units \(page 396\)](#)

Create a cast unit

You need to specify which parts form the cast unit. Cast units can include reinforcement, as well as concrete parts.

1. On the **Concrete** tab, click **Cast unit** --> **Create cast unit** .
2. Select the objects you want to include in the cast unit.
3. Click the middle mouse button to create the cast unit.

See also

[Create cast units \(page 396\)](#)

Add objects to a cast unit

You can use different methods to add objects to cast units. The available methods depend on the objects' material and on the hierarchy you want to create in the cast unit.

To add an object to a cast unit, do one of the following:

To	Do this	Available for
Add an object as a secondary part	<ol style="list-style-type: none">1. On the Concrete tab, click Cast unit --> Add to cast unit .2. Select the object you want to add.3. Select an object in the cast unit.	Concrete, timber, miscellaneous materials
Add an object as a sub-assembly	<ol style="list-style-type: none">1. If you are adding a custom part, ensure that the  Select components (custom objects) selection switch (page 122) is active.2. On the Steel tab, click Assembly --> Add as sub-assembly .3. Select the object you want to add.4. Select the cast unit to which you want to add the object.	Steel, concrete, timber, miscellaneous materials

See also

[Create cast units \(page 396\)](#)

Change the cast unit main part

The *main part* in a concrete cast unit is the one with the largest volume of concrete. You can change the main part in a cast unit.

1. If needed, [check \(page 398\)](#) what is currently the main part of the cast unit.
2. Ensure that the  **Select objects in assemblies selection switch** [\(page 122\)](#) is active.
3. Select the new main part.
4. Right-click and select **Set as New Main Part of Assembly**.

See also

[Add objects to a cast unit \(page 397\)](#)

Remove objects from a cast unit

1. On the **Concrete** tab, click **Cast unit --> Remove from cast unit**.
2. Select the objects you want to remove.

See also

[Create cast units \(page 396\)](#)

Check and highlight objects in a cast unit

Use the **Inquire** tool to check which objects belong to a particular cast unit.

1. On the ribbon, click the down arrow next to , and then select **Assembly objects**.
2. Select a part that belongs to a cast unit.

Tekla Structures highlights the other parts that belong to the same cast unit. The following colors are used:

Object type	Highlight color
Concrete - main part	magenta

Object type	Highlight color
Concrete - secondary part	cyan
Reinforcement	blue
Steel part - main part	orange
Steel part - secondary part	yellow

See also

[Create cast units \(page 396\)](#)

Explode a cast unit

1. On the **Concrete** tab, click **Cast unit** --> **Explode** .
2. Select an object in the cast unit you want to explode.

See also

[Create cast units \(page 396\)](#)

Casting direction

To indicate the casting direction of a concrete part, you can define which part face you want to face upwards in the casting form. The top-in-form face is displayed in the front view of a drawing.

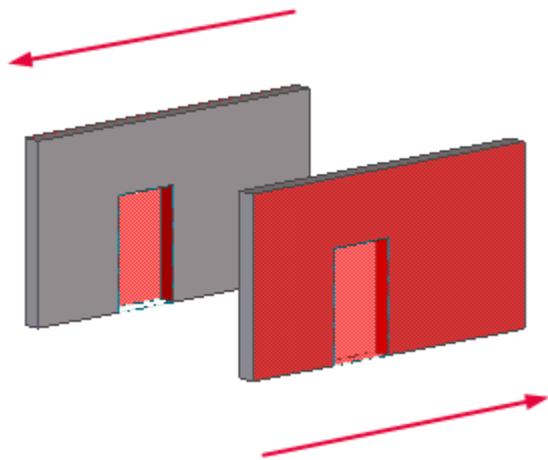
To enable this feature for non-concrete parts as well, and to indicate in the model the part face that is shown in the drawing main (front) view, use the advanced option `XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING`.

The casting direction affects the numbering of parts. If you define the casting direction for parts that differ only by their modeling direction, they get different position numbers. This is because the modeling direction affects the top-in-form face of the parts. By default, the casting direction of the parts is undefined, which means the modeling direction does not affect numbering.

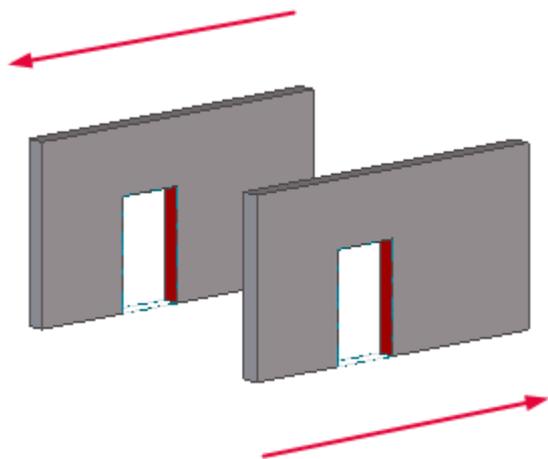
NOTE In drawings, use the **Fixed** coordinate system to show the top-in-form face in the front view.

Example

In the following example, each cast unit gets a **different** position number, because the top-in-form setting and the orientation of the panels is different. The red arrow indicates the modeling direction.



In the following example, the cast units get the **same** position number, because their top-in-form setting has not been defined. The red arrow indicates the modeling direction.



See also

[Create cast units \(page 396\)](#)

[Define the casting direction of a part \(page 400\)](#)

[Number the model \(page 627\)](#)

Define the casting direction of a part

You can define the casting direction for concrete parts.

1. Set the rendering of parts to **Rendered** by doing one of the following:
 - On the **View** tab, click **Rendering --> Parts rendered** .
 - Press **Ctrl + 4**.
2. Select a concrete part.
3. Right-click and select **Cast Unit --> Set Top in Form Face** .
4. Select the part face that will face upwards in the form.

TIP Alternatively, you can do this in the user-defined attributes of the part.

- Concrete parts: Select an option for the **Top in form face** user-defined attribute.
 - Non-concrete parts: With `XS_SET_FIXEDMAINVIEW_UDA_TO_AFFECT_NUMBERING` set to `STEEL`, `TIMBER`, and/or `MISC`, select an option for the **Fixed drawing main view** user-defined attribute.
-

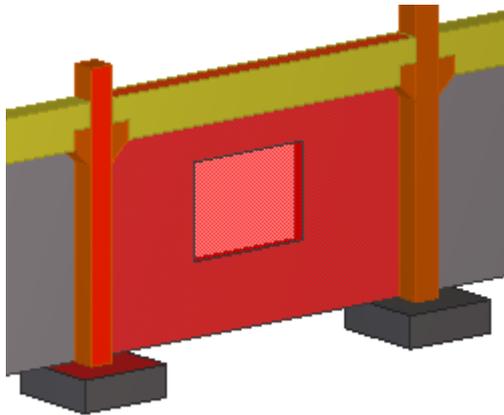
See also

[Casting direction \(page 399\)](#)

Show the top-in-form face

You can display the top-in-form face of a concrete part in a model view.

1. On the **Concrete** tab, click **Cast unit --> Show top-in-form face** .
2. Click the concrete part whose top-in-form face you want to show.
Tekla Structures highlights the top-in-form face in red:



TIP To hide the top-in-form face again, right-click the view and select **Update Window**.

See also

[Casting direction \(page 399\)](#)

2.7 Manage pours

With the pour management functionality of Tekla Structures, you can view the geometry of cast-in-place concrete structures, show them as parts or as pour objects, plan pours and pour breaks, and report pour information, such as concrete volumes and formwork areas. You can define pours, pour units, pour objects, and pour breaks for concrete parts whose cast unit type is **Cast in place**.

In Tekla Structures, a *pour object* is a building object that consists of one or more cast-in-place concrete parts, or parts of them. The cast-in-place concrete parts are merged into one pour object if they have the same material grade and they touch each other. They also need to be in the same *pour phase* to be merged. Pour objects are visible in *pour views*.

A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement, embeds, and other objects that need to be in place before concrete can be poured on the building site.

A *pour* is a group of pour objects that is poured at one go.

With a *pour break* you can split a pour object into smaller pour objects.

NOTE Pour management is mainly targeted at contractors for quantity take-off, planning, and on-site activities. By default, pour management is disabled in new models in most roles. You can [enable pour management \(page 402\)](#) in the current model using the advanced option XS_ENABLE_POUR_MANAGEMENT.

See also

[Enable pour management \(page 402\)](#)

[View cast-in-place concrete structures \(page 404\)](#)

[Define the pour phase of a part \(page 407\)](#)

[Pour objects \(page 408\)](#)

[Pour units \(page 410\)](#)

[Pour breaks \(page 415\)](#)

[Troubleshoot pours \(page 422\)](#)

[Example: Create concrete geometry and work with pours \(page 425\)](#)

Enable pour management

By default, pour management is disabled in new models in most roles. You can enable pour management in the current model in the **Advanced Options** dialog box.

WARNING If pour management is enabled in the model, do not disable it using `XS_ENABLE_POUR_MANAGEMENT`, especially in the middle of the project. This may cause problems if you have drawings containing pour objects, and if you are sharing your model. The pour objects and pour breaks in the model and in the drawings may get invalid, and you may lose all pour-related modeling work.

1. On the **File** menu, click **Settings** --> **Advanced options** to open the **Advanced Options** dialog box.
2. Under **Concrete Detailing**, set `XS_ENABLE_POUR_MANAGEMENT` to `TRUE`.
3. Click **OK**.
4. Save and re-open the model for the change to take effect.

The commands that show and create pour objects and pour breaks are now available in the model and in drawings.

See also

[Disable pour management temporarily \(page 403\)](#)

Disable pour management temporarily

You can temporarily disable pour management. This may be needed if pour management seems to significantly slow down your model, for example, when the pours and pour objects are very large and require splitting into smaller ones.

When pour management is temporarily disabled, the existing pour objects and pour breaks are still present in the model, but any modifications to the model geometry that would normally update pour objects and pour breaks automatically will not do that. Any information related to pours will be outdated and inaccurate, for example, in reports, and the pour breaks will not be adaptive. They will be updated automatically when you re-enable pour management.

To disable or re-enable pour management:

1. Go to **Quick Launch**, start typing `pours` and `pour breaks`, and select the **Toggle pours and pour breaks** command from the list that appears.
2. Click **Yes** in the confirmation dialog box.

NOTE If you are working in a Tekla Model Sharing model, remember to re-enable pour management before you write out. Similarly, if you are working in the multi-user mode, re-enable pour management before you save the model. In this way, information related to pours stays up to date for all users of the model.

TIP If you have problems opening a large model with pour objects that contain many parts, you may need to disable pour management before you open the model. You can do this by modifying the `xs_user.[user name]` file located in the model folder. Set `PAPB` to 0 to disable pours, and then save the file.

When needed, remember to re-enable pour management.

See also

[Enable pour management \(page 402\)](#)

View cast-in-place concrete structures

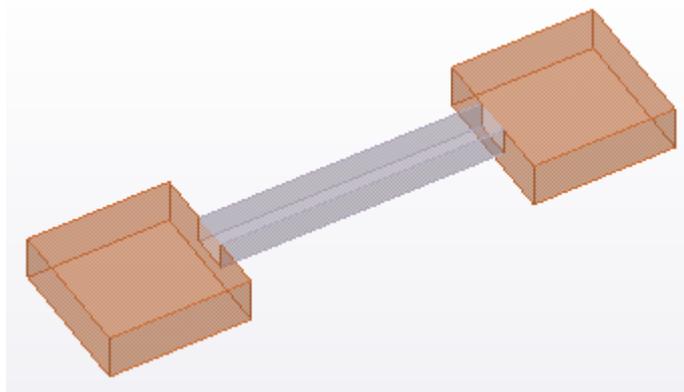
When pour management is enabled, you can view cast-in-place concrete structures in model views either as parts or as pour objects.

Depending on your needs, you can switch between the different representation options for cast-in-place concrete structures. For example, working in a part view is useful when you want to reinforce individual parts or change their geometry. A pour view is useful when you want to find out the volume of concrete to be poured or check which objects belong to a pour unit, or when you want to reinforce continuous structures that span multiple parts.

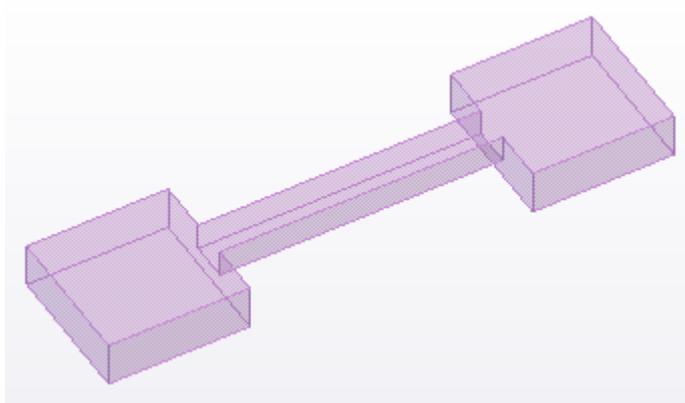
Set the appearance of cast-in-place concrete structures

You can define how cast-in-place concrete structures appear in a model view.

1. Ensure that pour management is [enabled \(page 402\)](#).
2. Double-click the view to open the **View Properties** dialog box.
3. Click **Display** to open the **Display** dialog box.
4. Ensure that the **Parts** check box is selected.
5. In the **Cast in place** list, select either:
 - **Parts**



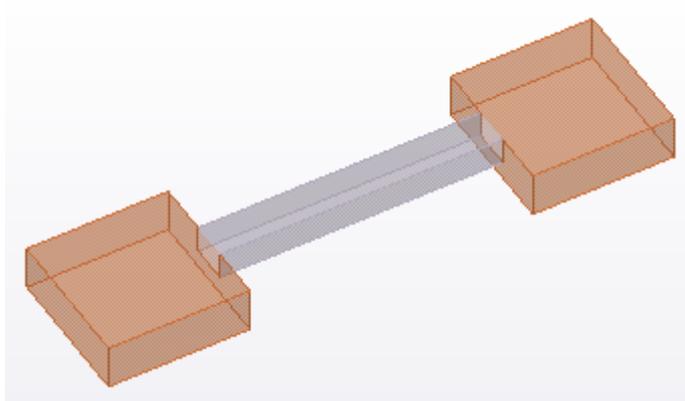
- **Pours**



6. If you selected **Parts** for the cast-in-place concrete structures, in the **Cast in place parts** list, select either:

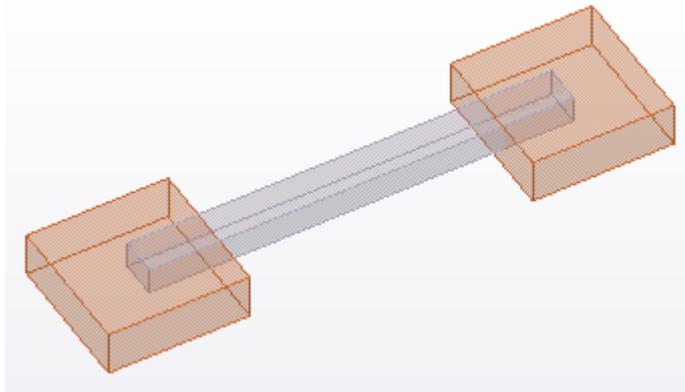
- **Merged**

Tekla Structures shows concrete parts as merged in the model if their cast unit type is **Cast in place**, if they have the same material grade and [pour phase \(page 407\)](#) number, and if they touch or overlap one another. When these criteria are met, Tekla Structures removes the outlines of the individual parts within each continuous concrete structure.



- **Separated**

Tekla Structures shows concrete parts as individual parts and separated by their outlines.



7. Ensure that the view is selected.
8. Click **Modify** to save the changes.

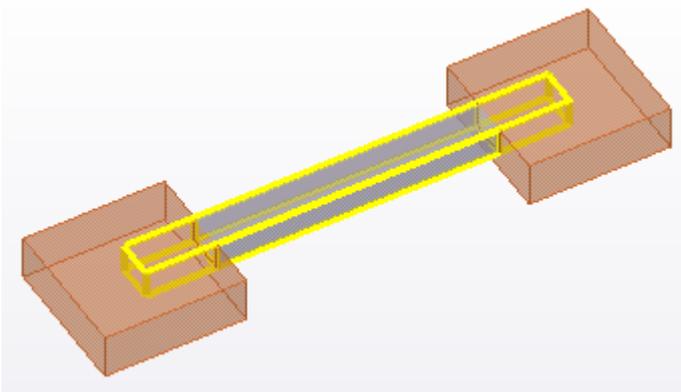
TIP To quickly change the representation of the active view from **Parts** to **Pours**, and

the other way round, click  **Pour view** on the **Concrete** tab.

You can also create two views, a pour view and a part view, and keep them both open side by side on your screen.

Part view vs. pour view

Continuous concrete structures cannot be selected or highlighted in part views. When you hold the mouse pointer over a concrete structure in a part view, Tekla Structures highlights the original parts belonging to it. You can select a part and modify it if needed:



Duplicates and overlapping parts are counted only once in the volume calculations for pour objects. Note that single part and cast-unit volumes are still calculated the same way as before, which means that the sum of single

part and cast-unit volumes may be higher than the volume of pour objects that are defined from exactly the same part geometry.

When you reinforce a concrete structure, you need to reinforce the individual concrete parts within it in part views, or you can reinforce pour objects by using **Reinforcing Bar Shape Catalog** or rebar sets in pour views. Therefore, you can reinforce a part of a continuous concrete structure independently from the whole continuous concrete structure. All reinforcement are visible both in part views and in pour views.

Define the pour phase of a part

Use the pour phase property to separate pour objects from one another. By defining pour phases, you can prevent cast-in-place concrete parts from merging even if they have the same material grade and they touch or overlap one another.

NOTE Pay attention to pour phases when you create cast-in-place concrete parts. For example, use pour phase 0 for horizontal structures, like beams and slabs, and pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects. In this way you can make sure that the number of parts included in each single pour object is reasonable, and that your models do not slow down due to too large pour objects.

To modify the pour phase of a part:

1. Double-click a concrete part to open the part properties in the property pane.
2. Under **Cast unit**:
 - a. In the **Cast unit type** list, ensure that the cast unit type is set to **Cast in place**.
 - b. In the **Pour phase** box, enter a pour phase.

By default, the pour phase is 0. If you are unable to change the value, the cast unit type that you set in step 2a is incorrect.
3. Click **Modify**.

NOTE When you define pour phases, make sure that parts in different pour phases do not overlap. If you use parts (not pour objects) to report geometric information, overlapping volumes of different pour phases are not merged but counted twice in calculations, and you may end up with incorrect volume, area, or weight information.

See also

[View cast-in-place concrete structures \(page 404\)](#)

Pour objects

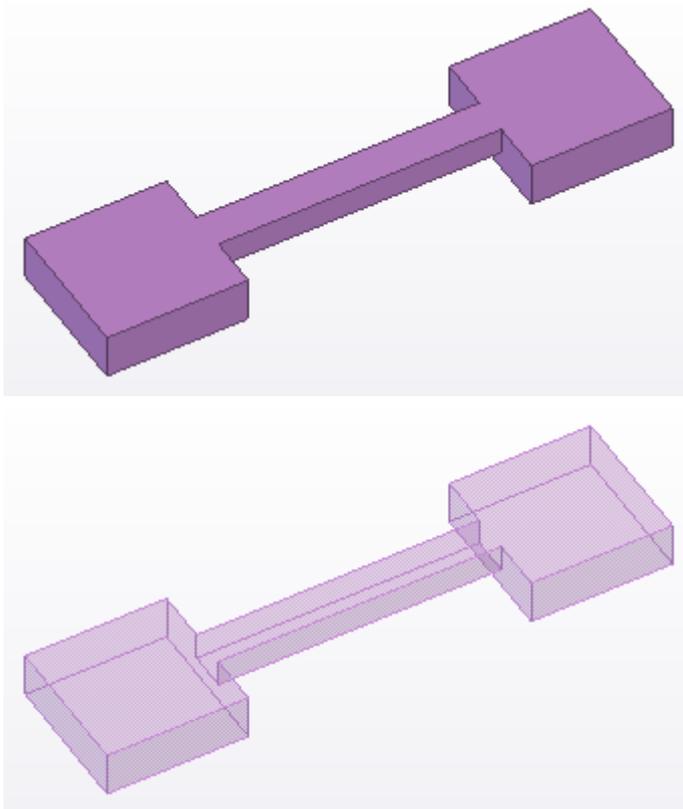
When pour management is [enabled \(page 402\)](#), each concrete part whose cast unit type is **Cast in place** automatically forms a pour object.

Tekla Structures automatically merges multiple cast-in-place concrete parts to a pour object if they have the same material grade and [pour phase \(page 407\)](#) number, and if they touch or overlap one another.

By creating [pour breaks \(page 415\)](#) you can split pour objects into smaller pour objects.

NOTE Make sure that the number of parts included in each single pour object is reasonable. Too large number of parts and part surfaces in a pour object slows down the model.

Pour objects are visible in [pour views \(page 404\)](#). All pour objects are displayed using the same color, regardless of the individual parts' color within a concrete structure. You can change the default color by using the advanced option XS_POUR_OBJECT_COLOR in **File --> Settings --> Advanced options --> Concrete Detailing**.



You can also use different [color and transparency settings \(page 409\)](#) to show pour object groups, for example, by pour number.

TIP You can group pour objects using **Organizer** or **Task manager**.

Limitations

The following commands are not available for pour objects: **Copy**, **Move**, **Delete**, **Split**, and **Combine**. This is because pour object geometry is defined by parts. If you want to change the geometry of pour objects, you have to modify the parts instead of pour objects, or you can create pour breaks.

See also

[Modify the properties of a pour object \(page 410\)](#)

[Reinforce pour objects using Rebar shape catalog \(page 459\)](#)

[Create a rebar set \(page 428\)](#)

Change the color and transparency of pour objects

By default, all pour objects are displayed using the same color in pour views, regardless of the individual parts' color. You can customize the pour object color and transparency in model views by defining object groups and then selecting specific color and transparency settings for each group.

TIP To change the default color of pour objects, use the advanced option XS_POUR_OBJECT_COLOR in **File --> Settings --> Advanced options --> Concrete Detailing** .

1. On the **View** tab, click **Representation**.

The **Object Representation** dialog box is displayed.

2. Create a new object group for the pour objects whose color and transparency you wish to change.
 - a. In the **Object Representation** dialog box, click **Object group....**
 - b. In the **Object Group - Representation** dialog box, click **Add row**.
 - c. To direct the settings to pour objects instead of parts, select the following options for the row:
 - **Category = Object**
 - **Property = Object type**
 - **Condition = Equals**
 - **Value = Pour object**
 - d. If needed, add any additional filtering criteria.

For example, to filter pour objects by a certain user-defined attribute, add a row with **Pour object** as **Category**, and define the **Property**, **Condition**, and **Value** as desired.

- e. Enter a unique name in the box next to the **Save as** button.
 - f. Click **Save as** to save the object group.
 - g. Click **Close**.
3. Repeat step 2 if you wish to create more object groups.
 4. In the **Object Representation** dialog box, select an object group from the **Object group** list.
 5. In the **Color** list, choose a color for the object group.
 6. In the **Transparency** list, set the transparency of the object group.
 7. Click **Modify**.
The object group's color and transparency changes in the model.

See also

[Define color and transparency settings \(page 597\)](#)

[Define your own colors for object groups \(page 595\)](#)

Modify the properties of a pour object

Pour objects have properties and user-defined attributes which you can view, define, and modify.

For example, you can enter a **Pour number** that you can use to define the sequence of pours, and a **Pour type** that you can use to describe each pour object.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select objects in assemblies** [selection switch \(page 122\)](#) is active.
3. Double-click the pour object whose properties you want to modify.
4. In the property pane, enter or modify the pour object properties.
5. Click **Modify**.

See also

[Pour objects \(page 408\)](#)

Pour units

When pour management is enabled, you can create pour units that combine together pour objects and other objects. A *pour unit* is an entity for cast-in-place concrete and it consists of a pour object and all related reinforcement,

embeds, and other objects that need to be in place before concrete can be poured on the building site.

For each [pour object \(page 408\)](#) in the model, there is a corresponding pour unit to which the pour object belongs. You can automatically add other objects to pour units by using the **Calculate pour units** command. You can also modify the pour units manually.

The following model objects can be added to pour units:

- Reinforcement, such as single reinforcing bars, bar groups, reinforcement meshes and strands
- Assemblies (for example, embeds)
- Sub-assemblies (for example, embeds in cast-in-place cast units)
- Bolts (for example, anchor bolts and shear studs)
- Precast cast units
- Surfaces added to the pour object

Note that some model objects such as parts and welds cannot be directly added to a pour unit. Instead, these objects are indirectly linked to the pour unit through the assemblies and cast units they belong to.

One model object can be included in only one pour unit at a time.

Calculate pour units

You can have Tekla Structures detect which objects form pour units and automatically add the objects to pour units.

1. Ensure that pour management is [enabled \(page 402\)](#).
2. On the **Concrete** tab, click **Calculate pour units**.

Tekla Structures [adds the objects \(page 414\)](#) to pour units.

You can check the pour units in a pour view, or using the **Inquire** tool, **Organizer**, or reports.

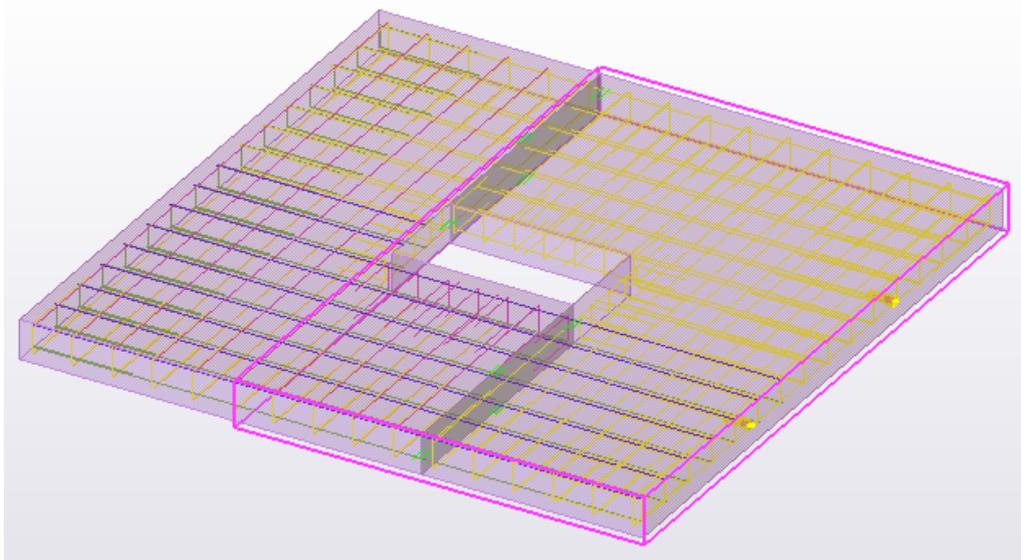
If you want to modify the pour units, you can manually add and remove objects. The manual additions will be kept even if you use the **Calculate pour units** command again, but the objects you manually remove from pour units will be added again.

Check and inquire objects in a pour unit

You can visually check which objects are included in a pour unit. You can also use the **Inquire** tool to get information about the pour unit and the objects in it.

1. Ensure that you are using a [pour view \(page 404\)](#). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.

2. Ensure that the  **Select assemblies** selection switch (page 122) is active.
3. Click a pour object to select its pour unit.
A magenta box indicates the pour unit.



4. Right-click and select **Inquire** .
Tekla Structures lists the objects in the pour unit and shows their properties in the **Inquire object** dialog box.

Add objects to a pour unit

In addition to using the **Calculate pour units** command, you can manually add objects to pour units.

1. Ensure that you are using a [pour view \(page 404\)](#). If not, on the **Concrete** tab, click **Pour view** to show the pour objects.
2. Select the objects that you want to add to a pour unit.
You can add reinforcement, assemblies, precast cast units, and bolts.
If you have other objects selected, they will not be added.
3. Right-click and select **Pour --> Add to pour unit**.
Alternatively, you can go to **Quick Launch** and use the **Add selected objects to pour unit** command. You can also [assign a keyboard shortcut \(page 195\)](#) to this command.
4. Click a pour object to add the objects to the corresponding pour unit.
Tekla Structures adds all the objects that can be added to the pour unit. The unallowed objects are not added.

TIP If you do not have objects selected, you can first start the **Add to pour unit** command by using **Quick Launch** or a customized [keyboard shortcut \(page 195\)](#), and then select the object that you want to add to the pour unit.

Remove objects from a pour unit

After using the **Calculate pour units** command, you can manually remove objects from pour units.

1. Select the object that you want to remove from a pour unit.
2. Right-click and select **Pour --> Remove from pour unit**.

Alternatively, you can go to **Quick Launch** and use the **Remove selected objects from pour unit** command. You can also [assign a keyboard shortcut \(page 195\)](#) to this command.

The removed object can then be added to another pour unit either manually with the **Pour --> Add to pour unit** command, or automatically with the **Calculate pour units** command.

TIP If you do not have objects selected, you can first start the **Remove from pour unit** command by using **Quick Launch** or a customized [keyboard shortcut \(page 195\)](#), and then select the object that you want to remove from the pour unit.

Reset pour unit relations

In some cases you may need to reset all or some of the pour unit contents and relations defined using the **Calculate pour units** command and/or the **Add to pour unit** command.

To do this:

1. Go to **Quick Launch**.
2. Search for and select the appropriate command of the following:
 - **Reset all pour unit relations**
 - **Reset all manually assigned pour unit relations**
 - **Reset all pour unit relations except manual assignments**
3. In the confirmation dialog box, click **Yes** to reset pour unit relations.

Note that if you use the **Reset all pour unit relations** command, and then click **No** in the confirmation dialog box to cancel the resetting of the manual assignments, the automatic relations are still reset. If you want to recreate the automatic relations, use the **Calculate pour units** command again.

Modify the properties of a pour unit

You can modify the pour unit properties in the same way as pour object properties, but using a different selection switch.

1. Ensure that you are using a [pour view \(page 404\)](#). If not, click **Pour view** on the **Concrete** tab to show the pour objects.
2. Ensure that the  **Select assemblies selection switch (page 122)** is active.
3. Double-click the pour unit whose properties you want to modify.
4. In the property pane, enter or modify the pour unit properties.
For example, you can define the pour unit name and user-defined attributes.
5. Click **Modify**.

How Tekla Structures automatically adds objects to pour units

When you use the **Calculate pour units** command, Tekla Structures automatically adds objects to pour units.

Each object that collides with a pour object, meaning that the object at least partly overlaps the pour object, is added to the same pour unit to which the pour object belongs.

If any object in an assembly or precast cast unit collides with a pour object, the entire assembly or cast unit is added to the pour unit.

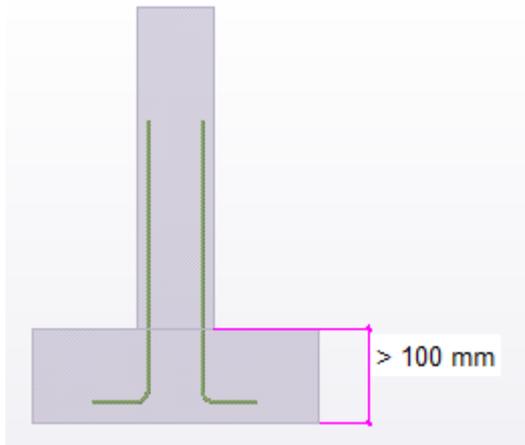
Only reinforcement objects that belong to cast-in-place parts are added to pour units.

If any object in a reinforcing bar group or strand group collides with a pour object, the entire group is added to the pour unit. On the other hand, individual reinforcing bars in a rebar set can be added to different pour units.

Objects colliding with more than one pour object

If an object collides with more than one pour object, the object will be associated with the pour object whose bounding box has the lowest global z coordinate.

For example, the reinforcing bars colliding with a footing pour object and a column pour object are associated with the footing pour object because its bottom face has a lower global z coordinate than the column pour object.



If the lowest global z coordinates of the pour object bounding boxes are the same or differ less than 100 mm, the object will be associated with one of the pour objects according to these rules:

1. If the object's center of gravity is within only one of the colliding pour object bounding boxes, the object will be associated with that pour object.
2. If the object's center of gravity is within more than one pour object bounding box, or completely outside any pour object bounding box, the object will be associated with the pour object whose center of gravity is closest to the object's center of gravity.

If there are changes in pour units

Whenever there are changes in a pour object or pour unit, all associations to that pour unit are reset. Similarly, if there are changes in an object associated with the pour unit, this association is reset. The next time you use the **Calculate pour units** command, only the unresolved associations are calculated.

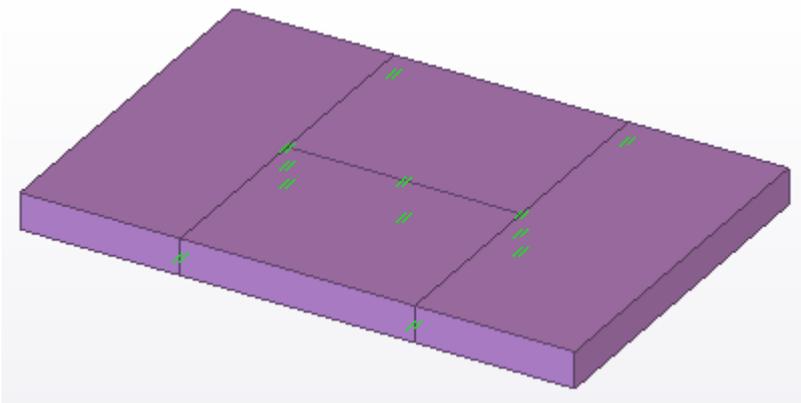
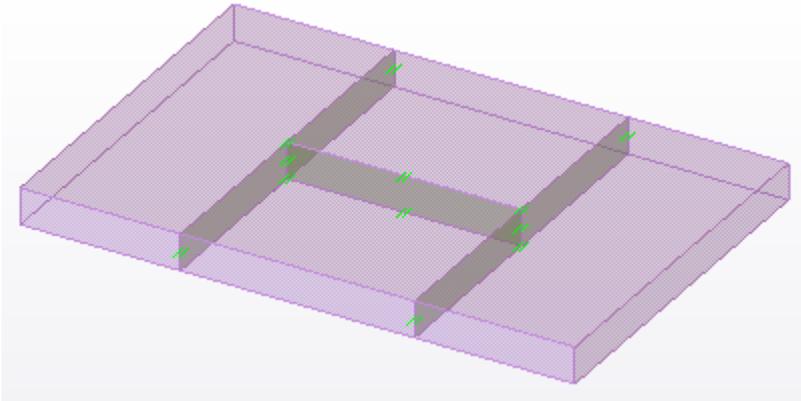
See also

[Pour units \(page 410\)](#)

Pour breaks

When pour management is enabled, you can use pour breaks to split pour objects into smaller pour objects.

Pour breaks are visible and you can work with them both [in pour views](#) and [in part views \(page 404\)](#). Pour breaks are displayed as a thin plane or line, depending on the [part rendering option \(page 584\)](#) you use.



WARNING If you move or copy a part, pour breaks do not follow the part. Pour breaks remain at their original locations, and [adapt to any cast-in-place concrete parts \(page 417\)](#) they still touch.

If a pour break does not split a pour object completely into two, the pour break is displayed in red by default. This means that it is invalid and needs to be remodeled.

See also

[Set the visibility of pour breaks \(page 417\)](#)

[Create a pour break \(page 418\)](#)

[Select a pour break \(page 420\)](#)

[Copy a pour break \(page 420\)](#)

[Move a pour break \(page 420\)](#)

[Modify a pour break \(page 421\)](#)

[Remove a pour break \(page 422\)](#)

Pour break adaptivity

Pour breaks are adaptive to changes in cast-in-place concrete parts and pour objects. This means that if you change the geometry or location of a cast-in-place concrete part or pour object, its pour breaks change accordingly.

If you delete a cast-in-place concrete part, its pour breaks disappear as well.

If you modify a cast-in-place concrete structure in any of the following ways, its pour breaks adapt:

- Change the profile or dimensions of a part
- Add or remove cuts or fittings
- Change chamfer shape or dimensions
- Add or remove parts of the cast-in-place concrete structure by:
 - Changing the cast unit type of a part from **Precast** to **Cast in place** or vice versa
 - Changing the pour phase of a part
 - Changing the concrete grade of a part
 - Moving, copying, or deleting parts

If you move a cast-in-place concrete part outside its pour breaks, the pour breaks disappear. If you move a part so that it still hits one or more pour breaks, the pour breaks that are inside the part stay in their original locations and adapt to the part in the new location.

If you copy or move a pour break and it hits a cast-in-place concrete part in the destination location, the pour break adapts to the part. Also the pour breaks that you copy from another model adapt to the parts in the model they are copied to.

If a pour break is dependent on another pour break that is split or deleted, the dependent pour break is deleted as well. If a pour break is dependent on another pour break that is moved, the dependent pour break adapts inside the pour object, as long as the pour break plane can touch the moved pour break.

If a pour break is split so that it becomes partial, the pour break is deleted. A partial pour break can only split a cast-in-place part or pour object in conjunction with other pour breaks.

Set the visibility of pour breaks

You can show pour breaks in model views.

Before you start, ensure that pour management is [enabled \(page 402\)](#).

1. Double-click a model view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.

3. Select the **Pour break** check box.
4. Click **Modify**.

See also

[Pour breaks \(page 415\)](#)

Create a pour break

You can add pour breaks to pour objects or concrete parts whose cast unit type is **Cast in place**.

You can create pour breaks by picking one, two, or more points in the model.

When you create a pour break that traverses more than two points, the pour break will be limited to the pour object it splits and perpendicular to the current work plane. If you need to create an inclined or horizontal pour break using multiple points, [shift the work plane \(page 51\)](#) first.

TIP Use the  **Snap to nearest points (points on line) snap switch** [\(page 79\)](#) to start or end pour breaks on part or pour object edges.

Use the  **Snap to any position snap switch** [\(page 79\)](#) to pick intermediate points for pour breaks.

To create a pour break, do any of the following:

To	Do this
Create a pour break, perpendicular to a part face, using one point	<ol style="list-style-type: none"> 1. On the Concrete tab, click Pour break --> Single point . 2. Pick the location for the pour break.
Create a pour break that splits all cast-in-place concrete parts and pour objects located between two points	<ol style="list-style-type: none"> 1. On the Concrete tab, click Pour break --> Two points . 2. Pick two points to define the location of the pour break.
Create a pour break using multiple points	<ol style="list-style-type: none"> 1. If needed, shift the work plane. 2. On the Concrete tab, click Pour break --> Multiple points . 3. Pick the points you want the pour break to go through.
Create a pour break defined by the opposite corners of a rectangle	<ol style="list-style-type: none"> 1. If needed, shift the work plane. 2. On the Concrete tab, click Pour break --> Multiple points .

To	Do this
	3. Hover over  , and then click  on the toolbar that appears. 4. Pick two opposite corner points of the pour break.
Create a pour break defined by the center and one corner of a rectangle	1. If needed, shift the work plane. 2. On the Concrete tab, click Pour break --> Multiple points . 3. Hover over  , and then click  on the toolbar that appears. 4. Pick the center point of the pour break. 5. Pick one corner point of the pour break.
Create a pour break defined by three corners of a rectangle	1. If needed, shift the work plane. 2. On the Concrete tab, click Pour break --> Multiple points . 3. Hover over  , and then click  on the toolbar that appears. 4. Pick three corner points of the pour break.
Create a pour break defined by one side midpoint and two corners of a rectangle	1. If needed, shift the work plane. 2. On the Concrete tab, click Pour break --> Multiple points . 3. Hover over  , and then click  on the toolbar that appears. 4. Pick one side midpoint of the pour break. 5. Pick two corner points of the pour break.

If the pour break you are creating does not split a pour object or cast-in-place part completely into two, Tekla Structures does not add the pour break to the model. You may need to use another **Pour break** command to create a valid pour break, for example **Multiple points** instead of **Single point**.

See also

[Select a pour break \(page 420\)](#)

[Copy a pour break \(page 420\)](#)

[Move a pour break \(page 420\)](#)

[Modify a pour break \(page 421\)](#)

Select a pour break

1. Ensure that the  **Select pour breaks** [selection switch \(page 122\)](#) is active.
2. Select the pour break.

See also

[Pour breaks \(page 415\)](#)

Copy a pour break

1. Ensure that the  **Select pour breaks** [selection switch \(page 122\)](#) is active.
2. Select the pour break.
3. [Copy \(page 133\)](#) the pour break like any other object in Tekla Structures. For example, right-click and select **Copy**.

See also

[Pour breaks \(page 415\)](#)

Move a pour break

You can move existing pour breaks. This may be needed, for example, if you have moved the part, because the pour break does not follow the part.

1. Ensure that the  **Select pour breaks** [selection switch \(page 122\)](#) is active.
2. Select the pour break.
3. [Move \(page 144\)](#) the pour break like any other object in Tekla Structures. For example, right-click and select **Move**.

See also

[Pour breaks \(page 415\)](#)

[Modify a pour break \(page 421\)](#)

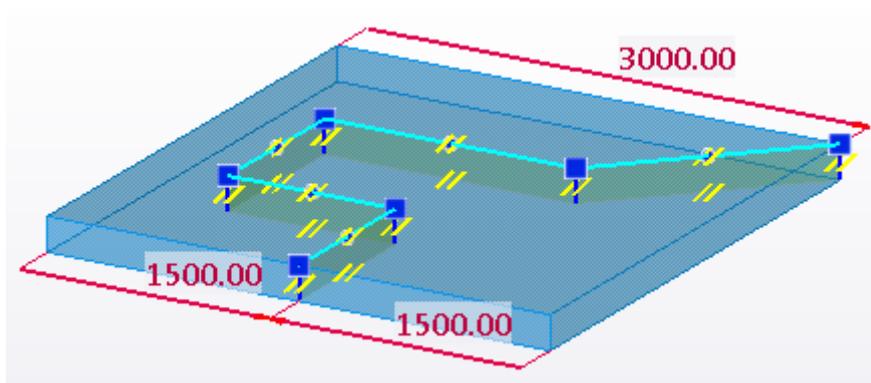
Modify a pour break

You can modify existing pour breaks.

Before you start:

- Ensure that the  **Direct modification** switch is active.
- Ensure that the  **Select pour breaks** selection switch ([page 122](#)) is active.
- Select the pour break.

Tekla Structures displays the handles and dimensions that you can use to modify the pour break.



To modify a pour break:

To	Do this
Change the shape or location of the pour break	Drag a corner point or an end point to a new location.
Change a location dimension	Drag a dimension arrowhead to a new location, or: <ol style="list-style-type: none">1. Select the dimension arrowhead which you want to move.2. Using the keyboard, enter the value with which you want the dimension to change. To start with the negative sign (-), use the numeric keypad. To enter an absolute value for the dimension, first enter \$, then the value.

To	Do this
	3. Press Enter , or click OK in the Enter a Numeric Location dialog box.
Add an intermediate point to the pour break	Drag a midpoint handle  to a new location.
Remove an intermediate point from the pour break	1. Select an intermediate corner point. 2. Press Delete .
Modify the pour break properties	1. Double-click the pour break to open the property pane. 2. Modify the properties. 3. Click Modify .

See also

[Pour breaks \(page 415\)](#)

[Resize and reshape model objects \(page 103\)](#)

Remove a pour break

1. Ensure that the  [Select pour breaks selection switch \(page 122\)](#) is active.
2. Select the pour break.
3. Press **Delete**.

See also

[Pour breaks \(page 415\)](#)

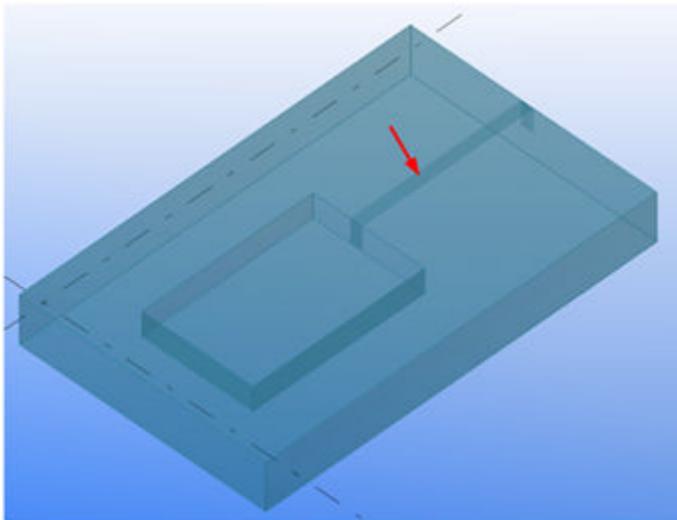
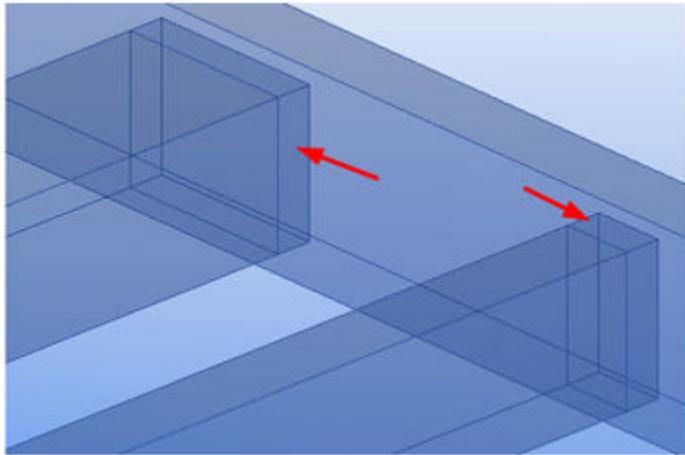
Troubleshoot pours

When you work with cast-in-place concrete parts, it is important that you regularly check the resulting pour objects, and try to get rid of the errors related to them, before you start detailing, or creating drawings and reports. The errors in solid pour objects may lead to inaccuracies in volume and other quantity calculations, and to incorrect representation and hatching in drawings.

While you model, use the following methods to check the model for pour-related errors:

- Check if there are `Solid error` rows in the [session history log file \(page 623\)](#).

- Ensure that the cast-in-place concrete parts and pour objects look continuous in model views. They should not have part outlines or shadow lines inside them, like in the following images:



If you notice errors or overlapping volumes or faces, try remodeling some of the parts.

You can also try out the following tips to avoid pour-related errors:

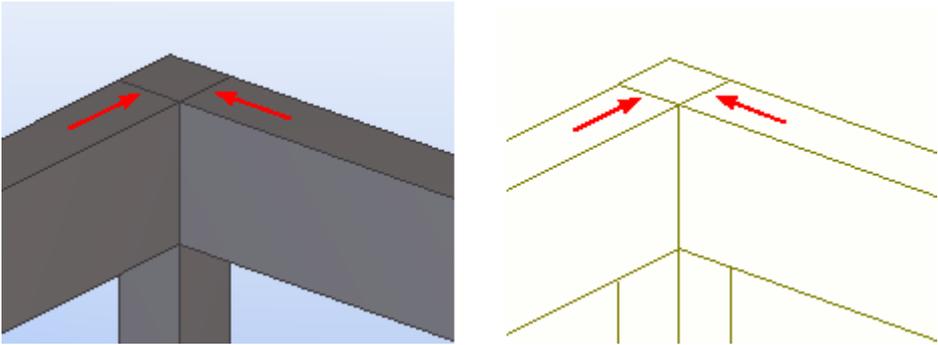
- Make sure that the number of parts included in a single pour object is reasonable.
- Sometimes modeling the parts in a different order may fix errors in pour objects.
- To control which lines are visible in drawings, use the advanced options `XS_DRAW_CAST_PHASE_INTERNAL_LINES` and `XS_DRAW_CAST_UNIT_INTERNAL_LINES`.

This may help because the cast-in-place concrete parts that have errors are treated in the same way in drawings as the precast concrete parts.

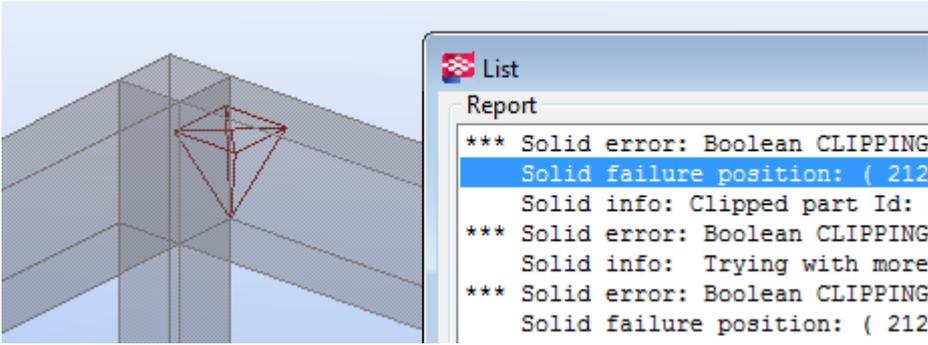
If remodeling the parts does not fix errors in solid pour objects, then overlap the parts as little as possible to ensure that the volume and quantity calculations are close to the correct values.

Example: Identify and fix a pour error

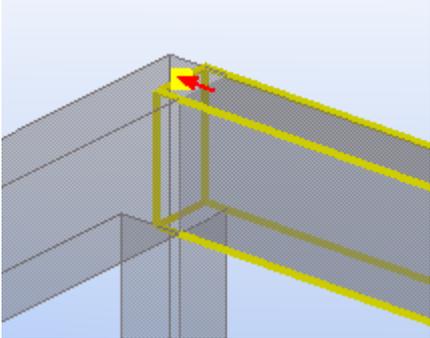
This is how an error related to a solid pour object can be indicated in a model view and in a drawing. The pour object is not shown as continuous, and there are extra lines between the parts in the pour object:



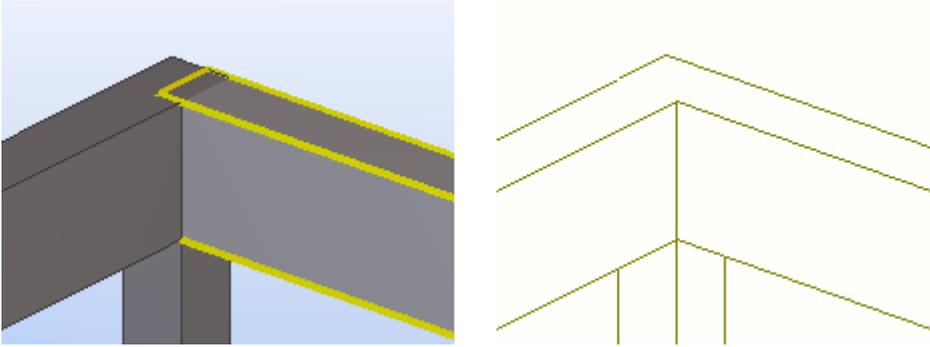
Checking the [session history log file \(page 623\)](#) and clicking a Solid failure position row helps you to locate the error in the model (press **Ctrl+2** to see through the parts):



Try moving a beam end so that it is no longer on the same surface as the column side:



This is how the model and the drawing look like after you have fixed the model:



Also the volume of the pour object is now correct, for example, in reports. The overlapping volume of the beam and the column is only counted once.

See also

[View solid errors \(page 623\)](#)

[Manage pours \(page 402\)](#)

Example: Create concrete geometry and work with pours

The guidelines in this example help you to efficiently model cast-in-place concrete geometries, and to define, visualize, sequence, and report pours and pour breaks.

Before you start, ensure that you have pour management enabled. See [Enable pour management \(page 402\)](#).

1. If possible, use an existing engineering or architectural model or drawing as a basis when you create concrete structures in Tekla Structures.

Import the existing model or drawing as a reference model to your Tekla Structures model.

See [Import a reference model](#) and [Reference models and compatible formats](#).

2. If you are using an IFC model as a reference model:
 - a. Convert the concrete structures you need from the IFC model to native Tekla Structures objects.
See [Convert IFC objects into native Tekla Structures objects](#) and [Example: Convert IFC objects into Tekla Structures objects in one go](#).
 - b. Check the conversion results.
 - c. If needed, modify the converted objects.

For example, you may need to change the profile, material, or cast unit type of the converted objects.

TIP Use **Organizer** for checking and selecting objects.

3. If you are using a different reference model type, or if there are structures that cannot be converted from an IFC model, model the needed concrete structures as cast-in-place concrete parts in Tekla Structures.

You can model by tracing over the reference model.

See [Create parts and modify part properties \(page 235\)](#).

4. For each cast-in-place concrete part, define a pour phase number to divide your Tekla Structures model into pour objects.

For example, use the default pour phase 0 for horizontal structures, like beams and slabs, and the default pour phase 1 for vertical structures, like columns and walls, to separate them to different pour objects.

See [Define the pour phase of a part \(page 407\)](#).

TIP Use selection filters or **Organizer** to efficiently select multiple parts and to modify them all at the same time.

5. View and check the pour objects in a pour view.

See [View cast-in-place concrete structures \(page 404\)](#) and [Pour objects \(page 408\)](#).

6. If needed, modify the pour phases or create pour breaks to fine-tune the pour objects.

For example, create pour breaks to split large slabs into smaller pour objects.

See [Create a pour break \(page 418\)](#) and [Pour breaks \(page 415\)](#).

7. Once you are ready with the concrete geometry and pour objects, you can define pour sequences by entering pour numbers for pour objects, or by using the **Organizer** categories.

See [Modify the properties of a pour object \(page 410\)](#) and Categories in Organizer.

8. Calculate the pour units, and modify them by adding and removing objects if needed.

See [Pour units \(page 410\)](#).

9. You can also define other properties for pour objects and pour units, for example, concrete mixtures, or dates or status of workflow.

See [Modify the properties of a pour unit \(page 413\)](#) and Categories in Organizer.

10. Use **Organizer** to categorize pours. Then you can select them by their sequence and report pour-specific information, such as pour volumes and formwork areas.

See View object properties in Organizer and Example: Organize the model into location and custom categories, and view quantities.

11. If you wish, use **Task manager** to include pour objects and pour units in tasks and to schedule pours. You can then visualize pour status information based on planned and actual dates by using **Project status visualization**.

See Create a task in Task manager and Project status visualization.

12. Create general arrangement drawings for pour units.

Select a pour unit using the  **Select assemblies** switch, create a 3D view of the pour unit, and then create a GA drawing using the 3D view.

This way you can automatically include in the drawing all reinforcement, embeds, and other objects that need to be shown with the pour object.

See Pours in drawings.

2.8 Create reinforcement

Once you have created a model of concrete parts, you will need to reinforce the parts to gain higher strength for the parts.

In Tekla Structures, you can use different methods to create reinforcement. In many cases, you may need to use a combination of several reinforcement tools to get the desired results.

The most automated method is to use the various reinforcement components Tekla Structures contains. We recommend that you use reinforcement components to create reinforcement whenever possible. The components are adaptive, attached to a concrete part, and updated automatically if the dimensions of the reinforced part change.

Rebar sets are another flexible and versatile method to create reinforcement. Also the rebar sets are adaptive to concrete geometry, and easy to modify using direct modification.

In addition to these methods, you can manually create:

- [single reinforcing bars \(page 451\)](#)
- [reinforcing bar groups \(page 452\)](#)

For more automated reinforcing bar group creation you can use [Rebar shape catalog \(page 454\)](#), which contains predefined reinforcement shapes.

- reinforcement meshes

- [prestressed strands \(page 473\)](#)
- [reinforcement splices \(page 476\)](#)

Create a rebar set

Rebar sets are reinforcing bars that you can modify by using direct modification and the rebar set guidelines, leg faces, and local modifiers. You can create rebar sets when you want to flexibly reinforce various areas in concrete parts or pour objects.

You have several options for creating rebar sets: longitudinal bars, crossing bars, planar bars, and bars by point input. Longitudinal, crossing, and planar rebar sets are attached and adaptive to a concrete part or pour object. Using the **Create rebars by point input** command you can create rebar sets even outside concrete objects. You can also use Rebar shape placing tool to create rebar sets.

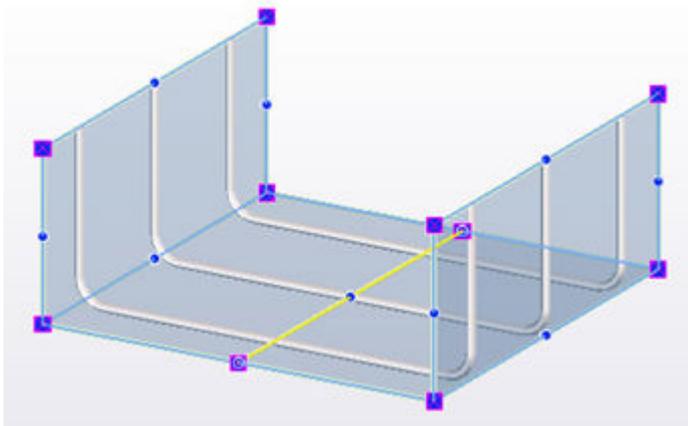
NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

Basic concepts related to rebar sets

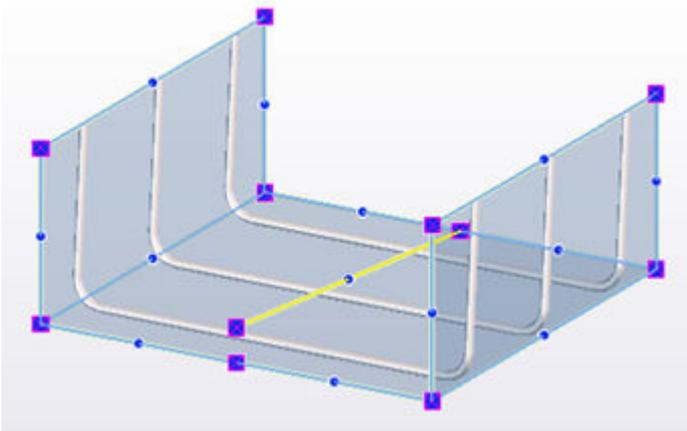
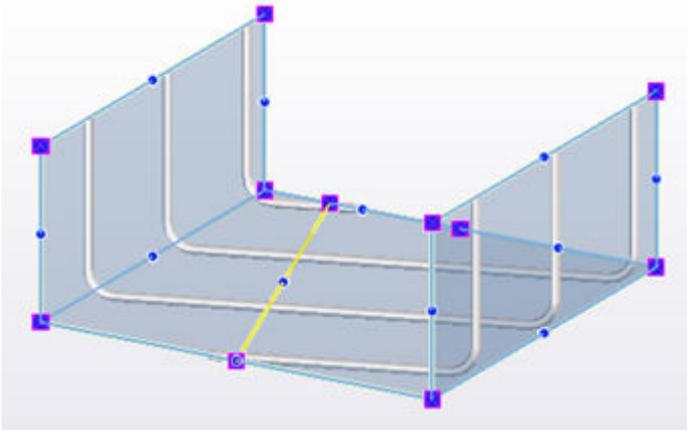
The leg faces of a rebar set are planes that define where the reinforcing bar legs are created. Tekla Structures creates leg faces at the reinforced faces of concrete parts or pour objects, or according to the points you pick when you create rebar sets.

Each rebar set has at least one *guideline* that defines the distribution direction of the bars. The spacing of the bars is also measured along the guideline. The guideline can be a line, or a polyline that may have corner chamfers.

In the example below, the leg faces are shown in gray, and the guideline is highlighted in yellow:

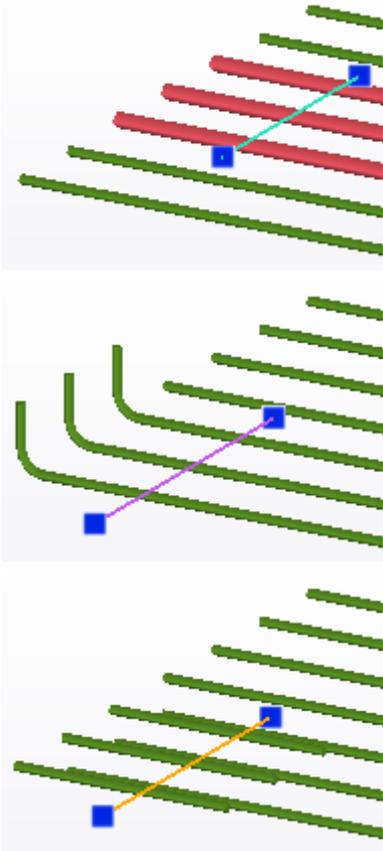


The position of the guideline affects the bar creation. If you move or lift an end of the guideline, the bars turn respectively. For example:



If needed, you can create up to two *secondary guidelines*, and use them to define different spacings along the rebar set. You can also use secondary guidelines when you create longitudinal bars for [curved structures \(page 445\)](#). Tekla Structures automatically creates three guidelines for longitudinal rebar sets in curved beams, polybeams, strip footings, and wall panels.

If you need to modify a rebar set only at certain locations, you can create local *property modifiers*, *end detail modifiers*, and *splitters*.

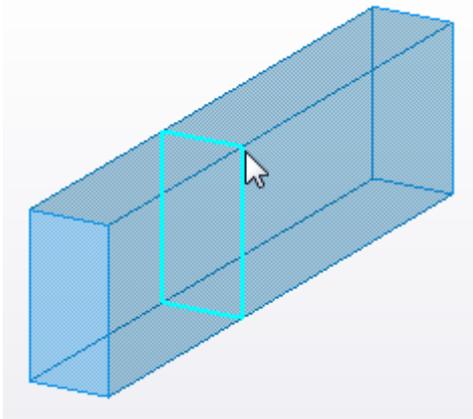
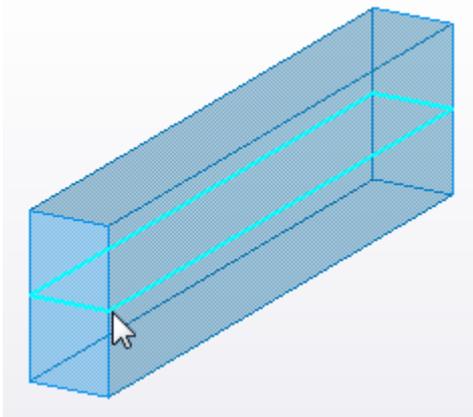


Create longitudinal rebars

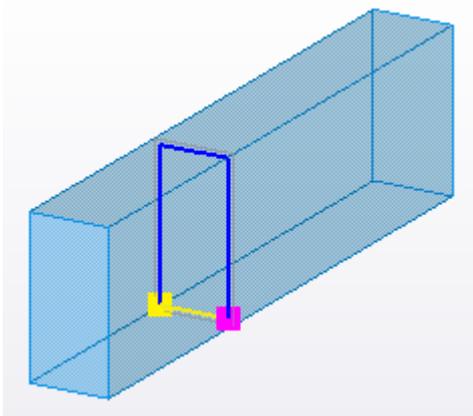
You can create a rebar set that consists of longitudinal reinforcing bars in a concrete part or pour object.

1. Depending on the concrete object you want to reinforce, [use a part view](#) or [a pour view \(page 404\)](#).
2. On the **Concrete** tab, click **Rebar set** --> **Create longitudinal rebars**.
3. Move the mouse pointer over the edges of a concrete part or pour object.

Tekla Structures highlights the cross sections that you can select.



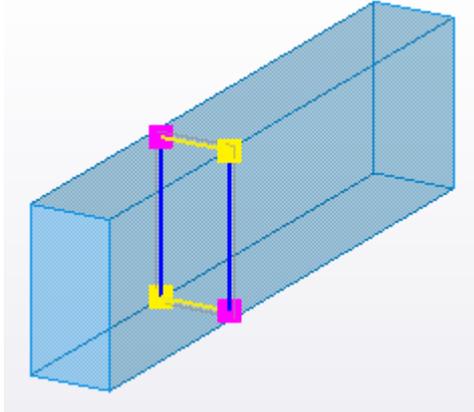
4. Select the cross section that you want to reinforce.



5. If needed, modify the cross section size or shape for the bars.
To do this, click  on the contextual toolbar, and then drag the cross section handles.
6. In the selected cross section, select the faces that you want to reinforce.

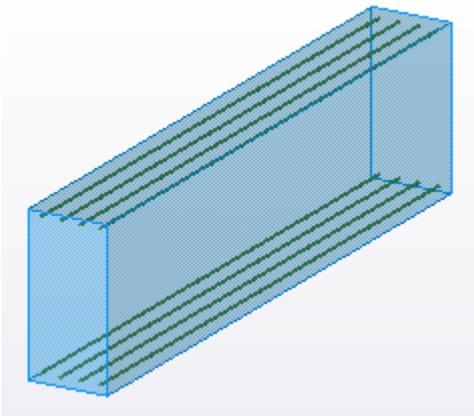
By default, only one of the faces is selected. To select multiple faces, hold down **Shift** or **Ctrl**.

Tekla Structures highlights the selected faces in yellow.



7. To extend or shorten individual faces, click  on the contextual toolbar. Then drag the yellow and magenta end handles.
8. To finish, click the middle mouse button, or click  **Create the rebar set** on the contextual toolbar.

Tekla Structures creates a rebar set at each selected face, and the bars perpendicular to the selected cross section.

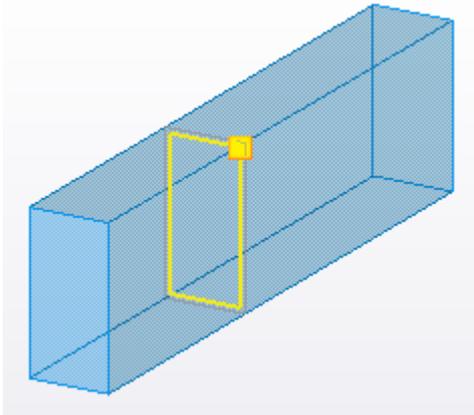


Create crossing rebars

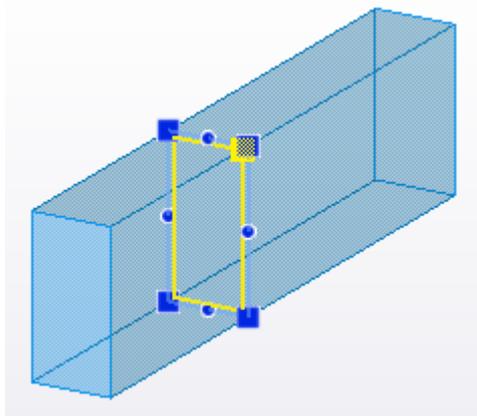
You can create a rebar set that consists of crossing reinforcing bars in a concrete part or pour object.

1. Depending on the concrete object you want to reinforce, [use a part view](#) or [a pour view \(page 404\)](#).
2. On the **Concrete** tab, click **Rebar set** --> **Create crossing rebars**.

3. Move the mouse pointer over the edges of a concrete part or pour object. Tekla Structures highlights the cross sections that you can select.
4. Select the cross section that you want to reinforce.



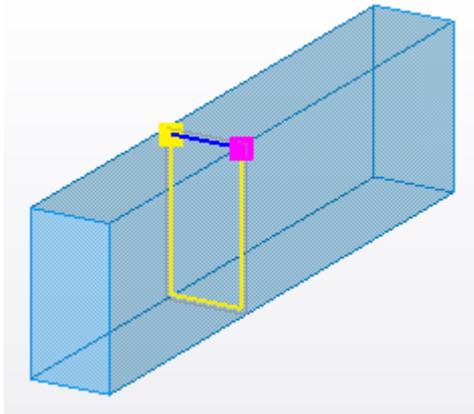
5. If needed, modify the shape of the bars.
 - To extend or shorten individual legs, click  on the contextual toolbar. Then drag the bar end handles. This way you can also create overlapping bar shapes, or extend bar ends outside the concrete object.
 - To change the cross section size for the bars, click  on the contextual toolbar. Then drag the cross section handles.



6. In the selected cross section, select the bar legs that you want to create. By default, all legs are selected and Tekla Structures creates a leg for each object face.
 - To unselect a selected leg, hold down **Ctrl** and click the leg.
 - To unselect all legs, click  on the contextual toolbar.

- To select more than one leg, select the first leg, and then hold down **Ctrl** or **Shift** when selecting the rest of the legs.
- To select all legs, click  on the contextual toolbar.

Tekla Structures highlights the selected legs in yellow, and creates a continuous bar shape of the legs.

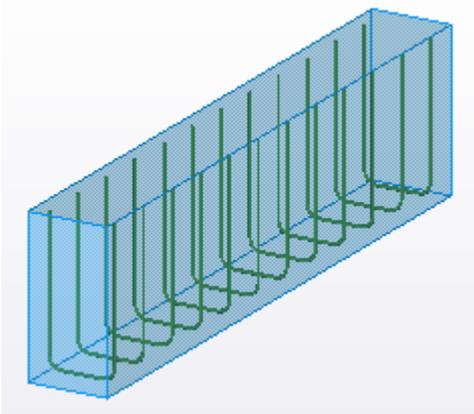


7. If you want to rotate the bar shape, for example, to move stirrup hooks to another corner, press **Tab** for counterclockwise or **Shift+Tab** for clockwise direction.
8. If you want to modify the distribution length of the bars, adjust the length of the rebar set guideline.

Click  on the contextual toolbar and then drag the guideline end handles .

9. To finish, click the middle mouse button, or click  **Create the rebar set** on the contextual toolbar.

Tekla Structures creates the bars parallel to the selected cross section, and distributes the bars along the length of the guideline.



Create planar rebars

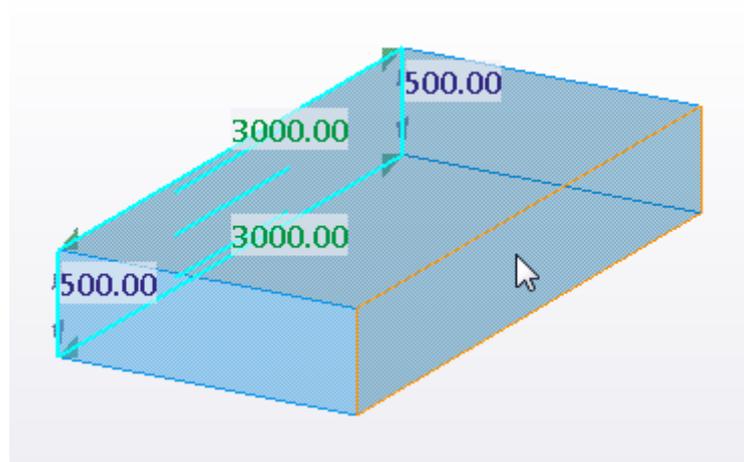
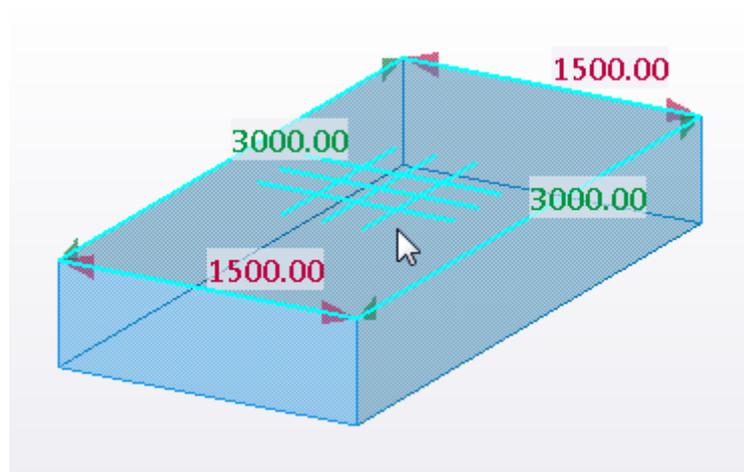
You can create a rebar set that consists of planar reinforcing bars in a concrete part or pour object.

1. Depending on the concrete object you want to reinforce, [use a part view or a pour view \(page 404\)](#).
2. On the **Concrete** tab, click **Rebar set** --> **Create planar rebars**.
3. Define the faces and areas of a concrete object you want to reinforce, and the direction of the bars using the following options on the contextual toolbar:

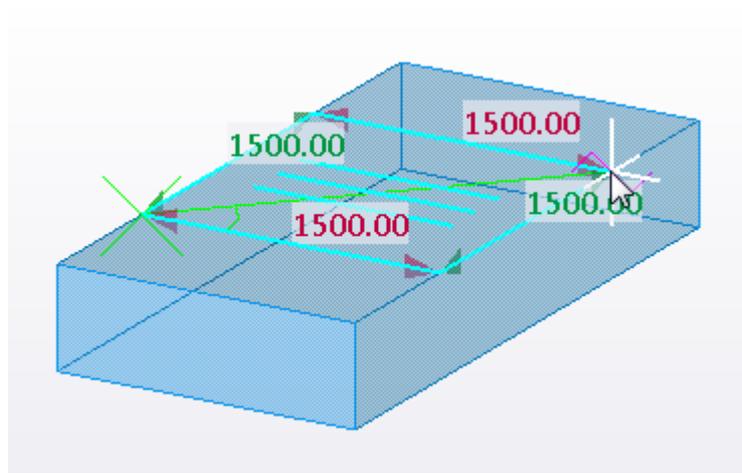
Click this button	To do this
	Create bars at the near face of the concrete object.
	Create bars at the far face of the concrete object.
	Create bars parallel to the longest object face edge.
	Create bars perpendicular to the longest object face edge.
	Create bars in two directions: one set of bars parallel to the longest object face edge, and the other set of bars perpendicular to it.
	Create bars for an entire object face.
	Create bars for a rectangular area on an object face.
	Create bars for a polygonal area on an object face.

4. Depending on the area you selected to reinforce, do one of the following:
 - To reinforce an entire object face:
 - a. Move the mouse pointer over the faces of a concrete part or pour object.

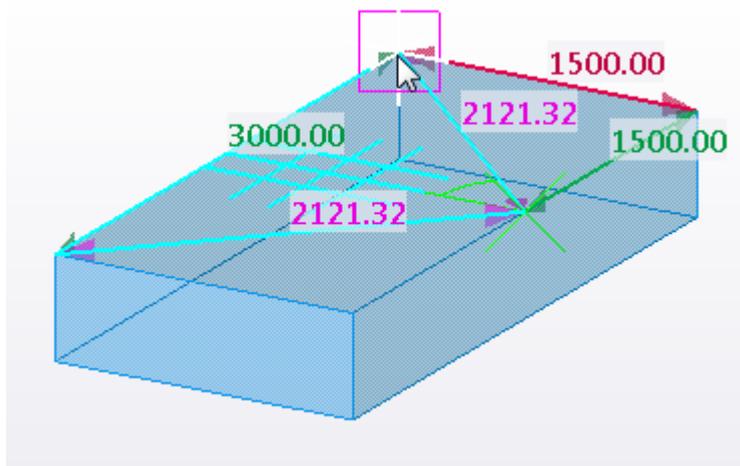
Tekla Structures shows the object face dimensions, and a symbol that indicates the direction of the bars.



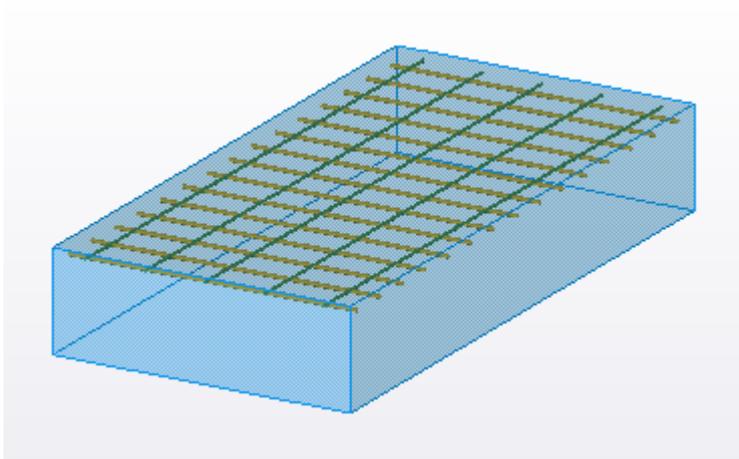
- b. Select the object face.
- To reinforce a rectangular area, pick two opposite corners of the area.



- To reinforce a polygonal area, pick the polygon corners.



Tekla Structures creates the bars according to the options you selected. If you selected to create bars in two directions, Tekla Structures creates two rebar sets: one with bars parallel to the longest object face edge, and one with bars perpendicular to it.



Create rebars by point input

You can create a set of reinforcing bars so that you define the shape of the bars by picking points in the model.

1. On the **Concrete** tab, click **Rebar set** --> **Create rebars by point input**.
2. On the contextual toolbar, select an option to define the rebar set type and the number of cross sections in the rebar set.

The options are:

- **Normal**
- **Tapered**
- **Tapered ridge**
- **Tapered curved**

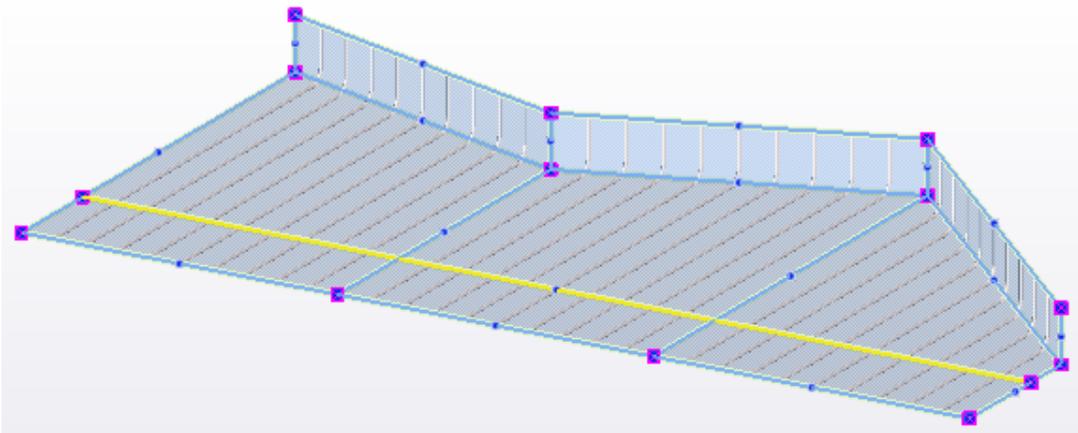
- **Tapered N**

If you select the **Tapered N** option, enter the number of cross sections.



3. Pick points to define the shape of the bar at the first cross section.
You can use different [snapping \(page 77\)](#) methods, such as **Ortho** and temporary reference points.
4. Click the middle mouse button to finish picking.
5. For the second and subsequent cross sections, pick points to define the shape of the bar, and then click the middle mouse button to finish picking at each cross section.

Tekla Structures creates a rebar set with leg faces between each cross section.



Rebar set properties

Use the contextual toolbar or the property pane to view and modify the properties of rebar sets. The file name extension of the properties file is `.rst`.

See also [Rebar set properties \(page 933\)](#) and [Modify a rebar set \(page 478\)](#).

Limitations

- Roundings in bent bar corners are not taken into account in automatic clash avoidance when Tekla Structures creates rebar sets and arranges them to layers.
- You cannot create rebar sets in deformed parts.

Create a rebar set using Rebar shape placing tool

You can create a rebar set by selecting a predefined bar shape from **Rebar shape placing tool**. The predefined shapes in **Rebar shape placing tool** are

based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Use **Rebar shape placing tool** to reinforce parts and pour objects. The rebar sets may extend across single or multiple objects.

Rebar shape placing tool does not work with round, spiral, or 3D bar shapes, or in tapered variable cross sections.

Create rebar sets

1. On the **Concrete** tab, click **Rebar set** --> **Rebar shape placing tool**.

The **Rebar shape placing tool** dialog box opens.

2. If you want to create bars that extend across several parts or pour objects, for example dowel bars, select **Multiple objects** from the list at the bottom of the dialog box.

3. If you want to create several rebar sets in the same cross section, select the **Keep cross section** check box.

4. Select one of the predefined bar shapes from the tree view on the left.

If the shape that you need is not available, or if you want to remove the shapes that you do not need, you can [reorganize the tree view \(page 444\)](#).

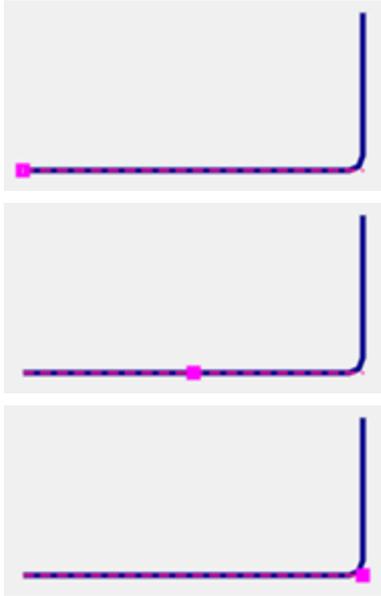
5. Define the bar dimensions.

The dimensions you can define vary depending on the selected bar shape.

The hook properties are visible only if you have set the advanced option `XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION` to `FALSE` in **File menu** --> **Settings** --> **Advanced Options** --> **Concrete Detailing**.

- To set a **Length of leg** value, click a leg in the preview of the shape.
If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete structure.
- To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.

6. Set the rebar set reference point to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.

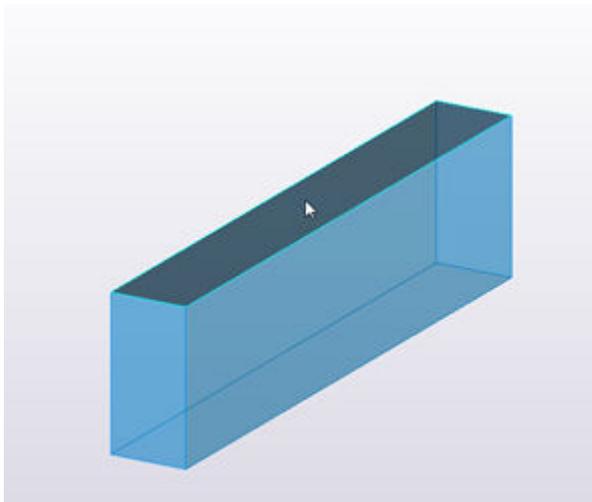
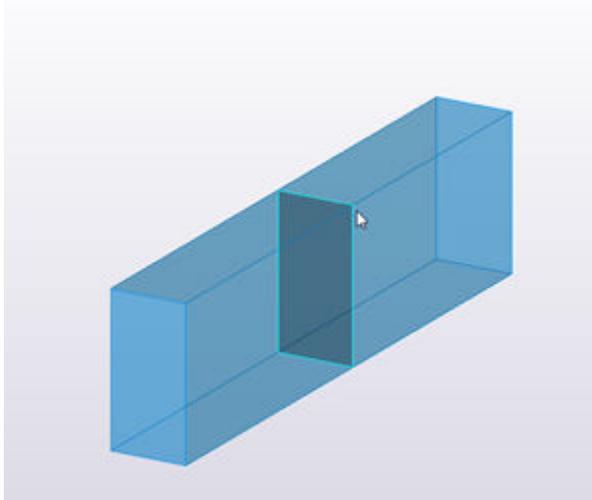


When you are placing the rebar set in the model, you can move the rebar set preview to a new location by dragging the reference point.

7. Modify the other bar properties as needed.
For example, you can use **Layer order number** to arrange bars to layers when two or more rebar sets overlap.
8. On the **Spacing** tab, define the spacing properties of the rebar set.
9. To place the rebar set in the model, move the mouse pointer over the edges and faces of a concrete structure.

Depending on the concrete structure you want to reinforce, [use a part view or a pour view \(page 404\)](#).

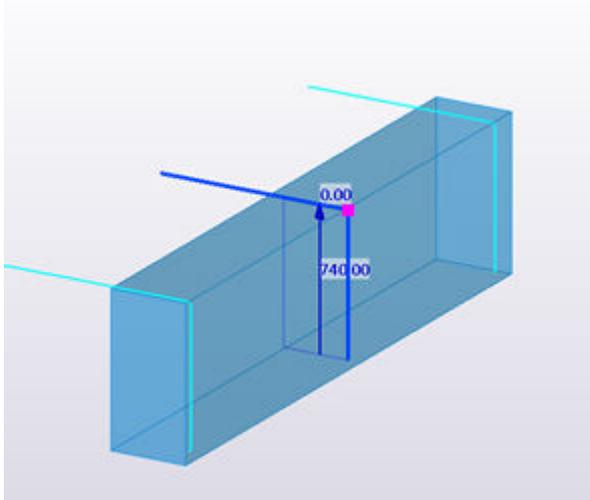
Tekla Structures highlights the cross sections and faces that you can select. For example:



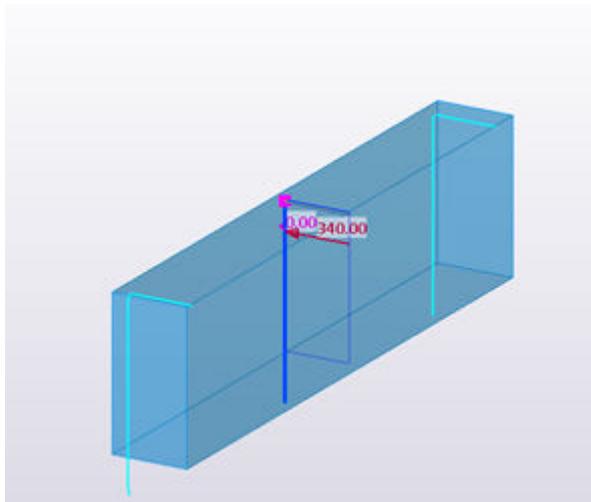
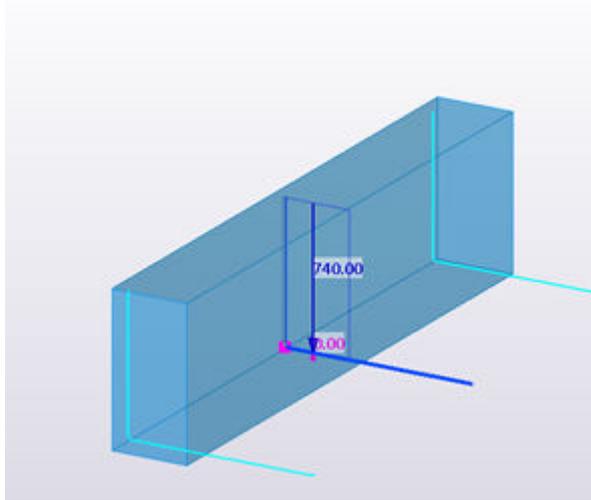
10. Select the cross section or face that you want to reinforce.

If you have **Multiple objects** selected, click each cross section or face to select them. Click the middle mouse button to finish selecting.

Tekla Structures shows a preview of the bar shape in the model, and the first and last bars in the rebar set in cyan.

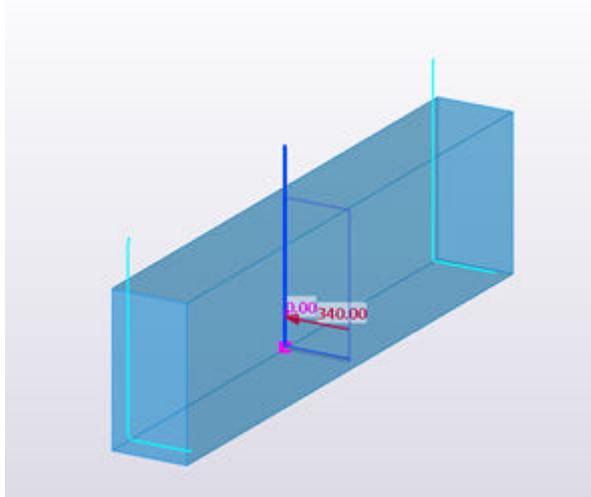


11. To move the rebar set to the desired location in the selected cross section or face, do any of the following:
 - Click a blue line segment to place the rebar set reference point on that line segment. For example:

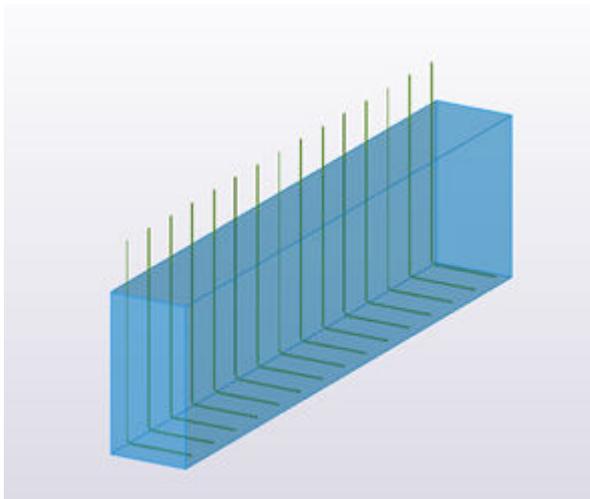


If needed, you can still change the reference point location in the **Rebar shape placing tool** dialog box by double-clicking the desired location in the preview.

- Drag the magenta reference point handle  to a new location on the blue line.
- To turn the bar shape, click  on the contextual toolbar.



12. Click the middle mouse button to create the rebar set.



13. If you have the **Keep cross section** check box selected, repeat steps 4–12 to create more rebar sets in the same cross section.

TIP If the **Rebar shape placing tool** dialog box is already open but the command is not active, click the **Select cross section** button to start creating rebar sets again.

Add and remove rebar shapes

You can modify the tree view in **Rebar shape placing tool** by adding frequently used bar shapes to the tree, or removing the bar shapes that you do not need.

1. On the **Concrete** tab, click **Rebar set** --> **Rebar shape placing tool**.
The **Rebar shape placing tool** dialog box opens.
2. Click **Organize catalog**.

3. To create a new category folder, click .
4. Drag and drop the selected shapes to the folder.
If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.
When the shapes are listed in a report, they all get the same shape code.
5. Change the name of the folder in a similar way, if needed.
6. To remove a shape from a category, select the shape and click .
7. Click **OK**.

Examples: Rebar sets in curved structures

You can reinforce curved concrete structures using rebar sets.

The curved concrete structures can include [curved beams \(page 279\)](#), [polybeams \(page 282\)](#) that have **Arc point** chamfers, and flat [spiral beams \(page 285\)](#) whose total rise is zero. You can also reinforce strip footings and wall panels the same way as you reinforce beams and polybeams.

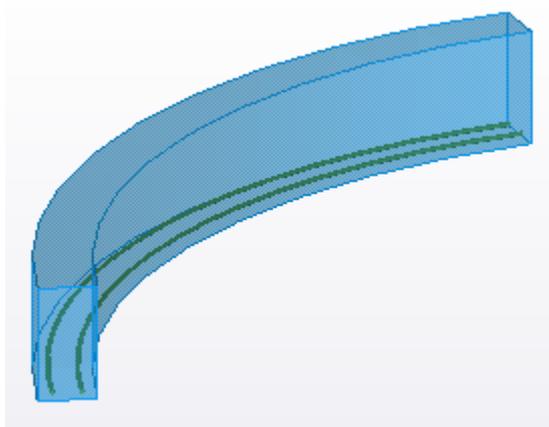
For more information about creating rebar sets, see also [Create a rebar set \(page 428\)](#).

Create longitudinal bars for a curved beam

In this example, we will create longitudinal bottom bars for a curved concrete beam.

1. Create a curved concrete beam.
 - a. On the **Concrete** tab, click **Beam**.
 - b. Pick two points.
 - c. Double-click the beam to modify its properties.
 - d. Set the radius and number of segments, and then click **Modify**.
2. Create longitudinal bars at the bottom face of the beam.
 - a. On the **Concrete** tab, click **Rebar set --> Create longitudinal rebars**.
 - b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
 - c. Click the middle mouse button to create the rebar set.

Tekla Structures creates the curved longitudinal bars according to the beam geometry. For example:



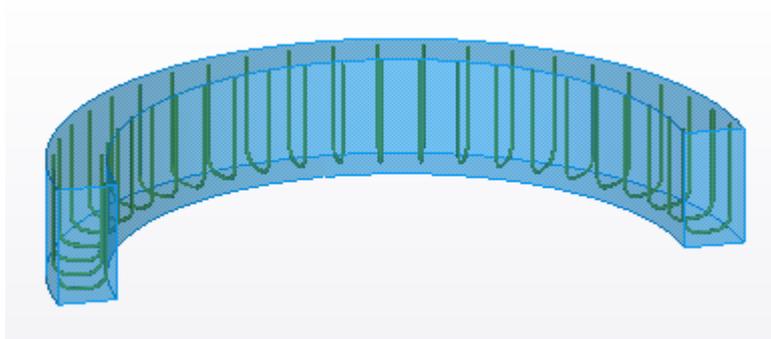
The rebar set has three guidelines: one at each end of the beam and one in the midpoint of the beam.

Create crossing bars for a curved polybeam

In this example, we will create crossing U-shaped stirrups for a curved concrete polybeam.

1. Create a concrete polybeam with curved segments.
 - a. On the **Concrete** tab, click **Beam** --> **Polybeam**.
 - b. Pick at least three points you want the beam to go through, and then click the middle mouse button.
 - c. Select the polybeam.
 - d. Select the handle in a polybeam corner, and then select the  **Arc point** chamfer type on the contextual toolbar.
2. Create crossing bars that follow the bottom and side faces of the beam.
 - a. On the **Concrete** tab, click **Rebar set** --> **Create crossing rebars**.
 - b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
 - c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.
 - d. Click the middle mouse button to create the rebar set.

Tekla Structures creates the crossing bars radially according to the beam geometry. For example:



The rebar set guideline is a polyline with three points, and the midpoint has the **Arc point** chamfer.

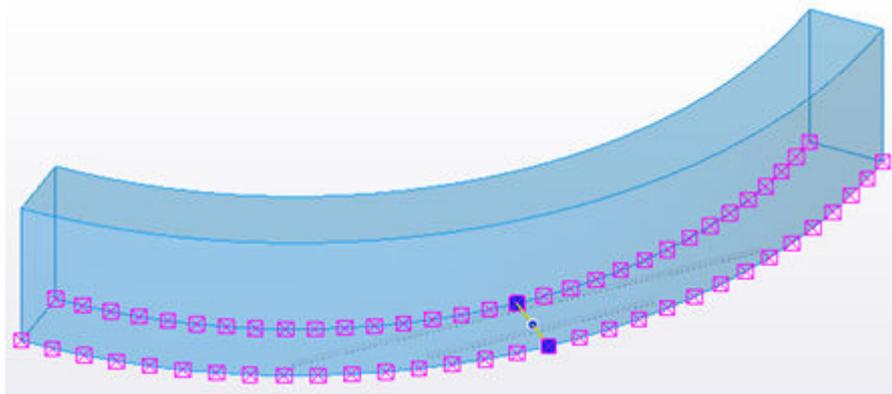
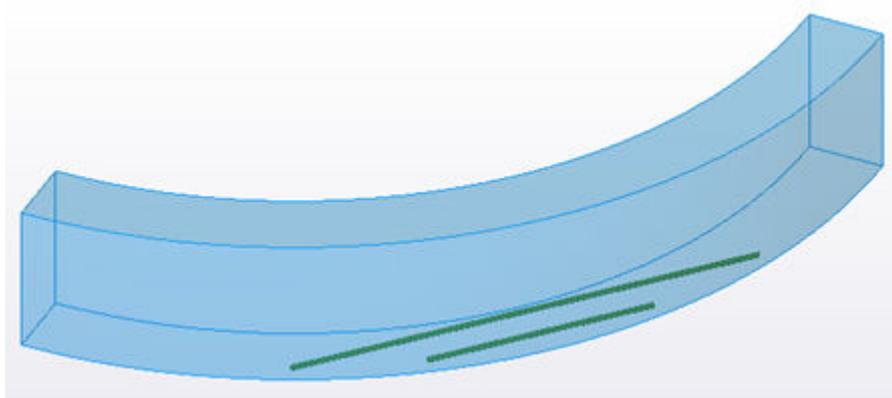
Create reinforcing bars for a spiral beam

In this example, we will reinforce a spiral beam whose total rise is zero. We will create longitudinal bottom bars and crossing U-shaped stirrups.

You can use the following manual method also for more complex concrete objects that have been imported and that cannot be reinforced automatically.

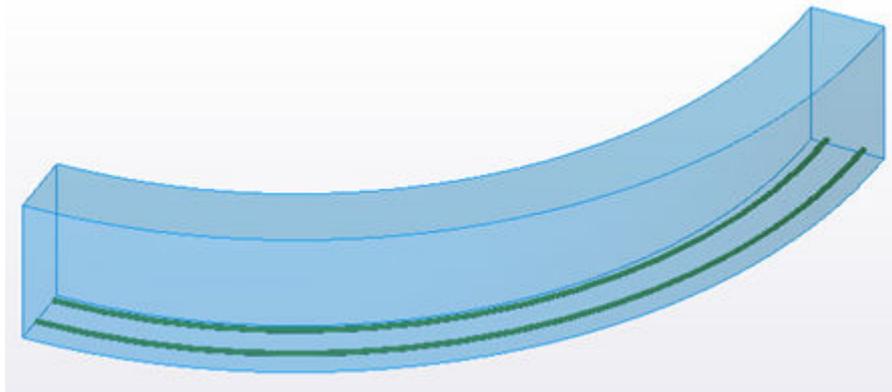
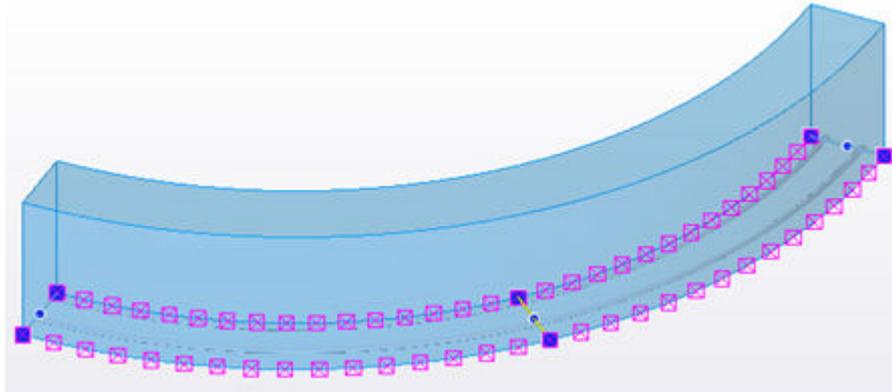
1. Create a flat spiral concrete beam.
 - a. On the **Concrete** tab, click **Beam** --> **Spiral beam**.
 - b. Pick the start point of the beam.
 - c. Pick a point to indicate the beam's center of curvature.
 - d. Click the middle mouse button.
 - e. Ensure that **Total rise** is 0.
2. Create longitudinal bars at the bottom face of the beam.
 - a. On the **Concrete** tab, click **Rebar set** --> **Create longitudinal rebars**.
 - b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
 - c. Click the middle mouse button to create the rebar set.

Tekla Structures creates a longitudinal rebar set with one guideline.



- d. Press **Esc** to interrupt the command.
3. Modify the longitudinal rebar set by creating more guidelines.
 - a. Select the rebar set.
 - b. On the contextual toolbar, click  **Add secondary guideline**.
 - c. Ensure that you are in the single point picking mode ( shown on the contextual toolbar).
 - d. Pick the start point for a secondary guideline.
 - e. Pick the start point for another secondary guideline.
 - f. Press **Esc** to finish creating secondary guidelines.
 - g. If needed, move the guidelines to desired locations by dragging them or their end point handles.

For example, you can move the primary guideline to the midpoint of the beam, one of the secondary guidelines to the start of the beam, and the other secondary guideline to the end of the beam.

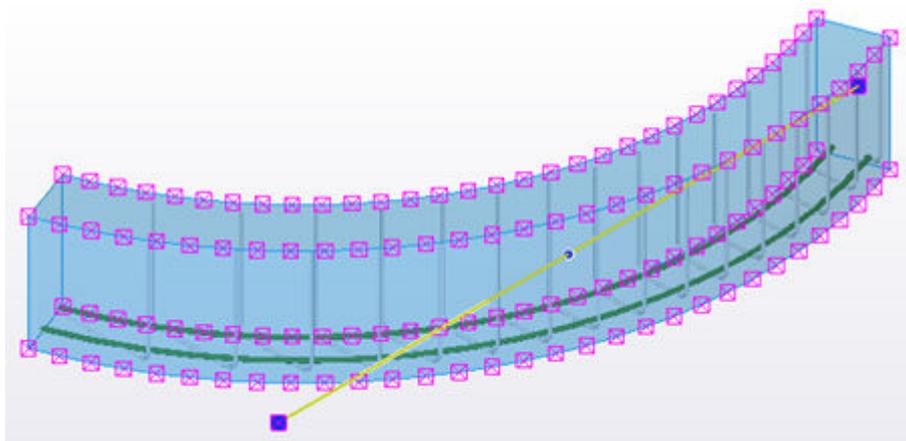
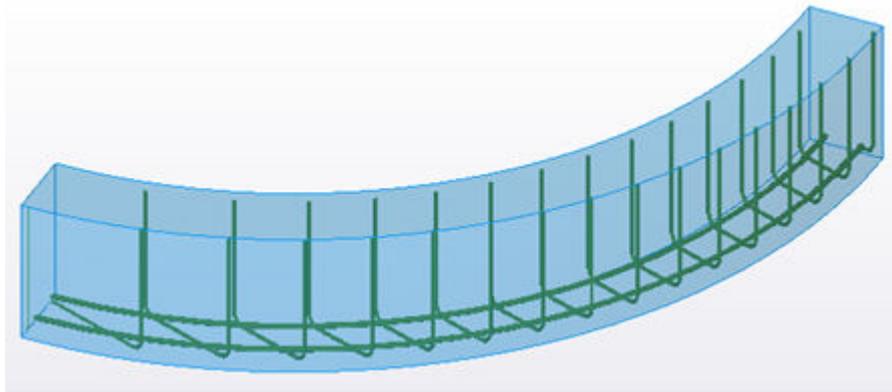


- h. Modify the geometry and [properties \(page 936\)](#) of the guidelines as needed.

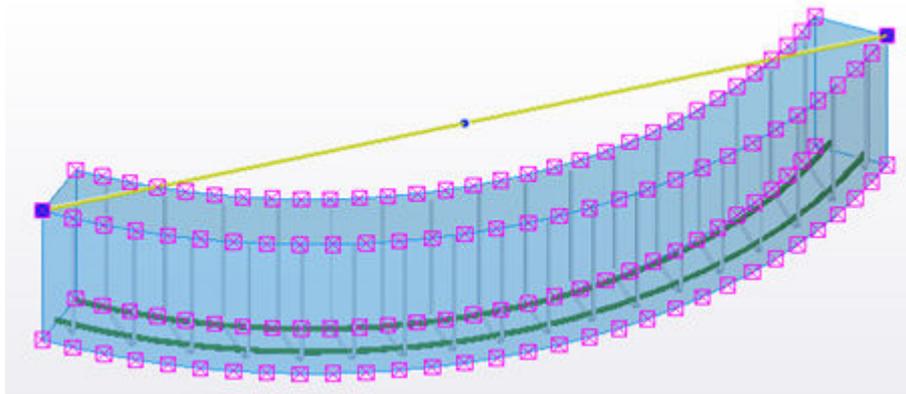
The bars are constructed according to the locations and spacing settings of these three guidelines.

- 4. Create crossing bars that follow the bottom and side faces of the beam.
 - a. On the **Concrete** tab, click **Rebar set** --> **Create crossing rebars**.
 - b. Move the mouse pointer over the edges of the beam, and select the cross section that you want to reinforce.
 - c. In the selected cross section, hold down **Ctrl** and click the top bar leg to unselect it.
 - d. Click the middle mouse button to create the rebar set.

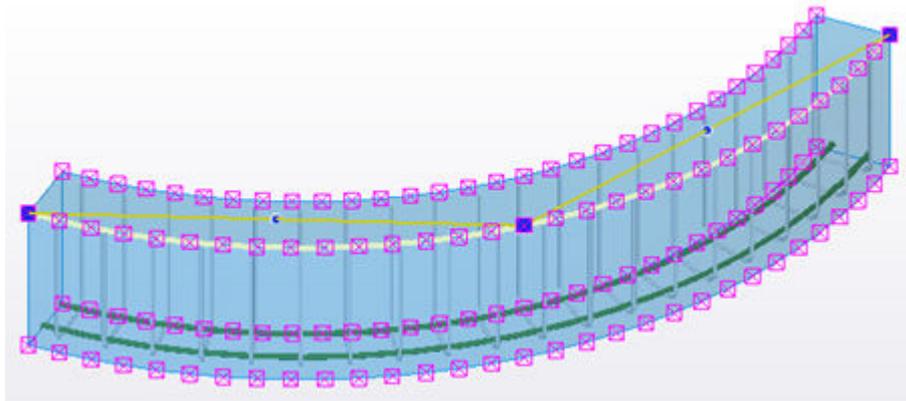
Tekla Structures creates a crossing rebar set with one guideline.



- e. Press **Esc** to interrupt the command.
- 5. Modify the crossing rebar set by modifying the guideline.
 - a. Select the rebar set to highlight the guideline.
 - b. Drag the guideline end points  to the beam ends.

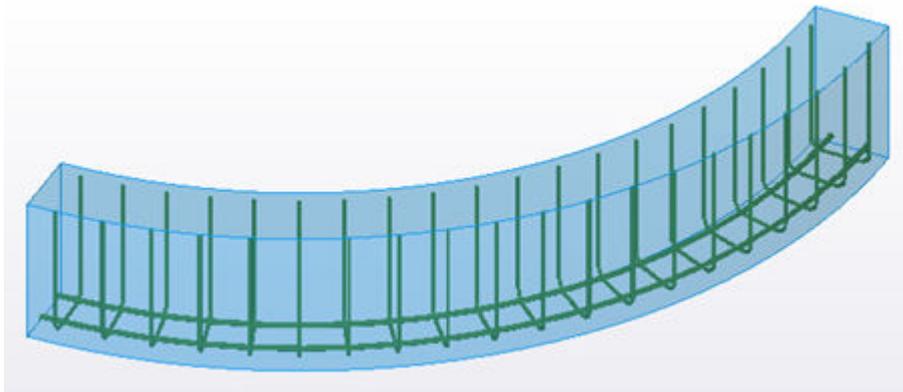


- c. Drag the guideline midpoint  to the beam midpoint.



- d. Ensure that the new guideline corner has the  **Arc point** chamfer.

Tekla Structures arranges the crossing bars radially along the beam.



Limitations

- If curved longitudinal bars have too small start and/or end offset values, the bars closest to leg face edges may be divided into small bar segments. To avoid this, increase the offset values.

Create a single reinforcing bar

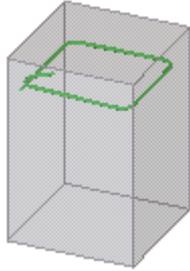
1. On the **Concrete** tab, click **Rebar** and select **Bar**.



If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Bar** command to open the **Single rebar** properties.

2. Select the part to reinforce.
3. Pick the bar start point.
4. Pick the other bar reference points to set the bar shape.
5. Click the middle mouse button to finish picking.

Tekla Structures attaches the bar to this part.



6. If you want to modify the reinforcement, do one of the following:
 - Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
 - Double-click the reinforcement to open the **Single rebar** properties and modify the [properties \(page 927\)](#).

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

Create a reinforcing bar group

A reinforcing bar group includes several identical, or very similar, reinforcing bars. Tekla Structures always treats these bars as a group, modifies them in the same way, deletes them all at the same time, and so on. You first define the shape of a single bar, then the direction in which Tekla Structures distributes the bars.

NOTE If you do not want to manually define the bar shape, use [Rebar shape catalog \(page 454\)](#) and its predefined reinforcement shapes instead.

1. On the **Concrete** tab, click:



If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Bar group** command to open the **Rebar group** properties.

2. Select the part to reinforce.

Tekla Structures attaches the bar group to this part.

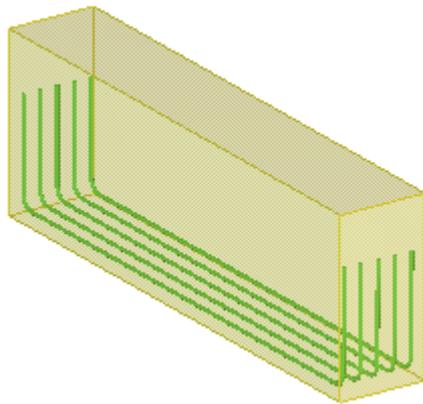
3. Pick the bar start point.
4. Pick the other bar reference points.

These points define the plane of the first bar and the shape of a single bar in the group.

5. Click the middle mouse button to finish picking.
6. Pick the start point of the bar group.
7. Pick the end point of the bar group.

The start and end points indicate the distribution length and direction of the bars. Usually the distribution length of the bars is perpendicular to the plane so that the cover thickness on the sides can be defined.





8. If you want to modify the reinforcement, do one of the following:
- Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
 - Double-click the reinforcement to open the **Rebar group** properties and modify the [properties \(page 927\)](#).

See also

[Create a curved reinforcing bar group \(page 461\)](#)

[Create a circular reinforcing bar group \(page 463\)](#)

[Create a tapered or spiral reinforcing bar group \(page 465\)](#)

Create a reinforcing bar group using Rebar shape catalog

A reinforcing bar group includes several identical, or very similar, reinforcing bars. You can create a reinforcing bar group by selecting a predefined reinforcement shape from **Rebar shape catalog**. The predefined shapes in **Rebar shape catalog** are based on the shapes that have been defined in **Rebar shape manager** and saved in the `RebarShapeRules.xml` file.

Rebar shape catalog does not work with [tapered reinforcing bar groups \(page 465\)](#) or with 3D bar shapes.

NOTE If you do not want to use the predefined shapes but want to manually define the bar shape, use the [Bar group \(page 452\)](#) command instead.

1. On the **Concrete** tab, click **Rebar** and select **Rebar shape catalog**.



The **Rebar Shape Catalog** dialog box opens.

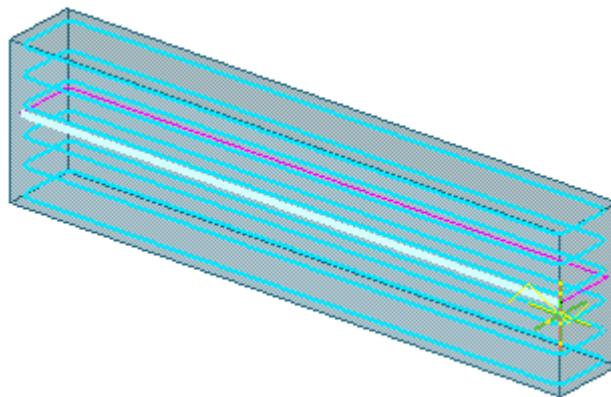
2. Select one of the predefined shapes from the tree view on the left.
You can [add frequently used shapes to the tree view \(page 456\)](#), or delete the shapes that you do not need.

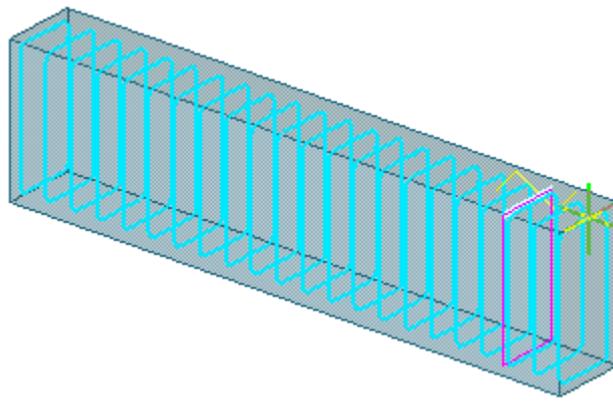
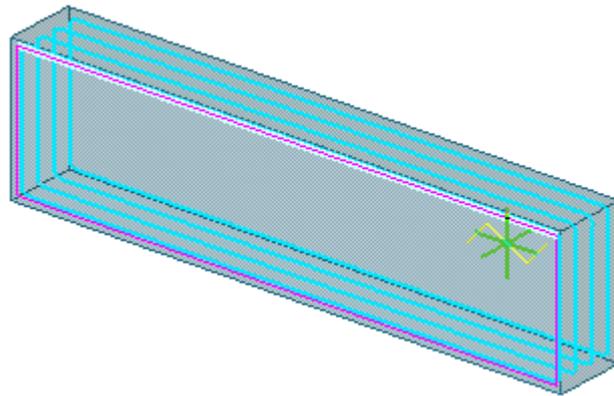
If you select an existing reinforcement in the model and click the **Get** button, the properties of that reinforcement are displayed in the **Rebar Shape Catalog** dialog box.

3. If needed, modify the bar properties.
 - To set a **Length of leg** value, click a leg in the preview of the shape.
If you do not enter a **Length of leg** value, the leg length is calculated automatically according to the dimensions of the concrete part.
 - To set a **Bending angle** value for a bend that is not 90 degrees, click one of the legs next to the bend.
 - For circular, polygonal and spiral reinforcement, you can enter **Circle diameter** and **Overlap distance** values.

The hook properties are visible only if you have set the advanced option XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION to `FALSE` in **File menu --> Settings --> Advanced Options --> Concrete Detailing**.

4. If needed, [set the reference point of the reinforcement \(page 457\)](#) to start, middle, or end by double-clicking the different legs or hooks in the preview of the shape.
5. Click **OK**.
6. In the model, place the mouse pointer over a part face or edge.
A preview showing the placing and dimensions of the reinforcement is displayed.





7. Based on the preview, select a placing for the reinforcing bar group and click the left mouse button.

Tekla Structures creates the reinforcement.

8. If you want to modify the reinforcement, do one of the following:

- Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
- Double-click the reinforcement to open the reinforcing bar group properties, and modify the [properties \(page 927\)](#).

See also

[Create a reinforcing bar group \(page 452\)](#)

[Create a rebar set using Rebar shape placing tool \(page 438\)](#)

Add more reinforcement shapes to the tree view in Rebar shape catalog

You can modify the tree view in **Rebar shape catalog** by adding frequently used shapes to the tree, or deleting the shapes that you do not need.

1. On the **Concrete** tab, click **Rebar** and select **Rebar shape catalog**.



The **Rebar Shape Catalog** dialog box opens.

2. Click **Organize catalog**.

3. Create a new category folder by clicking .

4. Drag and drop the selected shapes to the folder.

If multiple shapes have the same shape code and you drag them to the categories, the shape codes get a suffix **(1)**, **(2)**, and so on. You can rename the shapes as you wish by clicking the name twice and entering a new name or suffix, for example, **(a)**, **(b)**.

When the shapes are listed in a report, they all get the same shape code.

5. Change the name of the folder in a similar way, if needed.

6. To remove a shape from a category, select the shape and click .

7. Click **OK**.

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

Set the reinforcement reference point in Rebar shape catalog

When you use **Rebar shape catalog** and select a shape, you can set the reference point to the start, middle or end of the reinforcing bar leg. When you create the reinforcement in the model, you can move the reinforcement to a new location by dragging the reference point. This is useful, for example, when the reinforcing bar legs are of certain length and you want to aim the reference point, for example, to the middle of a part edge. You can also move the reference point of circular reinforcement shapes.

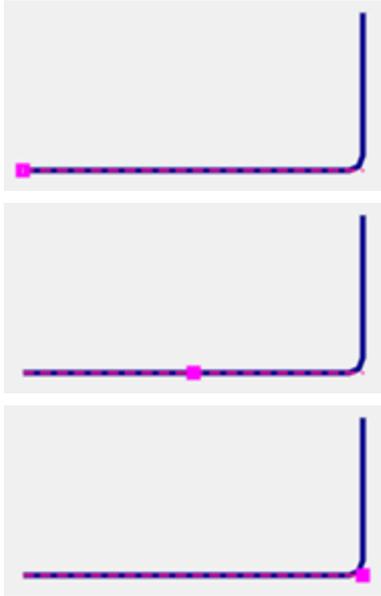
1. On the **Concrete** tab, click **Rebar** and select **Rebar shape catalog**.



The **Rebar Shape Catalog** dialog box opens.

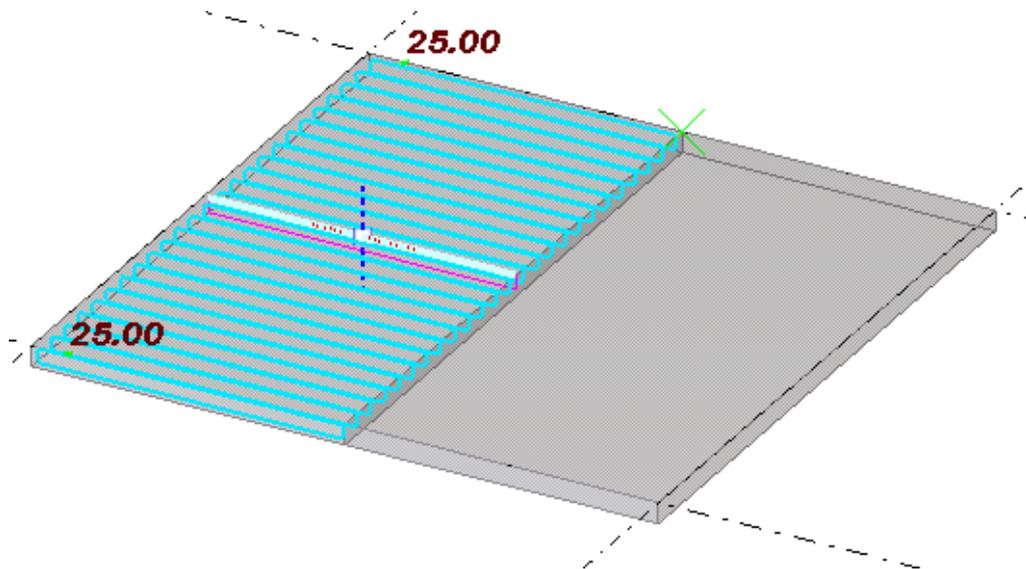
2. Select a reinforcement shape.

3. Set the reference point to the desired location (start, middle, end) by double-clicking the position in the preview of the shape.

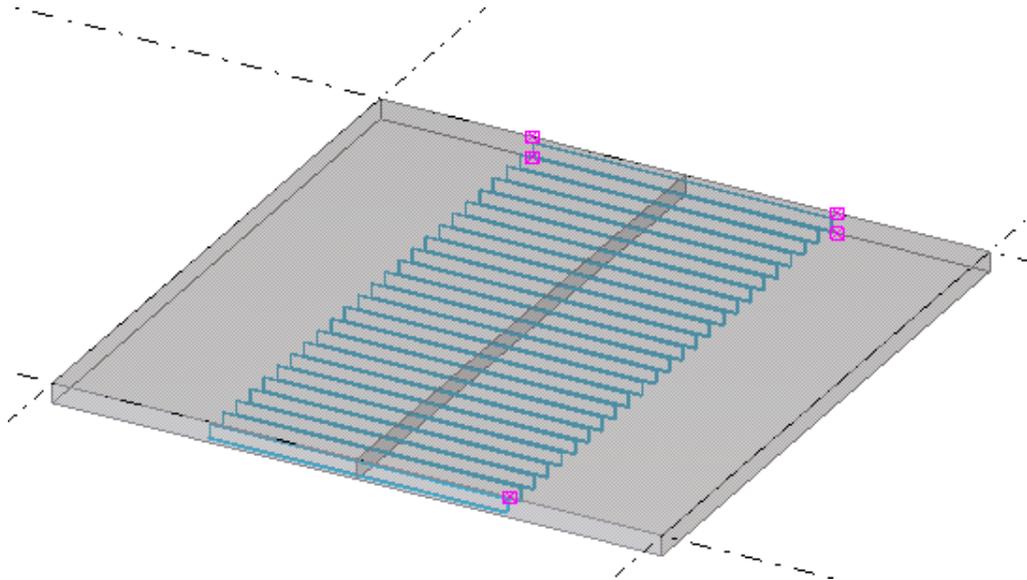


4. If needed, modify the bar properties.
5. Click **Apply** or **OK**.
6. In the model, place the mouse pointer over a part face or edge.
7. Based on the preview, select the desired placing and hold down the **Alt** key and click the left mouse button.

The reference point is displayed.



8. Move the reinforcement to a new location by dragging the reference point.
9. Click the middle mouse button to create the reinforcement.



NOTE For circular reinforcement you can set the reference point to the center line as follows:

- a. Place the mouse pointer over a column edge to have the reinforcement oriented correctly.
 - b. Hold down the **Alt** key and click the left mouse button.
 - c. Drag the reference point and hold down **Shift** key to snap to the center of column.
 - d. Click the middle mouse button to create the reinforcement.
-

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

Reinforce pour objects using Rebar shape catalog

You can reinforce pour objects in pour views using **Rebar shape catalog**.

NOTE [Rebar sets \(page 428\)](#) and **Rebar shape catalog** are the methods to reinforce pour objects in pour views. If you want to use other reinforcement commands, such as [Bar group \(page 452\)](#), or reinforcement components, you need to reinforce single parts in part views. All reinforcement are visible both in part views and in pour views.

When you reinforce pour objects using **Rebar shape catalog**:

- The reinforcement is attached to the reinforced part, not to the pour object.

- The reinforcement geometry is defined in accordance with the pour object geometry even though the reinforcement is attached to a part. For example, pour breaks can limit the length of reinforcing bars.
- In reports the reinforcement information is listed according to the part, not to the pour object.

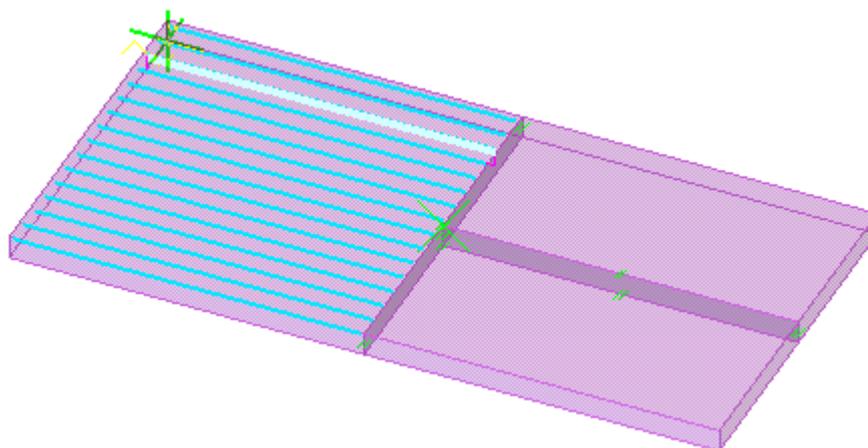
Before you start, create concrete parts whose cast unit type is **Cast in place**. Tekla Structures automatically forms pour objects of them.

1. Ensure that you are using a pour view. If not, click **Pour view** on the **Concrete** tab.
2. If needed, create pour breaks by selecting any of the **Pour break** commands on the **Concrete** tab:
 - **Single point**
 - **Two points**
 - **Multiple points**
3. To insert a reinforcement to a pour object, on the **Concrete** tab click **Rebar** and select **Rebar shape catalog**.

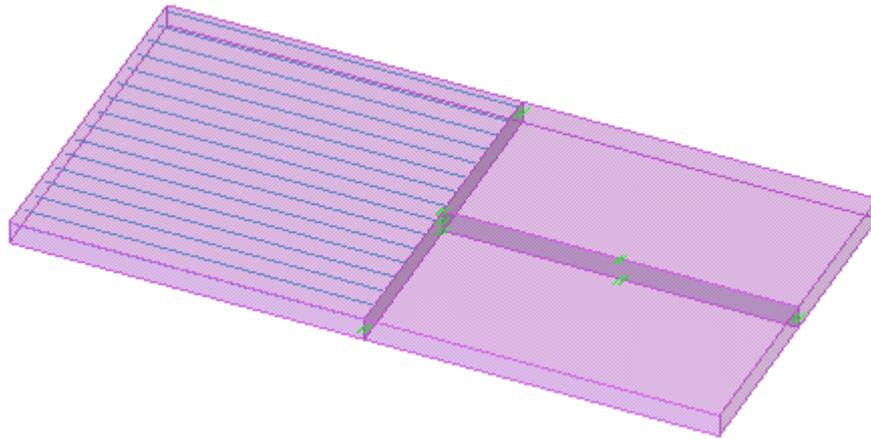


The **Rebar Shape Catalog** dialog box opens.

4. Select a shape from the tree view on the left and modify the properties, if needed.
5. Click **OK**.
6. In the model, place the mouse pointer over a face or an edge of a pour object.



7. Based on the preview, select a placing for the reinforcement and click the left mouse button to create the reinforcement.



See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Manage pours \(page 402\)](#)

Create a curved reinforcing bar group

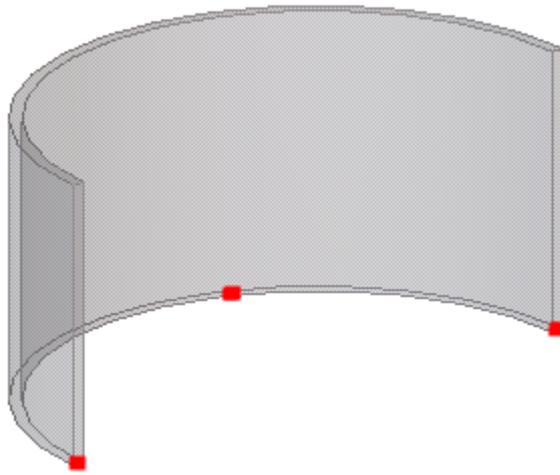
You can reinforce curved segments in a concrete beam or a curved wall.

1. On the **Concrete** tab, click **Rebar** and select **Curved bar group**.

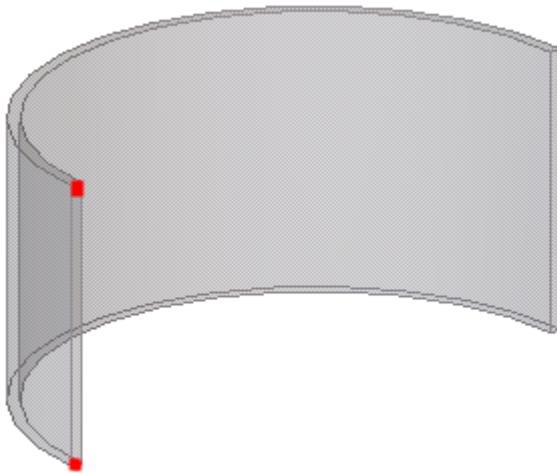


If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Curved bar group** command to open the **Curved bar** properties.

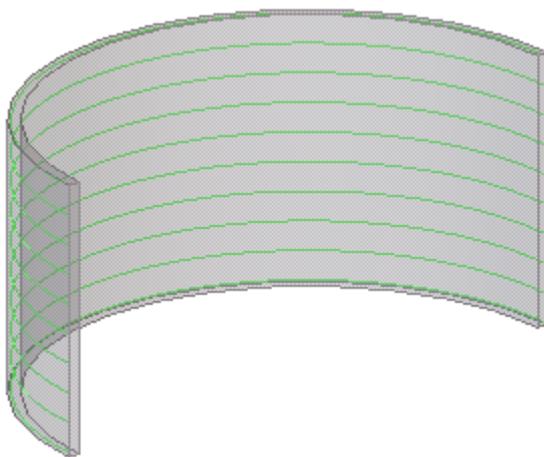
2. Select the part to reinforce.
Tekla Structures attaches the bar group to this part.
3. Pick three points on an arc to define the curve.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of curved reinforcing bars.



5. If you want to change the curved reinforcing bar group properties:
 - a. Double-click the curved reinforcing bar group to open the **Curved bar** properties.
 - b. Modify the [properties \(page 927\)](#).
 - c. Click **Modify**.

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a circular reinforcing bar group \(page 463\)](#)

[Create a tapered or spiral reinforcing bar group \(page 465\)](#)

[Modify reinforcement \(page 477\)](#)

Create a circular reinforcing bar group

You can reinforce round circular columns.

1. On the **Concrete** tab, click **Rebar** and select **Circular bar group**.



If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Circular bar group** command to open the **Circular rebar** properties.

2. Select the part to reinforce.
Tekla Structures attaches the bar group to this part.
3. Pick three points on the outer contour of the concrete part to define the circular bars.

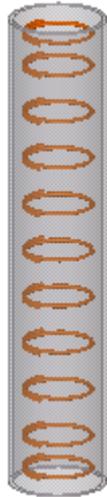
The radius is automatically calculated from these three points.



4. Pick two points to indicate the distribution direction of the bars.



Tekla Structures creates a group of circular reinforcing bars.



NOTE If you want to modify the splice length of the round stirrups, enter negative values in the **Start** and **End** boxes in the **Circular rebar** properties.

5. If you want to change the circular reinforcing bar group properties:
 - a. Double-click the circular reinforcing bar group to open the **Circular rebar** properties.
 - b. Modify the [properties \(page 927\)](#).
 - c. Click **Modify**.

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a curved reinforcing bar group \(page 461\)](#)

[Create a tapered or spiral reinforcing bar group \(page 465\)](#)

[Modify reinforcement \(page 477\)](#)

Create a tapered or spiral reinforcing bar group

For rectangular concrete parts it is sufficient to pick two points to define the distribution area of the reinforcing bar group. If the part shape is not rectangular, an alternative shape can be selected.

Use the **Rebar group type** list on the **Group** tab in the **Reinforcing Bar Properties** dialog box to select and modify the reinforcing group types.

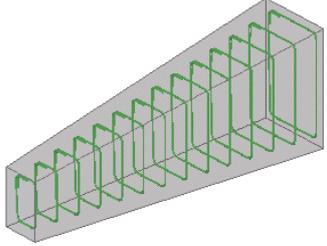
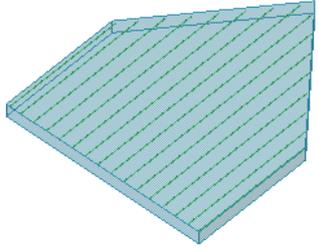
1. On the **Concrete** tab, hold down **Shift** and click:

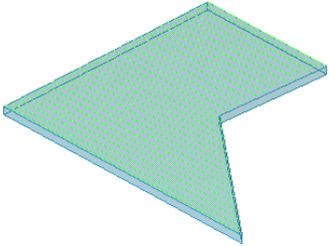
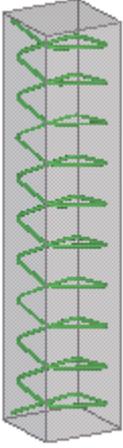


The **Reinforcing Bar Properties** dialog box opens.

2. If needed, enter or modify the bar properties.
3. On the **Group** tab, select a tapered or spiral option from the **Bar group type** list.
4. Click **OK**.
5. Select the part to reinforce.
Tekla Structures attaches the bar group to the part.
6. Pick points to define the shape of the bar at the first cross section.
7. Click the middle mouse button to finish picking.
8. For the second and subsequent cross sections, pick points to define the shape of the bar.
9. Click the middle mouse button to finish picking.
Tekla Structures creates the reinforcement.

Reinforcing bar group types

Option	Description	Example
 Normal	Not tapered. Pick two points to define the distribution area of the bar group.	
 Tapered	One bar dimension changes linearly in the group.	
 Tapered ridge	One bar dimension changes linearly in the group. The dimension is longest in the middle of the group.	

Option	Description	Example
 Tapered curved	One bar dimension changes along a curve. The dimension is longest in the middle of the group.	
 Tapered N	One bar dimension changes linearly between N cross sections. Enter the number of cross sections in the Number of cross sections box.	
 Spiral	The reinforcing bars rise in a polygonal or circular shape along the longitudinal axis of the part.	

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Reinforcing bar and bar group properties \(page 927\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

Create a reinforcement mesh

You can create a reinforcement mesh that consists of two perpendicular bar groups. Tekla Structures treats mesh bars as one unit but distinguishes the main and crossing bars.

The reinforcement mesh can be rectangular, polygonal, or bent. You can also create a customized reinforcement mesh.

NOTE You cannot change the mesh type once the mesh has been created.

Create a rectangular reinforcement mesh

1. On the **Concrete** tab, hold down **Shift** and click **Rebar** --> **Mesh**.

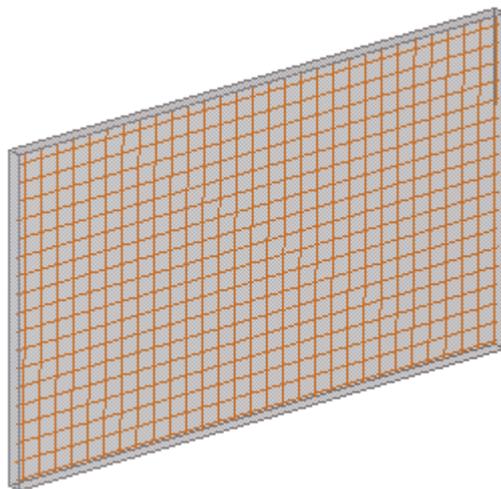


The **Rebar mesh** properties open in the property pane.

2. In the **Mesh type** list, select **Rectangle**.

NOTE You cannot change the mesh type once the mesh has been created.

3. Select the part to reinforce.
Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick a point to indicate the direction of the longitudinal bars.
6. Click the middle mouse button to finish picking.
Tekla Structures creates the mesh parallel to the work plane, to the left of the points you picked.



7. If you want to modify the reinforcement mesh, do one of the following:

- Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
- Double-click the reinforcement to open the **Rebar mesh** properties and modify the [properties \(page 929\)](#).

Create a polygonal reinforcement mesh

1. On the **Concrete** tab, hold down **Shift** and click **Rebar --> Mesh**.



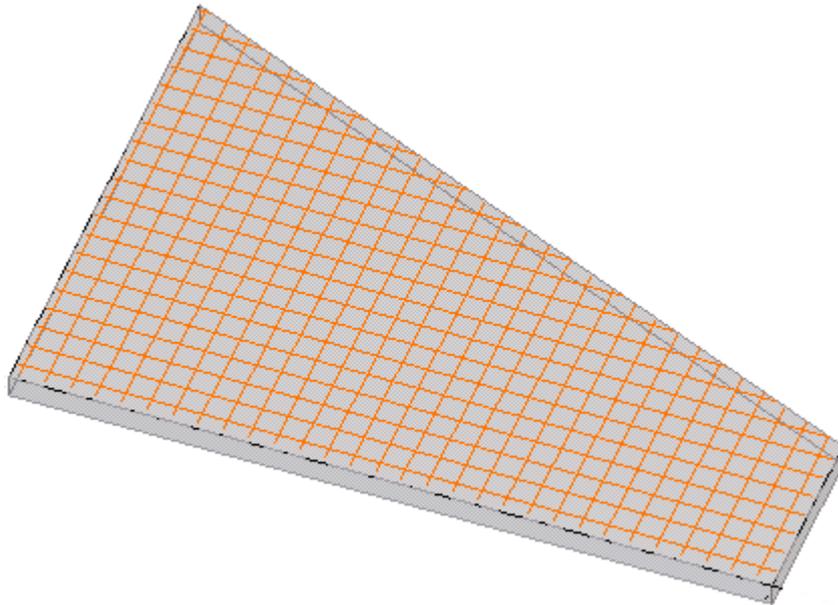
The **Rebar mesh** properties open in the property pane.

2. In the **Mesh type** list, select **Polygon**.

NOTE You cannot change the mesh type once the mesh has been created.

3. Select the part to reinforce.
Tekla Structures attaches the mesh to this part.
4. Pick the start point of the mesh.
5. Pick the corner points of the mesh.
6. Click the middle mouse button to finish picking.
7. Pick a point to indicate the direction of the longitudinal bars.

Tekla Structures creates the mesh.



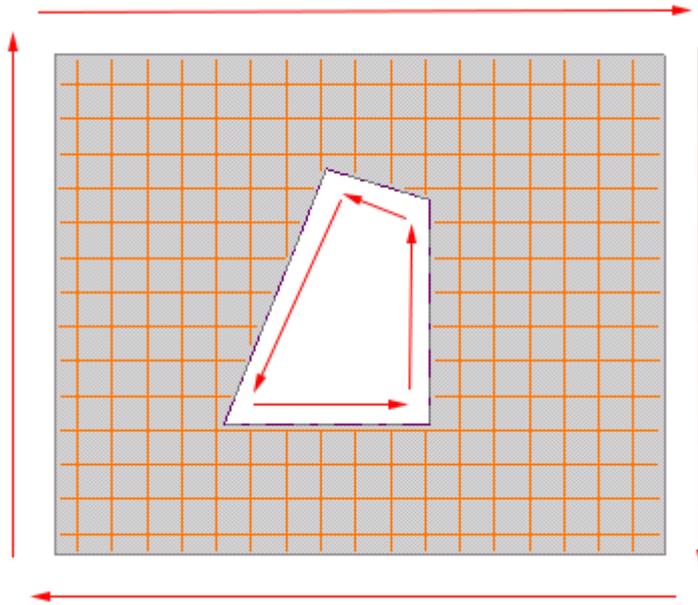
8. If you want to modify the reinforcement, do one of the following:
 - Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
 - Double-click the reinforcement to open the **Rebar mesh** properties and modify the [properties \(page 929\)](#).

Reinforcement mesh with holes

If you want to reinforce a part that has holes, you need to pick the corner points of holes when you create the reinforcement.

1. Select the part to reinforce.
2. Pick the start point of the mesh.
3. Pick the corner points of the mesh.
4. Pick the corner points of the hole.

Note that you need to pick the corner points of the hole in the opposite direction than the corner points in the mesh.



5. Click the middle mouse button to finish the picking.
6. Pick a point to indicate the direction of the longitudinal bars.

Create a bent reinforcement mesh

1. On the **Concrete** tab, hold down **Shift** and click **Rebar** --> **Mesh**.

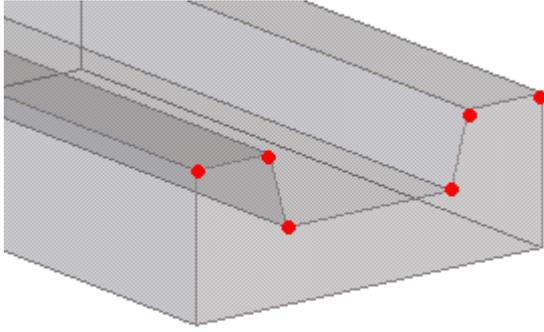


The **Rebar mesh** properties open in the property pane.

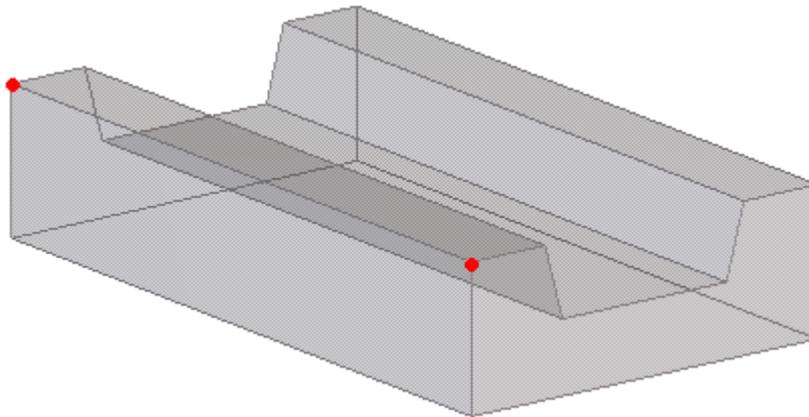
2. In the **Mesh type** list, select **Bent**.

NOTE You cannot change the mesh type once the mesh has been created.

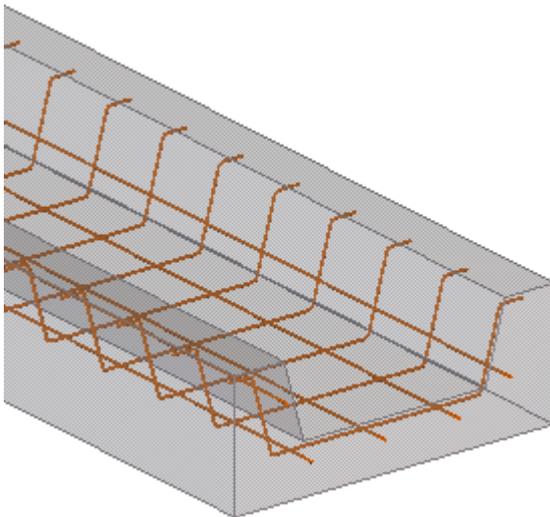
3. Enter the bending radius.
4. Select the part to reinforce.
Tekla Structures attaches the mesh to this part.
5. Pick points to indicate the bending shape of the crossing bars.



6. Click the middle mouse button to finish picking.
7. Pick two points to indicate the length and direction of the longitudinal bars.



Tekla Structures creates the mesh.



8. If you want to modify the reinforcement mesh, do one of the following:

- Use [direct modification \(page 496\)](#). Ensure that the  **Direct modification** switch is active.
- Double-click the reinforcement to open the **Rebar mesh** properties and modify the [properties \(page 929\)](#).

Create a customized reinforcement mesh

You can create a customized reinforcement mesh that consists of two perpendicular bar groups.

1. On the **Concrete** tab, hold down **Shift** and click **Rebar** --> **Mesh**.



The **Rebar mesh** properties open in the property pane.

2. In the **Layout** section, select the **Custom Mesh** option.
3. Enter a name for the mesh in the **Mesh** box.
The default name is **Custom Mesh**.
4. Modify the other mesh [properties \(page 931\)](#) as needed.
5. Select the part to reinforce.
Tekla Structures attaches the mesh to this part.
6. Pick two points to indicate the direction of the longitudinal bars.
7. If you want to define the mesh plane, pick one more point.
8. Click the middle mouse button to finish picking.
9. If needed, you can [save customized properties as property files \(page 111\)](#), and load these properties later on when you create new meshes.

Create a reinforcement strand pattern

You can create prestressed straight or deflected strands for concrete parts.

NOTE To allow positioning of the strands, first create points to the part you are creating the strands for. On the **Edit** tab, click **Points** and select **On plane** to open the **Point Array** dialog box. Define the point coordinates.

1. On the **Concrete** tab, click **Rebar** and select **Strand pattern**.

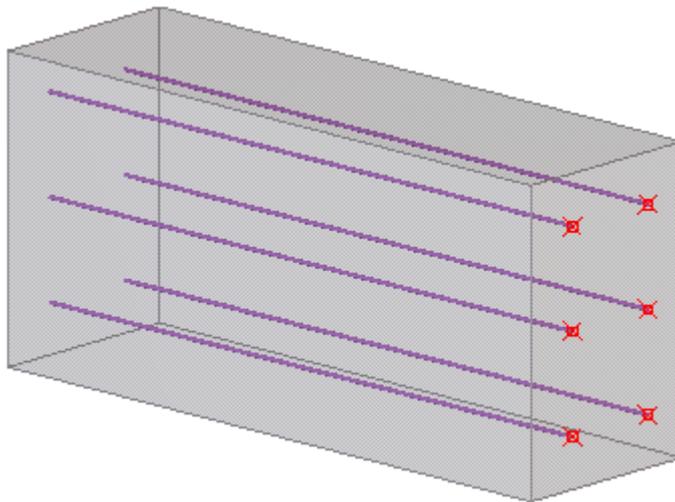


If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Strand pattern** command to open the **Strand pattern** properties.

2. Select the part you are creating strands for.
3. Pick each of the points that you are using to position the strands (for example, at the end of a part).

The points you pick define the first cross section.

4. Click the middle mouse button to finish picking.
 5. Pick points to position the strands.
 - If you create a single cross section, pick two points to define the length of the strands.
 - If you create two or more cross sections, for each cross section, pick points to indicate the strand positions. Pick the strand positions in the same order as for the first cross section.
 6. Click the middle mouse button to finish picking.
- Tekla Structures creates the strands.



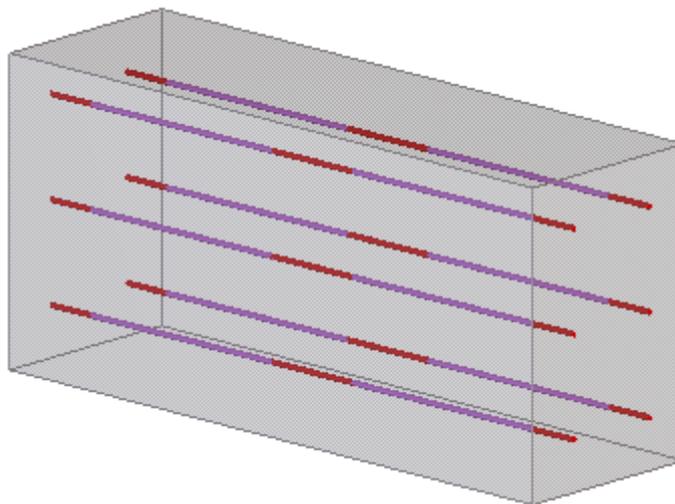
7. If you want to change the strand properties:
 - a. Double-click the strand pattern to open the **Strand pattern** properties.
 - b. Modify the [properties \(page 946\)](#).
 - c. Click **Modify**.

See also

[Debond reinforcement strands \(page 475\)](#)

Debond reinforcement strands

1. Double-click the strand pattern in which you want to debond strands.
The **Strand pattern** properties opens.
2. Click the **Debonding** button to open the debonding properties.
3. On the **Debonding** tab, click the **Add** button to create a new row in the table.
4. Enter the strand numbers in the **Debonded strands** field.
The strand number is the selection order number of the strand.
 - To set the same values for all the strands, enter all the strand numbers, separated by a space. For example, 1 2 3 4.
 - To set separate values for each strand, click **Add** to add a new row, then enter the strand number in the **Debonded strands** field.
5. Define the debonded lengths.
To set symmetrical lengths, select the **End lengths = start lengths** check box and only enter values in the **From start** or **Middle to start** fields.
6. Click **Modify**.
Tekla Structures displays the debonded section of the strand in red.



See also

[Create a reinforcement strand pattern \(page 473\)](#)

[Reinforcement strand properties \(page 946\)](#)

Create a reinforcement splice

You can join reinforcing bars or reinforcing bar groups together with reinforcement splices. There can be a gap between the bars or groups.

1. On the **Concrete** tab, click **Rebar** and select **Splice**.

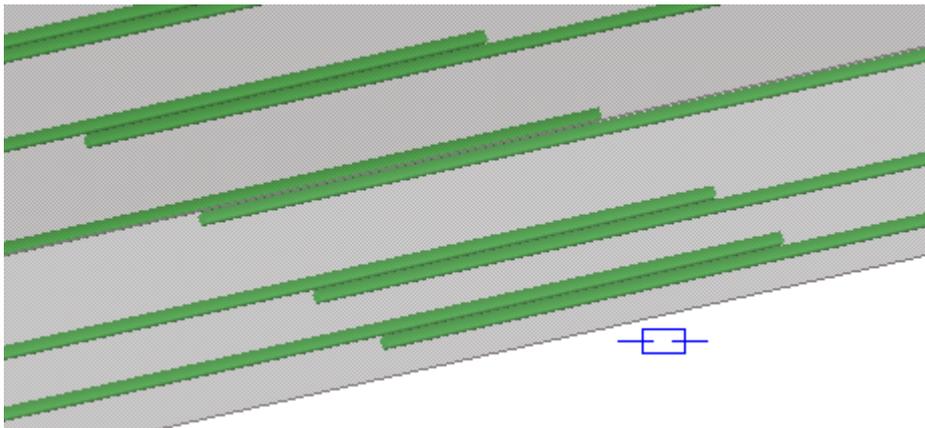


If you need to modify the properties before you create the reinforcement, hold down **Shift** and click the **Splice** command to open the **Rebar splice** properties.

2. Select the first reinforcing bar or bar group.
3. Select the second reinforcing bar or bar group.

Tekla Structures creates the splice. The splices have blue splice symbols

 in the model.



4. If you want to change the splice properties:
 - a. Double-click the splice to open the **Rebar splice** properties.
 - b. Modify the properties.
 - c. Click **Modify**.

Splice properties

Use the **Rebar splice** properties to view and modify the properties of splices. The file name extension of a saved splice property file is `.rsp`.

Option	Description
Joint type	Splice type. Lap left creates the lap to the direction of the first reinforcing bar or bar group selected, Lap right to the direction of the second. Lap both centers the lap between the bars or bar groups.
Lap length	Length of the lap joint.
Offset	Offset of the splice center point from the point where the bars originally met.
Bar positions	Select whether the lapping bars are on top of each other or parallel to each other.

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Split and splice a reinforcement \(page 515\)](#)

2.9 Modify reinforcement

Once you have added reinforcement to your model, you can modify the shape of the reinforcement, for example. Tekla Structures includes several methods for the modification.

Rebar sets

When you modify rebar sets, you can use direct modification on the rebar set guidelines, leg faces, and modifiers.

Single reinforcing bars, bar groups, and meshes

When you modify single reinforcing bars, bar groups, or meshes, you can use:

- [direct modification \(page 496\)](#)
- [handles \(page 506\)](#)
- [grouping \(page 504\)](#)
- [combining \(page 505\)](#)
- [splitting \(page 506\)](#)

See also

[Use adaptivity to modify a reinforcement \(page 512\)](#)

[Attach a reinforcement to a concrete part \(page 514\)](#)

[Split and splice a reinforcement \(page 515\)](#)

[Assign running numbers to reinforcement \(page 517\)](#)

[Classify reinforcement to layers \(page 517\)](#)

[How to calculate the reinforcing bar length \(page 518\)](#)

[How to calculate the reinforcing bar leg length \(page 521\)](#)

Modify a rebar set

You can modify rebar sets by changing the rebar set properties, by using the rebar set guidelines or leg faces, or by creating local rebar set modifiers. The guidelines, leg faces, and modifiers all have direct modification handles.

NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

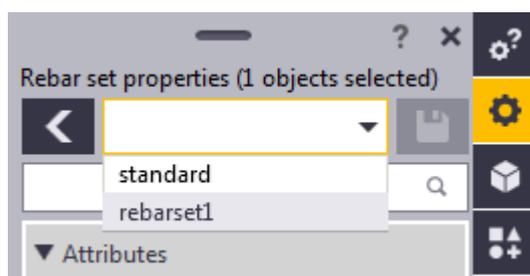
In case you open an existing model using a new version of Tekla Structures, always update the existing rebar sets first: on the **Concrete** tab, click **Rebar set** --> **Regenerate rebar sets**.

See also [Modify a rebar set using leg faces \(page 480\)](#) and [Modify a rebar set locally using modifiers \(page 486\)](#).

Modify the properties of a rebar set

You can change the properties of a rebar set in the property pane or on the contextual toolbar.

1. Double-click the rebar set that you want to modify.
2. If you want to use previously saved properties from a file, select the properties file from the topmost list in the property pane:



3. Modify the [rebar set properties \(page 933\)](#) in the property pane.
4. Click **Modify** to save the changes.
5. To save the properties for later use, enter a name for the properties file in the topmost box in the property pane, and then click .

TIP Alternatively, you can modify the rebar set properties on the contextual toolbar.

Change the layer order of a rebar set

You can adjust the order of the bar layers when two or more rebar sets overlap.

By default, the layer order is based on the creation order of the rebar sets. Tekla Structures automatically places the bars that are created first closest to the concrete surface, and the bars created last will be the furthest.

1. Select a rebar set.
2. On the contextual toolbar, adjust the layer order number by using the arrow buttons .

Alternatively, you can enter a number or use the arrow buttons in the property pane, and then click **Modify** to save the changes.

The smaller the layer order number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

If you set the same layer order number for several rebar sets, the bars will be placed on the same layer, and the bars may collide.

3. If needed, fine-tune the layer order at any individual [leg face \(page 480\)](#) separately.

These modifications override the default settings and the layer order settings of the entire rebar set.

Modify a rebar set using guidelines

The guidelines of a rebar set define the distribution direction of the bars. The spacing of the bars is also measured along the guidelines. You can modify the rebar set guidelines by using direct modification.

See also [Resize and reshape model objects \(page 103\)](#), [Distribute bars in a rebar set \(page 494\)](#), and [Create a secondary guideline \(page 490\)](#).

To modify a guideline, select a rebar set and do any of the following:

- To move a guideline, drag the line handle.
- To move a guideline point, drag the point handle .
- To add a new point to the start or end of a guideline:
 1. Select the start or end point of the guideline .
 2. Click  **Add new point** on the contextual toolbar.
 3. Pick a location for the new start or end point.

- To add an intermediate point to a guideline, drag a midpoint handle .
- To remove a point from a guideline, select the point and press **Delete**.
- To modify chamfers at intermediate corner points of a guideline:
 1. Select a corner point.
 2. Define the [chamfer type and dimensions \(page 368\)](#) on the contextual toolbar.

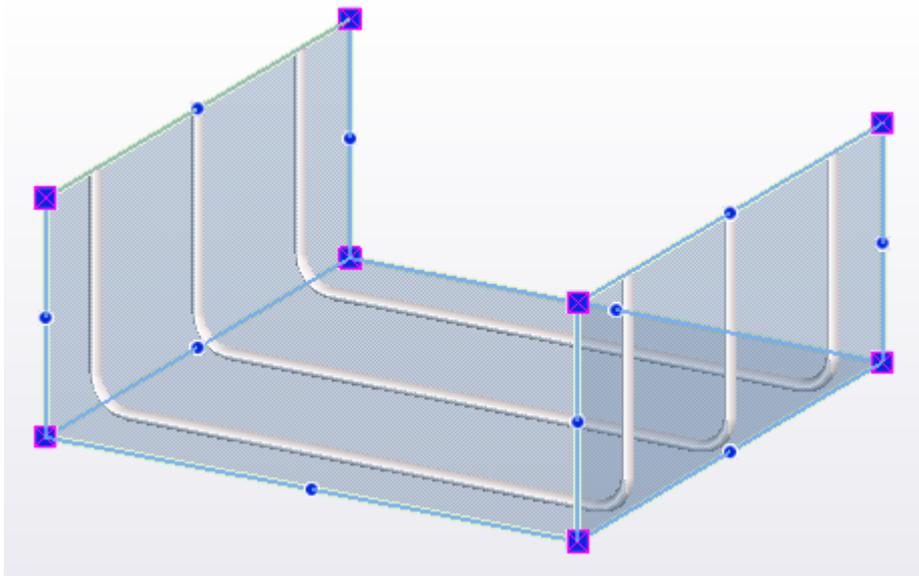
Modify a rebar set using leg faces

In addition to modifying an entire rebar set, you can make changes to any individual leg face.

Show the leg faces

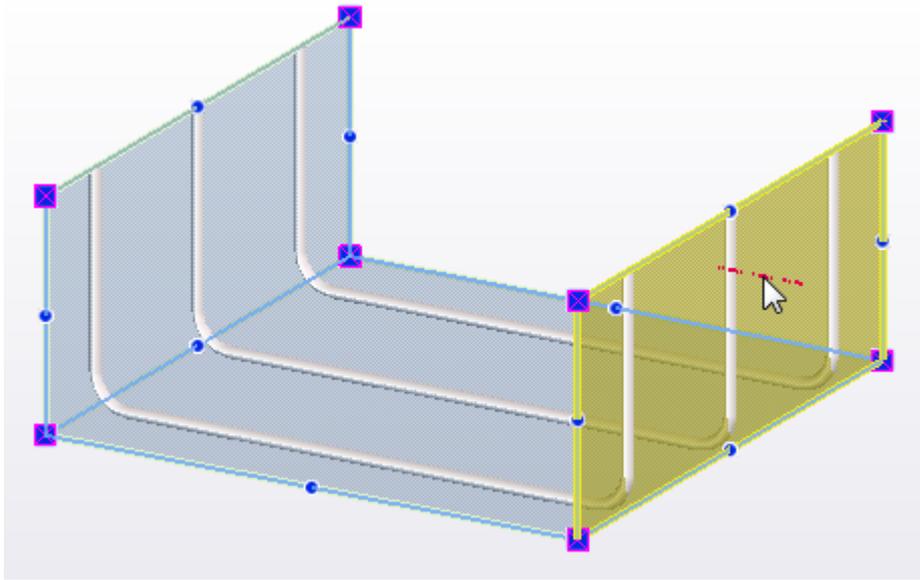
To modify rebar sets by using leg faces, you first need to make the leg faces visible.

1. Ensure that the  **Direct modification** switch is active.
2. On the **Concrete** tab, click **Rebar display options** --> **Leg face visibility**.
3. Select a rebar set.
Tekla Structures shows the leg faces.



4. Move the mouse pointer over a leg face and click to select it.

Tekla Structures highlights the leg face in yellow.

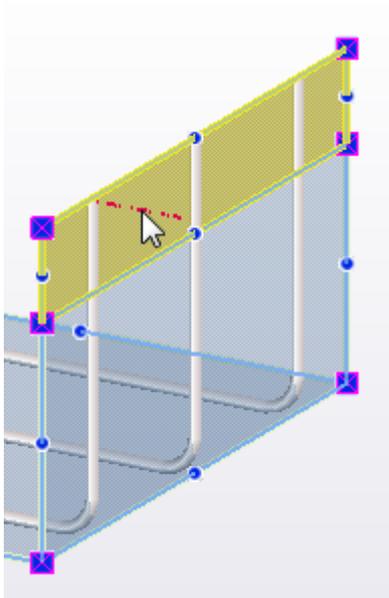
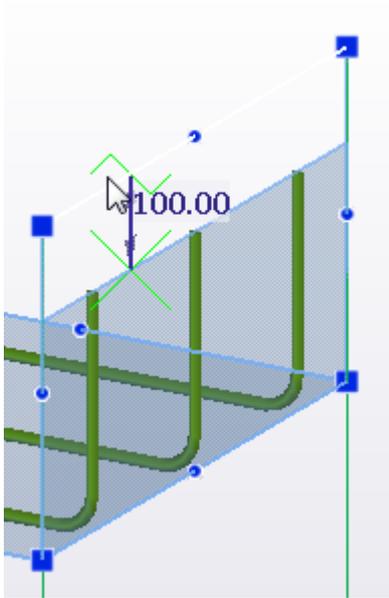


Alternatively, you can set the advanced option to `TRUE`.

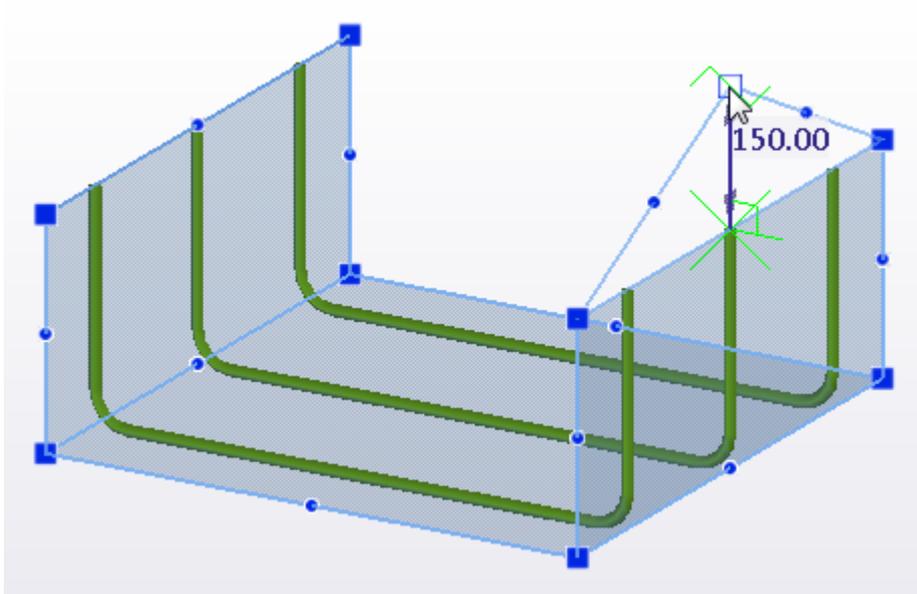
Modify the leg faces

You can use any of the following methods when you modify rebar set leg faces.

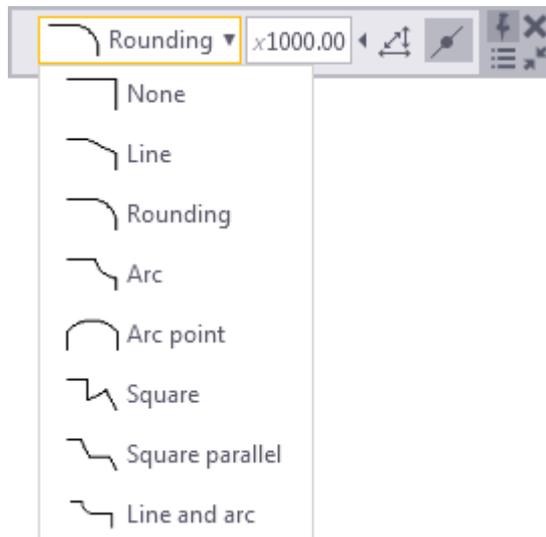
1. To move a leg face, drag it to a new location.
The connected leg face planes remain intact.
If you want the connected leg faces to follow the dragged leg face, hold down **Alt** when dragging. The size of the dragged leg face remains the same, but the connected leg face planes may change.
If you want to detach the leg face from the connected leg faces, hold down **Shift** when dragging.
2. To move an edge of a leg face, drag the edge to a new location.
The connected leg faces follow, if possible.
3. To create a parallel copy of a leg face, hold down **Ctrl** and drag the leg face.
4. To create a new, connected leg face, hold down **Ctrl** and drag an edge of the leg face.

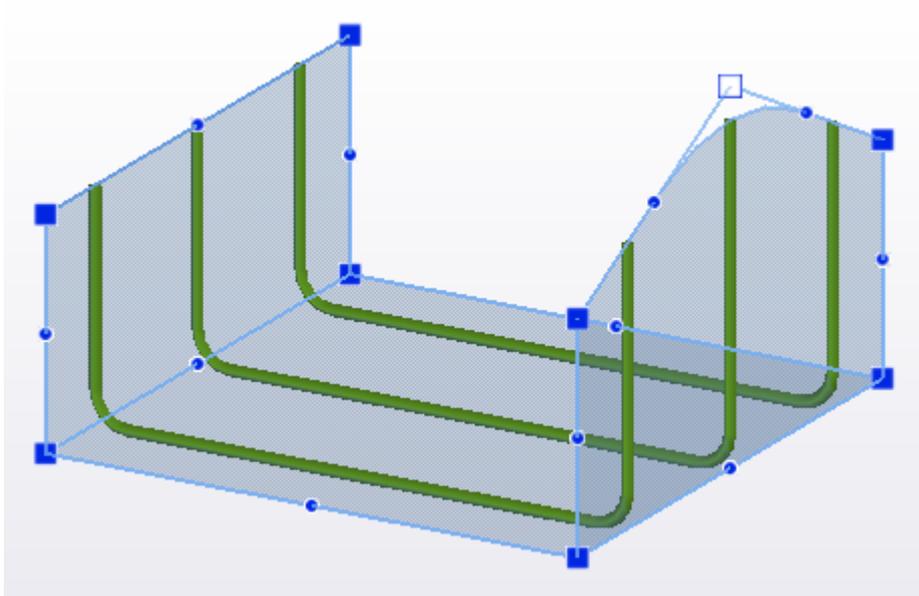


5. To add a new leg face at a part face or pour object face, click  **Add leg face** on the contextual tab on the ribbon, and then select the part face or pour object face.
6. To add a new corner point to a leg face, drag a midpoint handle.



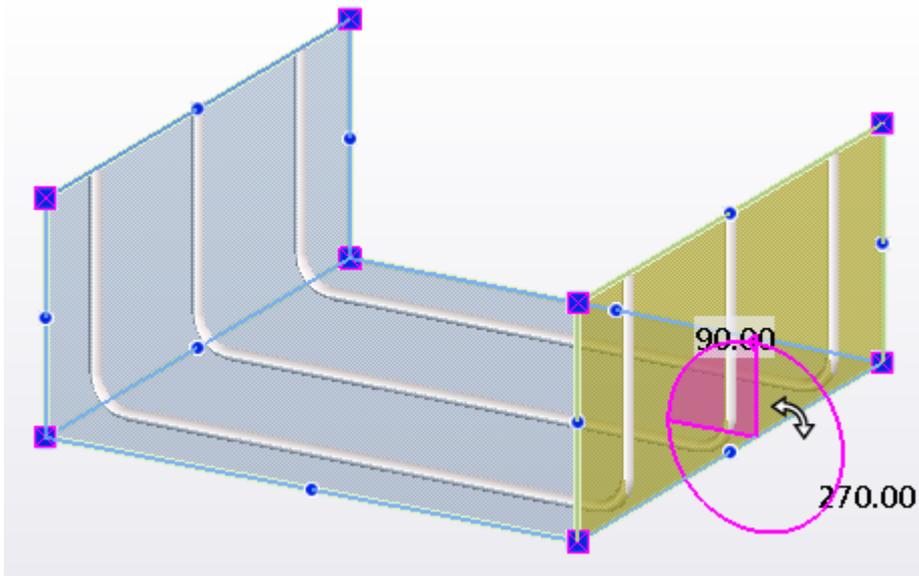
7. To remove a corner point from a leg face, select the point and press **Delete**.
8. To modify a corner chamfer of a leg face, select the corner point, and then select the [chamfer type \(page 368\)](#) and enter the chamfer dimensions on the contextual toolbar.





9. To rotate a leg face, select the leg face and click  **Enable leg face rotation** on the contextual toolbar.

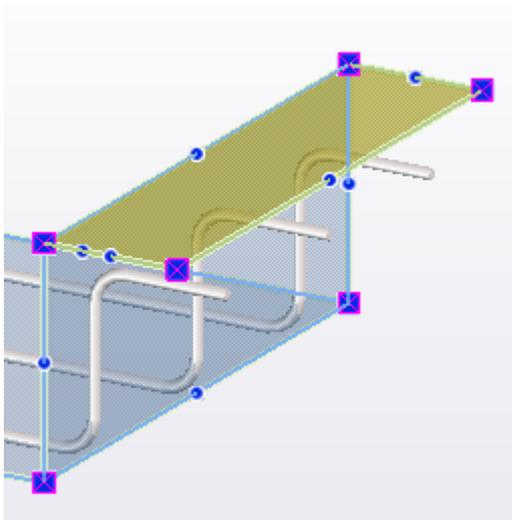
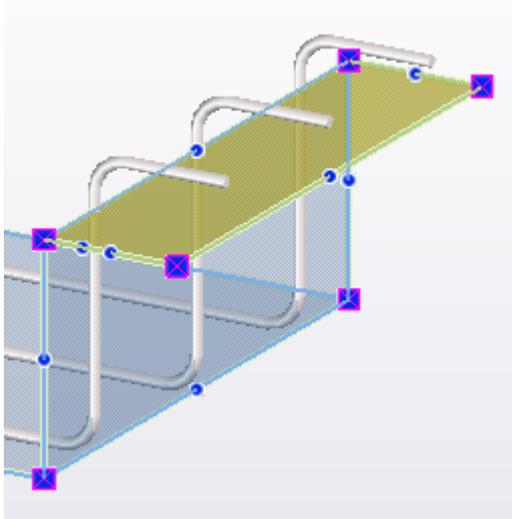
Tekla Structures shows a wheel symbol.



Drag the wheel symbol, or start typing the rotation angle. You can enter a positive or negative value in the **Enter a Numeric Location** dialog box.

10. To remove a leg face, select it and press **Delete**.

11. To define an additional offset between a leg face and the bars, select the leg face and enter a value for **Additional offset** on the contextual toolbar, for example, . A negative value moves the bars outside the concrete.
12. To flip bars over to the other side of a leg face, select the leg face and click  **Flip bar side** on the contextual toolbar.



Note that after flipping, Tekla Structures searches for concrete on the other side of the leg face to create a concrete cover and apply the concrete cover settings. If there is no concrete, the concrete cover thickness will be zero.

13. To change the order of the bar layers at an individual leg face, select the leg face and adjust the layer order number using the arrow buttons  on the contextual toolbar.

The smaller the layer order number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers.

These modifications override the layer order settings of the entire [rebar set \(page 478\)](#).

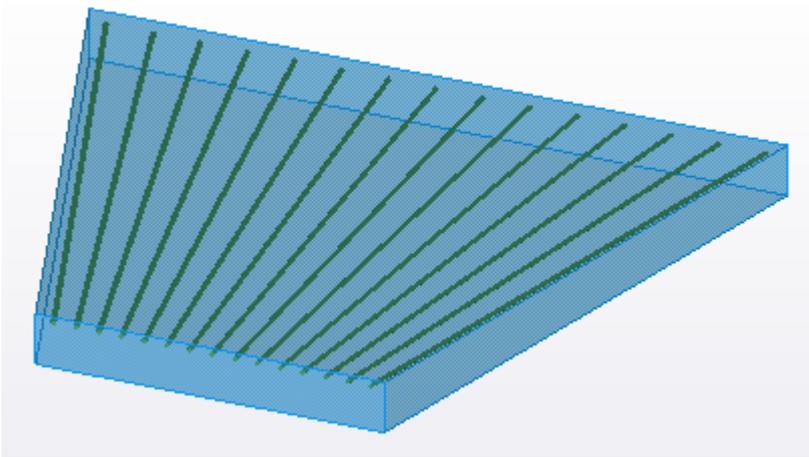
TIP You can modify [leg face properties \(page 938\)](#) also in the property pane.

Modify a rebar set locally using modifiers

You can use *modifiers* to modify a rebar set at certain locations only.

For example, you can create a local *property modifier* to only change the properties of certain bars in a rebar set, or you can create hooks or threading by adding an *end detail modifier*, or you can split rebar set bars with a *splitter*.

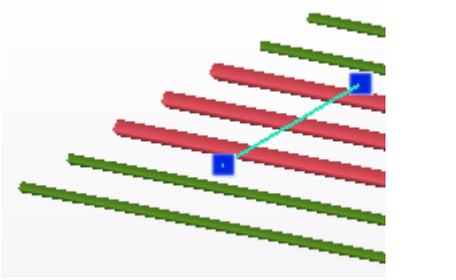
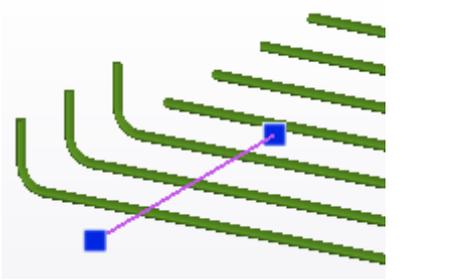
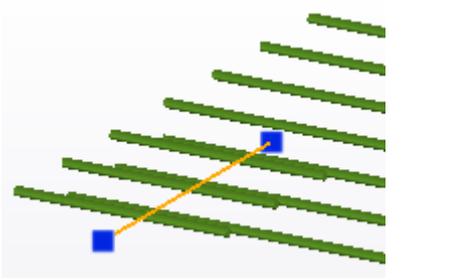
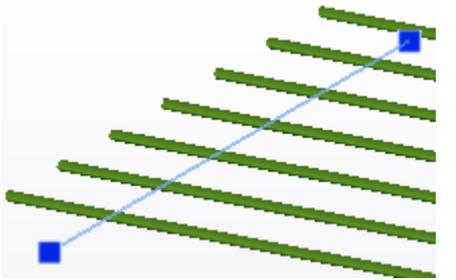
You can also create secondary guidelines for the rebar set. With a secondary guideline you can define a different spacing at the start and end of the rebar set bars, for example.



Modifiers are lines, or polylines that may have corner chamfers. Modifiers are projected to rebar set leg faces. Each modifier then only affects the rebar set bars that its projection touches.

NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

To select entire rebar sets, or bar groups or individual bars within rebar sets in the model, you can use the three reinforcement selection switches . Tekla Structures then shows the existing modifiers that affect the selected rebar set bars, and the direct modification handles of the modifiers. Different modifiers have different colors as follows:

Modifier	Color	Example
Property modifier	Light green	
End detail modifier	Magenta	
Splitter	Orange	
Secondary guideline	Light blue	

An arrowhead symbol  close to the midpoint of each modifier indicates the direction of the modifier, pointing from the start towards the end of the modifier.

When you select a modifier, Tekla Structures indicates the rebar set bars that are affected by the modifier, and shows the other, unaffected bars as semi-transparent.

You can modify the modifiers by using direct modification, or by changing their properties in the property pane or on the contextual toolbar. When you

change the modifier properties, the properties of the rebar set bars change at the location defined by the modifier.

When you delete a modifier, the rebar set reverts to the state it had without the modifier.

Create a property modifier

Property modifiers are shown in light green.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **+ Property modifier**.
3. Define how you want to place the modifier in the model.
Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

The  button indicates that you can pick one point, and the 

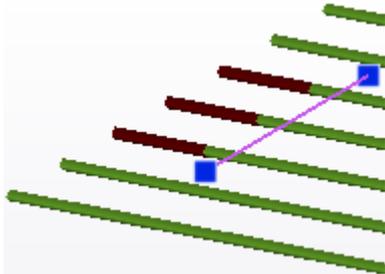
button indicates that you can pick multiple points. The  button indicates that the modifier will be created for the selected bars only.

4. Depending on the selected picking mode, do one of the following:
 - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
 - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
 - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the modifier location:
 - a. Select the modifier.
 - b. If you need to change the modifier geometry, drag the [direct modification \(page 103\)](#) handles.
 - c. Change the [property modifier properties \(page 938\)](#) on the contextual toolbar or in the property pane.
 - d. If you used the property pane, click **Modify** to save the changes.

Create an end detail modifier

End detail modifiers affect the nearest end of the bar, so by moving the modifier you can change the bar end that it affects. End detail modifiers are shown in magenta.

Tekla Structures displays the threaded bar ends in dark red.



1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.
2. On the **Rebar set** contextual tab on the ribbon, click  **End detail**.
3. Define how you want to place the modifier in the model.
Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

The  button indicates that you can pick one point, and the 

button indicates that you can pick multiple points. The  button indicates that the modifier will be created for the selected bars only.

4. Depending on the selected picking mode, do one of the following:
 - Pick one point to create a single line modifier for the selected rebar set, bar group, or bars.
 - Pick two points to define the end points of a single line modifier. Then click the middle mouse button.
 - Pick multiple points to create a polyline modifier. Then click the middle mouse button to finish picking.
5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the modifier location:
 - a. Select the modifier.
 - b. If you need to change the modifier geometry, drag the [direct modification \(page 103\)](#) handles.
 - c. Change the [end detail modifier properties \(page 941\)](#) on the contextual toolbar or in the property pane.

- d. If you used the property pane, click **Modify** to save the changes.

Create a splitter

Splitters split bars and create lap or crank splices. Splitters are shown in orange.

1. Using the reinforcement selection switches , select the rebar set bars for which you want to create the modifier.

2. On the **Rebar set** contextual tab on the ribbon, click  **Splitter**.

3. Define how you want to place the splitter in the model.

Click the **Picking mode** button on the contextual tab to cycle through the picking modes and to select the picking mode.

The  button indicates that you can pick one point, and the 

button indicates that you can pick multiple points. The  button indicates that the splitter will be created for the selected bars only.

4. Depending on the selected picking mode, do one of the following:
 - Pick one point to create a single line splitter for the selected rebar set, bar group, or bars.
 - Pick two points to define the end points of a single line splitter. Then click the middle mouse button.
 - Pick multiple points to create a polyline splitter. Then click the middle mouse button to finish picking.

TIP If you want to create the splitter at a rounded distance from a bar end, and the dimension shown is measured from the other bar end, hold down **Shift** when you are placing the splitter in the model to switch the measuring point to the other bar end.

5. To end the command, press **Esc**.
6. To apply local modifications to the rebar set bars at the splitter location:
 - a. Select the splitter.
 - b. If you need to change the splitter geometry, drag the [direct modification \(page 103\)](#) handles.
 - c. Change the [splitter properties \(page 943\)](#) on the contextual toolbar or in the property pane.
 - d. If you used the property pane, click **Modify** to save the changes.

Create a secondary guideline

You can create up to two secondary guidelines for a rebar set. Secondary guidelines are shown in light blue.

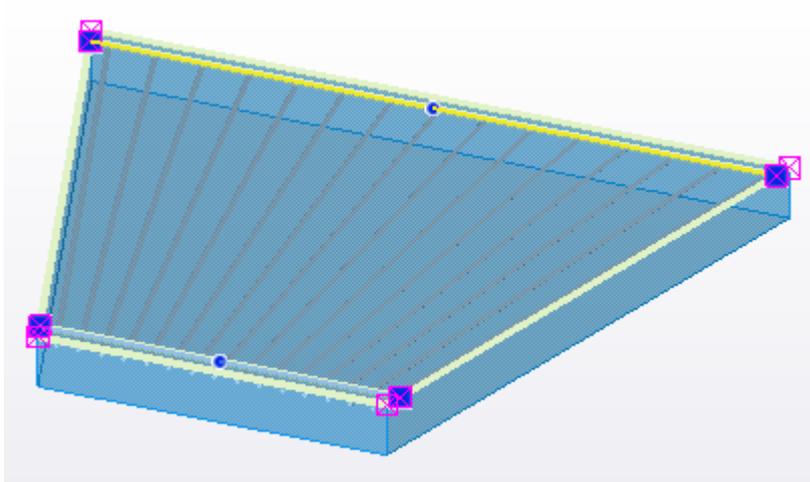
1. Select a rebar set.
2. On the **Rebar set** contextual tab on the ribbon, click  **Secondary guideline**.
3. Define how you want to place the guideline in the model.

The  button on the contextual tab indicates that you can pick one point, and the  button indicates that you can pick multiple points. Click the button to change the picking mode.

4. Depending on the picking mode, do one of the following:
 - Pick one point to create a single line guideline.
 - Pick two points to define the end points of a single line guideline. Then click the middle mouse button.
 - Pick multiple points to create a polyline guideline. Then click the middle mouse button to finish picking.
5. To create another secondary guideline, repeat steps 3 and 4.

If there are already two secondary guidelines in a rebar set, the tooltip of the  button changes to **Maximum number of secondary guidelines reached** and you cannot create more guidelines.
6. Press **Esc** to stop creating secondary guidelines.
7. If needed, select a secondary guideline and modify its [geometry \(page 103\)](#) and [properties \(page 936\)](#).

For example, you can adjust the secondary guideline's length or spacing values.



See also [Distribute bars in a rebar set \(page 494\)](#).

8. To set a secondary guideline as the primary guideline, select the secondary guideline and click **+1 Set as primary** on the contextual toolbar.

TIP Alternatively, you can create secondary guidelines the same way as you [copy other modifiers \(page 492\)](#): hold down **Ctrl** and drag the primary guideline.

Create a modifier by copying

You can copy rebar set modifiers.

1. Select a rebar set, bar group, or bar to show its modifiers.
2. Select the modifier that you want to copy.
3. Hold down **Ctrl** and drag the modifier to the desired location.
Tekla Structures creates a new modifier when you release the mouse button.
4. Select the modifier to modify its [geometry \(page 103\)](#) and properties as needed.

Show or hide rebar set modifiers

If you have many rebar set modifiers in your model, it may be useful to only show some of them at a time and hide the ones that are not currently needed. You can show and hide modifiers according to their type.

For example, you can show the end detail modifiers only, and hide all property modifiers and splitters.

1. Go to the **Concrete** tab and click **Rebar display options**.
2. Do any of the following:

- Click **Property modifier visibility** to switch the property modifiers on or off.
- Click **End detail modifier visibility** to switch the end detail modifiers on or off.
- Click **Splitter visibility** to switch the splitters on or off.

Alternatively, you can use the following advanced options:

-
-
-

How to cut rebar sets

You can cut rebar sets automatically by the existing cuts in the concrete parts or manually by using the cutting commands on the **Edit** tab. You can modify cuts in rebar sets the same way as you modify cuts in parts in the model, by using direct modification.

You can use the following commands to create cuts:

- [Line cut \(page 362\)](#)
- [Polygon cut \(page 363\)](#)
- [Part cut \(page 364\)](#)

Concrete cover settings are also applied to cuts, even on cut edges that are parallel to reinforcing bars.

Cut a rebar set using a cut in a concrete part

When you create rebar sets for concrete parts using the **Create longitudinal rebars**, **Create crossing rebars**, and **Create planar rebars** commands, Tekla Structures automatically cuts the new rebar sets using the existing cuts in the concrete parts. If you add a new cut to a concrete part with a rebar set, the rebar set is not automatically cut. If you want to cut the rebar set as well, use the **Part cut** command, and use the new cut as the cutting part.

1. On the **Edit** tab, click **Part cut**.
2. Select the rebar set you want to cut.
3. Select the cut in the concrete part.

Tekla Structures cuts the rebar set.

Modify a cut in a rebar set

You can modify cuts in rebar sets using direct modification. For example, you can make a cut in a rebar set of a different size or shape from a cut in a concrete part.

1. Ensure that the  **Direct modification** switch is active.
2. Select the cut in the rebar set.
3. Modify the cut using [direct modification \(page 103\)](#).

Distribute bars in a rebar set

Rebar sets may have zones with different spacing values between the bars. The spacing of the bars is measured along the rebar set guideline. You can modify the spacing settings in the spacing mode.

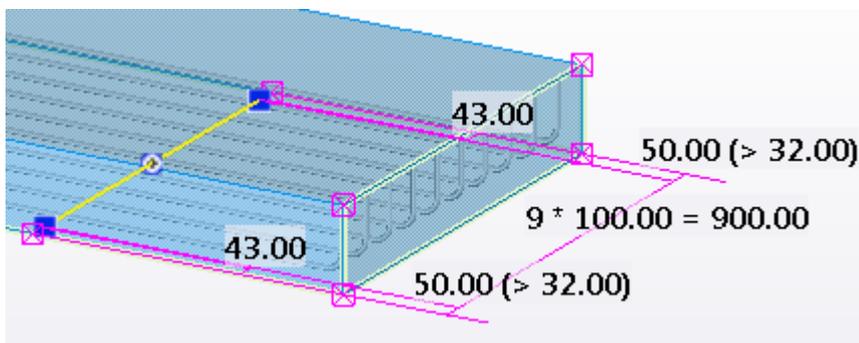
NOTE When you work with rebar sets, ensure that the  **Direct modification** switch is active.

Switch to the spacing mode

To define the spacing of bars in a rebar set, switch to the spacing mode. When the spacing mode is active, you cannot modify the geometry of the rebar set guideline.

1. Select a rebar set.
2. On the contextual toolbar, click  **Edit spacing properties**.

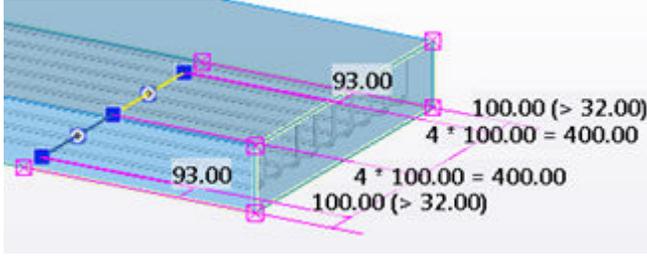
Tekla Structures shows the spacing zone dimensions in the model, and the spacing zone properties in the property pane, and on the contextual toolbar.



Add, move, and delete spacing zones

By default, there is always one spacing zone in each rebar set. You can add any number of spacing zones, and move and delete spacing zones.

To	Do this
Add a spacing zone	<ol style="list-style-type: none"> 1. In the spacing mode, click  in the middle of a spacing zone on the rebar set guideline. Tekla Structures splits the spacing zone in two.

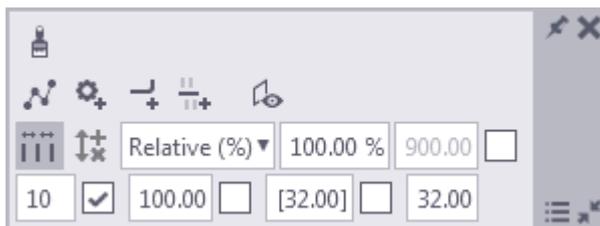
To	Do this
	 <ol style="list-style-type: none"> <li data-bbox="671 555 1347 689">2. To modify the spacing zone properties (page 933), select a spacing zone and modify its properties in the property pane or on the contextual toolbar.
Move, lengthen, or shorten a spacing zone	<ol style="list-style-type: none"> <li data-bbox="671 701 1302 775">1. In the spacing mode, drag a spacing zone handle to a new location.
Delete a spacing zone	<ol style="list-style-type: none"> <li data-bbox="671 784 1353 857">1. In the spacing mode, select a spacing zone in the model. <li data-bbox="671 864 919 898">2. Press Delete.

Modify the properties of a spacing zone

For each spacing zone within a rebar set, you can modify the length, number of spaces, and spacing value. You can also define offsets for the first and last spacing zone of a rebar set.

You can modify the spacing zone properties in the property pane or on the contextual toolbar.

On the contextual toolbar, the start offset is shown together with the first spacing zone, and the end offset with the last spacing zone. If there is only one spacing zone in a rebar set, both the start and end offsets are shown.

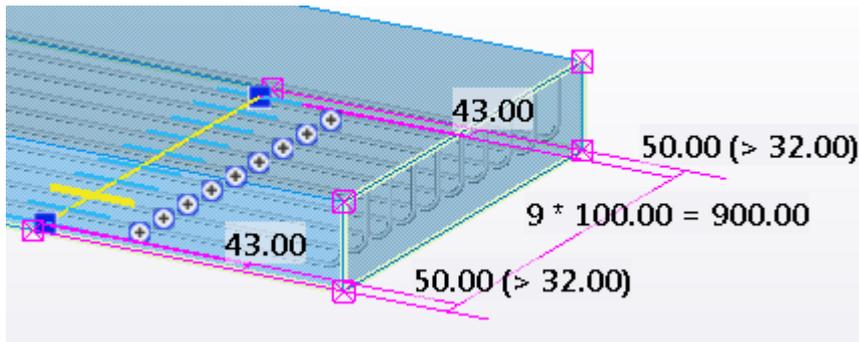


1. While in the spacing mode, select a spacing zone.
2. Modify the [properties \(page 933\)](#) in the property pane or on the contextual toolbar.
3. If you used the property pane, click **Modify** to save the changes.

Add, move, and delete single bars

When you add, move, or delete single bars within a rebar set, it also affects the spacing zones.

1. Select a rebar set.
2. On the contextual toolbar, click  **Edit spacing properties**.
The spacing mode is activated.
3. On the contextual toolbar, click  **Enable add/move/delete bars**.
Tekla Structures displays a line handle for each bar on the rebar set guideline.



4. Do any of the following:
 - To add a bar between two existing bars, click .
 - To move a bar, select the line handle of the bar and drag it to a new location.
You can also use the keyboard to [enter a numeric location \(page 87\)](#).
To start with the negative sign (-), use the numeric keypad. To enter an absolute coordinate, first enter \$, then the value. Press **Enter** to confirm.
Note that you cannot move the bar more than the spacing value, beyond the adjacent bars.
 - To delete a bar, select the line handle of the bar and press **Delete**.

Modify a single reinforcing bar, bar group, or mesh

You can modify a reinforcement by using direct modification. You can either modify the reinforcement simply by dragging handles, or select a command from the contextual toolbar.

NOTE Direct modification does not work for the following reinforcement types:

- [circular \(page 463\)](#) and [curved \(page 461\)](#) reinforcing bars
- [reinforcement strand patterns \(page 473\)](#)
- [detached reinforcing bars \(page 514\)](#).

If you have created the reinforcement using a component, you need to explode the component before using direct modification.

Before you start:

- Ensure that the  **Direct modification** switch is active.
- Select the reinforcement.

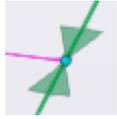
Tekla Structures displays the handles that you can use to modify the reinforcement, and a toolbar icon . Click the icon to open the toolbar and to select the appropriate command. The available commands depend on the type of the reinforcement you are modifying.



To modify single reinforcing bars, reinforcing bar groups, or reinforcement meshes:

To	Do this	Command available for
Change the cover thickness of a reinforcing bar	Drag a line handle to the desired location. 	Reinforcing bars, reinforcing bar groups, reinforcement meshes

To	Do this	Command available for
Add polygon points to a reinforcing bar	Drag a midpoint handle  to the desired location.	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add points to the start or end of a reinforcing bar	<ol style="list-style-type: none"> 1. Click the start or end reference point of the reinforcing bar . 2. Click the Add new point button  on the toolbar. 3. Pick a location for the new start or end point. 	Reinforcing bars, reinforcing bar groups
Remove points from a reinforcing bar	<ol style="list-style-type: none"> 1. Select one or more reference points. 2. Press Delete. 	Reinforcing bars, reinforcing bar groups, polygonal and bent reinforcement meshes
Add hooks	<ol style="list-style-type: none"> 1. Click the start or end point of the reinforcing bar . A toolbar for hook properties appears. 2. Select the desired shape for the hook. 3. If you selected Custom hook, enter the angle, radius, and length for the hook and click . 	Reinforcing bars, reinforcing bar groups
Change the bending radius of a reinforcing bar	<ol style="list-style-type: none"> 1. Click the Change bending radius button  on the toolbar. 2. Enter a value in the box next to the Change bending radius button and press Enter. 	Reinforcing bars, reinforcing bar groups
Change the diameter of a reinforcing bar	<ol style="list-style-type: none"> 1. Click the Change diameter button  on the toolbar. 	Reinforcing bars, reinforcing bar groups,

To	Do this	Command available for
	2. Select a value from the list next to the Change diameter button.	reinforcement meshes
Modify the spacings by adjusting the range	1. Click the Modify spacings button  on the toolbar. 2.  Drag a handle  to the desired location.	Reinforcing bar groups, reinforcement meshes
Modify the spacings by splitting the range in two	1. Click the Modify spacings button  on the toolbar. 2. Drag a midpoint handle  to the desired location and release the handle. Tekla Structures creates a new reinforcing bar and the range is split in two. The spacing in the two new ranges is as close as possible to the original spacing. 3. If needed, change the number of spaces or the spacing value. Click the midpoint handle and enter the desired values in the boxes on the toolbar and press Enter .	Reinforcing bar groups, reinforcement meshes
Move, add, or remove reinforcement	1. Click the Move, add, delete reinforcement button  on the toolbar. Tekla Structures displays the line handles for each reinforcing bar. 2. Do one of the following: <ul style="list-style-type: none"> • To move a reinforcing bar, highlight it and drag it to the desired location. • To add a reinforcing bar between two reinforcing bars, click . 	Reinforcing bar groups, reinforcement meshes

To	Do this	Command available for
	<ul style="list-style-type: none"> To delete reinforcing bars, select them and press Delete. 	

See also

[Use handles to modify a reinforcement \(page 506\)](#)

[Use adaptivity to modify a reinforcement \(page 512\)](#)

[Check the validity of reinforcement geometry \(page 514\)](#)

Distribute bars in a reinforcing bar group

You can select how reinforcing bars are distributed in a bar group by modifying the bar spacings.

To modify the spacings of bars in a reinforcing bar group, do one of the following:

To	Do this
Modify the spacings using direct modification (page 496)	<ol style="list-style-type: none"> Ensure that the Direct modification switch  is active. Select a reinforcing bar group. On the contextual toolbar, click the Modify the spacings button .  Drag a handle  to the desired location.
Modify the spacings using the Rebar group properties	<ol style="list-style-type: none"> Select a reinforcing bar group. Double-click the reinforcement to open the Rebar group properties. In the Distribution section, select a spacing option from the Creation method list. Enter the required values. Click Modify.

Spacing options in the **Creation method** list:

Option	Description	Example
Equal distribution by number of reinforcing bars	Enter the number of reinforcing bars. Tekla Structures divides the available distance by the number of bars. Enter the number of bars in the Number of reinforcing bars box.	
Equal distribution by target spacing value	Enter a spacing value. Tekla Structures aims the spacing value as close as possible to the value in the Target spacing value box.	
By exact spacing value with flexible first	Enter the spacing value in the Exact spacing value box. Creates fixed, regular spaces between the bars. The first space adjusts to even out the bar distribution. If the first space is less than 10% of the exact spacing value, Tekla Structures removes one bar.	
By exact spacing value with flexible last space	Enter the spacing value in the Exact spacing value box. Creates fixed, regular spaces between the bars. The last space adjusts to even out bar distribution.	
By exact spacing value with flexible middle space	Enter the spacing value in the Exact spacing value box. Creates fixed, regular spaces between the bars. The middle space adjusts to even out bar distribution. If there are an odd number of bars (two middle spaces), the other middle space adjusts to even out bar distribution.	
By exact spacing value with flexible first and last space	Enter the spacing value in the Exact spacing value box. Creates fixed, regular spaces between the bars. Both the first and last spaces adjust to even out bar distribution.	

Option	Description	Example
By exact spacings	Enter the spacing values manually in the Exact spacing values box. Use multiplication to repeat spacings, for example, 5*200 to create five spaces of 200.	

See also

[Create a reinforcing bar group \(page 452\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

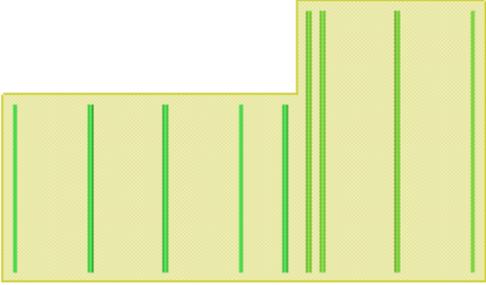
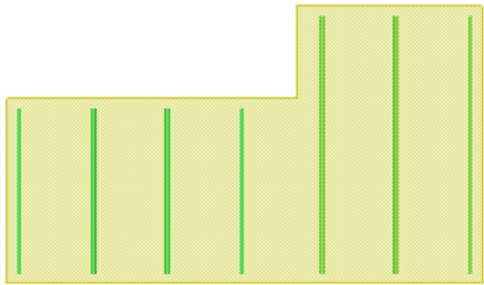
Delete bars from a reinforcing bar group

You may occasionally need to delete or exclude specific reinforcing bars. For example, when several reinforced areas intersect, causing reinforcing bars to overlap, or when you want to start bar distribution at a specific distance from the end of a part.

To delete reinforcing bars from a group, do one of the following:

To	Do this
Delete bars using direct modification (page 496)	<ol style="list-style-type: none"> 1. Ensure that the Direct modification switch  is active. 2. Select a reinforcing bar group. 3. On the contextual toolbar, click the Move, add, delete reinforcement button . 4. Select the bars to be deleted and press Delete.
Delete bars using the Rebar group properties	<ol style="list-style-type: none"> 1. Select a reinforcing bar group. 2. Double-click the reinforcement to open the Rebar group properties. 3. In the Creation section, select an option from the Exclude list. 4. Click Modify.

Examples when you have used the **Exclude** options:

Before excluding bars	After excluding bars
<p>Two reinforcing bar groups have been added to a concrete beam:</p> <ul style="list-style-type: none"> • one bar group with flexible last space • one bar group with flexible first space 	<p>Two reinforcing bar groups without the excluded bars:</p> <ul style="list-style-type: none"> • one bar group with the last bar excluded • one bar group with the first bar excluded 

See also

[Create a reinforcing bar group \(page 452\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

Ungroup a reinforcement

You can ungroup reinforcing bar groups and reinforcement meshes. Only reinforcement where each reinforcing bar is in one plane can be ungrouped.

NOTE You cannot ungroup [circular \(page 463\)](#) or [curved \(page 461\)](#) reinforcing bar groups.

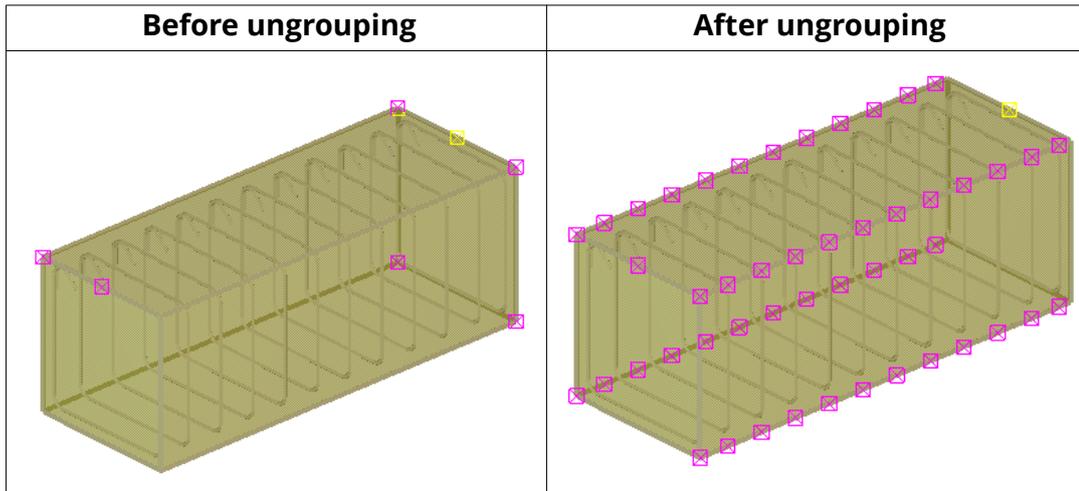
1. On the **Concrete** tab, click **Rebar** and select **Ungroup**.



2. Select one of the reinforcing bars in a reinforcing bar group or in a reinforcement mesh.

The reinforcing bar group is replaced with single reinforcing bars. The single bars get the same properties and offsets as the group.

If you ungroup a reinforcement mesh, the offsets for single bars are zero.



See also

[Modify reinforcement \(page 477\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a reinforcement mesh \(page 467\)](#)

Group reinforcement

You can group single reinforcing bars and reinforcing bar groups. Only reinforcement where each reinforcing bar is in one plane can be grouped. All groups are created with exact spacings. Single reinforcing bars need to have the same bending shape.

NOTE You cannot create [circular \(page 463\)](#) or [curved \(page 461\)](#) reinforcing bar groups by grouping.

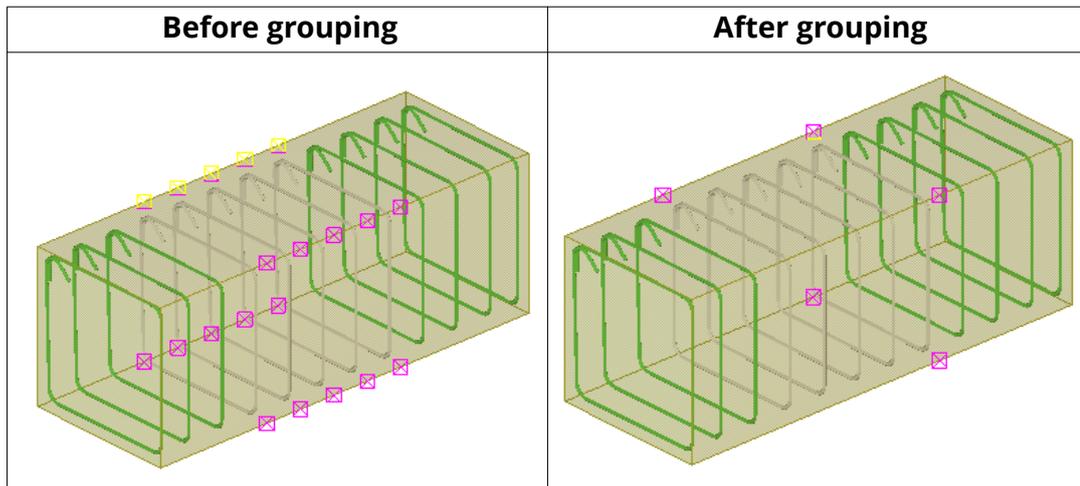
1. On the **Concrete** tab, click **Rebar** and select **Group**.



2. Select all the reinforcing bars or reinforcing bar groups that you want to group.
3. Click the middle mouse button.
4. Select one reinforcing bar or reinforcing bar group to copy the properties from.

The new group gets the same properties as the selected reinforcing bar.

NOTE The reinforcing bar or reinforcing bar group that you copy the properties from is also added to the group. This means, for example, that you cannot copy properties from a separate reinforcing bar group which you do not want to include in your new reinforcing bar group.



See also

[Modify reinforcement \(page 477\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a single reinforcing bar \(page 451\)](#)

Combine two reinforcing bars or reinforcing bar groups into one

You can combine two single reinforcing bars or reinforcing bar groups into one. Reinforcing bars can be combined if their end points are connected, or the bars are parallel and close to each other. However, in certain cases it is possible to combine bars or groups that are not connected nor parallel. The combined reinforcement gets the same properties as the bar selected first.

NOTE You cannot combine **Tapered N** reinforcing bar groups.

1. On the **Edit** tab, click **Combine**.
 2. Select the first single bar or bar group to be combined.
 3. Select the second single bar or bar group to be combined.
- Tekla Structures combines the reinforcing bar groups or bars into one.

See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a single reinforcing bar \(page 451\)](#)

[Modify reinforcement \(page 477\)](#)

Split a reinforcing bar group

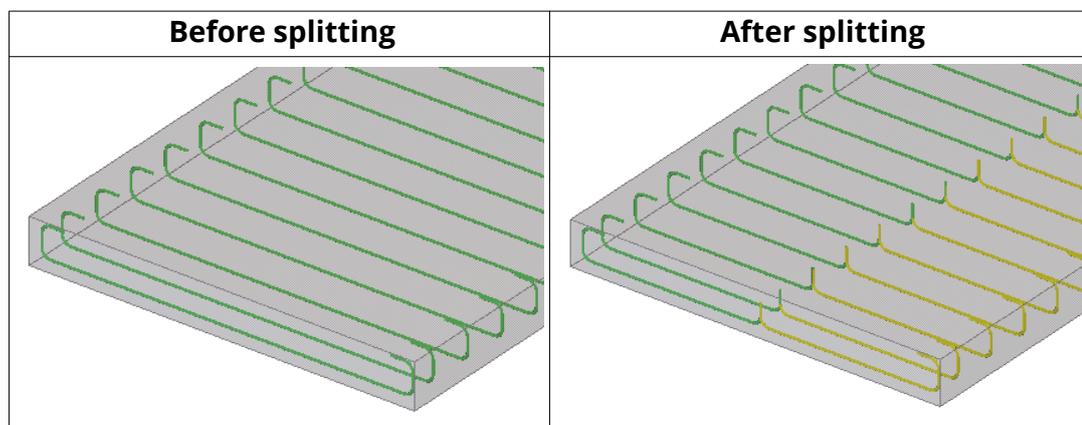
You can split normal and tapered reinforcing bar groups into two groups. You can also split single reinforcing bars into two.

1. On the **Edit** tab, click **Split**.
2. Select a reinforcing bar group.
3. Pick two points to indicate where to split the group.

Tekla Structures splits the reinforcing bar group.

NOTE You cannot split reinforcing bar groups diagonally.

Once split, each new reinforcing bar group retains the properties of the original group. For example, if the bars in the original group had hooks at both ends, bars in the new groups also have hooks at both ends. Modify the properties of the new groups, if needed.



See also

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a reinforcing bar group \(page 452\)](#)

[Create a single reinforcing bar \(page 451\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

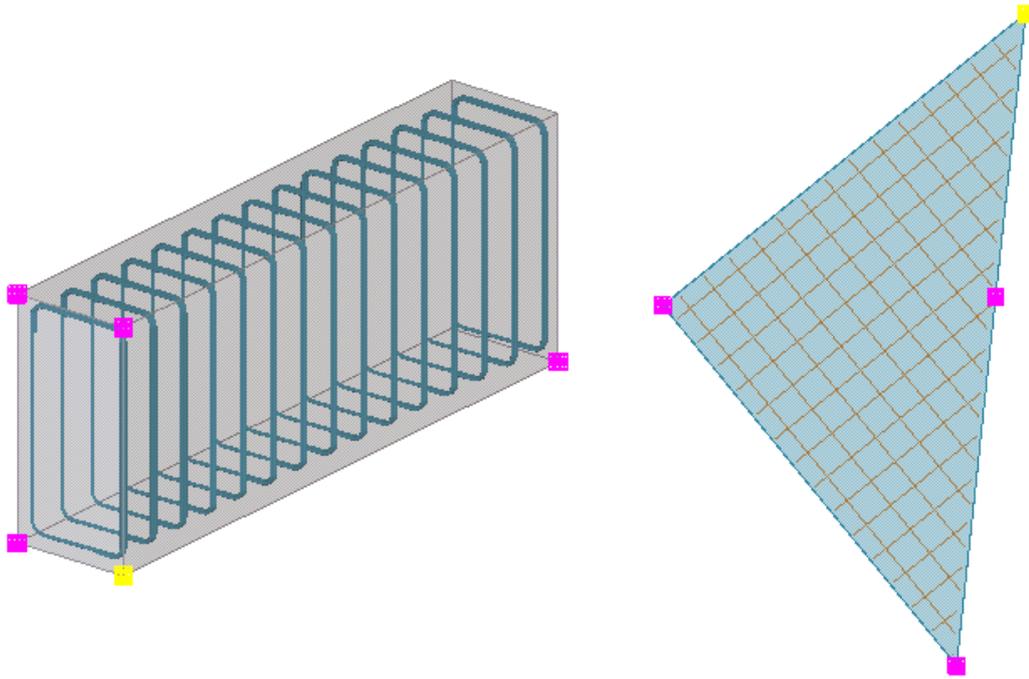
Use handles to modify a reinforcement

If you do not want to use direct modification to modify a reinforcement, you can use, for example, the reinforcement handles to modify the reinforcement.

Tekla Structures uses handles to indicate:

- The ends and corners of a reinforcing bar.
- The distribution length of a bar group.
- The corners and main bar direction of a mesh.

When you select a reinforcement, Tekla Structures highlights the handles. The handle of the first end point is yellow, the rest are magenta.



1. Select the reinforcement.
Tekla Structures highlights the handles.
2. Click one of the handles to select it.
3. Move the handle like any other object in Tekla Structures.

For example, if **Drag and drop** is active, just drag the handle to a new position.

NOTE If you want to use the reinforcement handles, ensure that the

Direct modification switch  is not active. If the switch is active and [direct modification \(page 496\)](#) is on, Tekla Structures displays direct modification handles for the reference points,

ends, legs, and leg midpoints of the selected reinforcement. These handles are blue.

See also

[Check the validity of reinforcement geometry \(page 514\)](#)

Add hooks to reinforcing bars

You can add hooks to the ends of reinforcing bars for anchoring purposes.

NOTE Hooks are intended to be used only for anchoring purposes. Do not use hooks as a method to model other reinforcing bar geometry, because it may cause problems with visibility in drawings, in adaptivity, and in bar bending shape recognition.

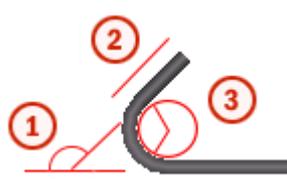
To add hooks to reinforcing bars, do one of the following:

To	Do this
Add hooks using direct modification (page 496)	<ol style="list-style-type: none"> 1. Ensure that the  Direct modification switch is active. 2. Select a single reinforcing bar or a reinforcing bar group. 3. Click the start or end point of the  reinforcing bar . A toolbar for hook properties appears. 4. Select the desired shape for the hook. 5. If you select Custom hook, enter the angle, radius, and length for the hook. Click .
Add hooks using the Single rebar or Rebar group properties	<ol style="list-style-type: none"> 1. Select a single reinforcing bar or a reinforcing bar group. 2. Double-click the reinforcement to open its properties. 3. In the Hooks section, select a hook type for the bar start and/or end from the Hook type list. 4. If you select Custom hook, enter the angle, radius and length for the hook.

To	Do this
	5. Click Modify .
Add hooks to rebar sets using end detail modifiers	See Modify a rebar set locally using modifiers (page 486) .

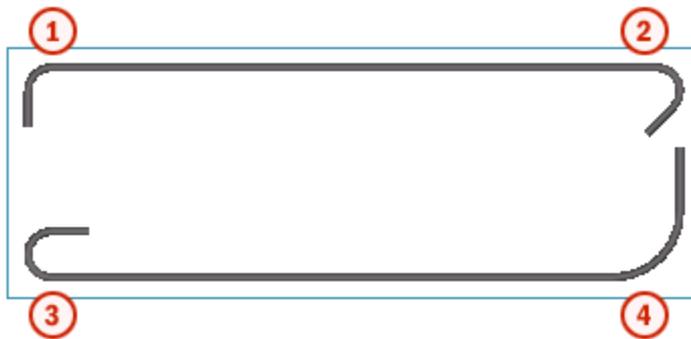
For custom hooks you need to enter the hook information:

Option	Description
Angle	Enter a value between -180 and +180 degrees.
Radius	Enter the internal bending radius of the hook. Use the same radius for the hook and for the reinforcing bar. If the hook and the reinforcing bar have different radiuses, Tekla Structures does not recognize the shape of the bar.
Length	Enter the length of the straight part. If the length is set to zero, no hooks are created.



1. Angle
2. Length
3. Radius

Hook examples



	Description
1	Standard 90-degree hook
2	Standard 135-degree hook
3	Standard 180-degree hook
4	Custom hook

If you select a standard hook, the **Angle**, **Radius**, and **Length** use predefined dimensions.

The `rebar_database.inp` file contains the predefined minimum bending radius and minimum hook length for all standard hooks.

See also

[Create a reinforcing bar group \(page 452\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

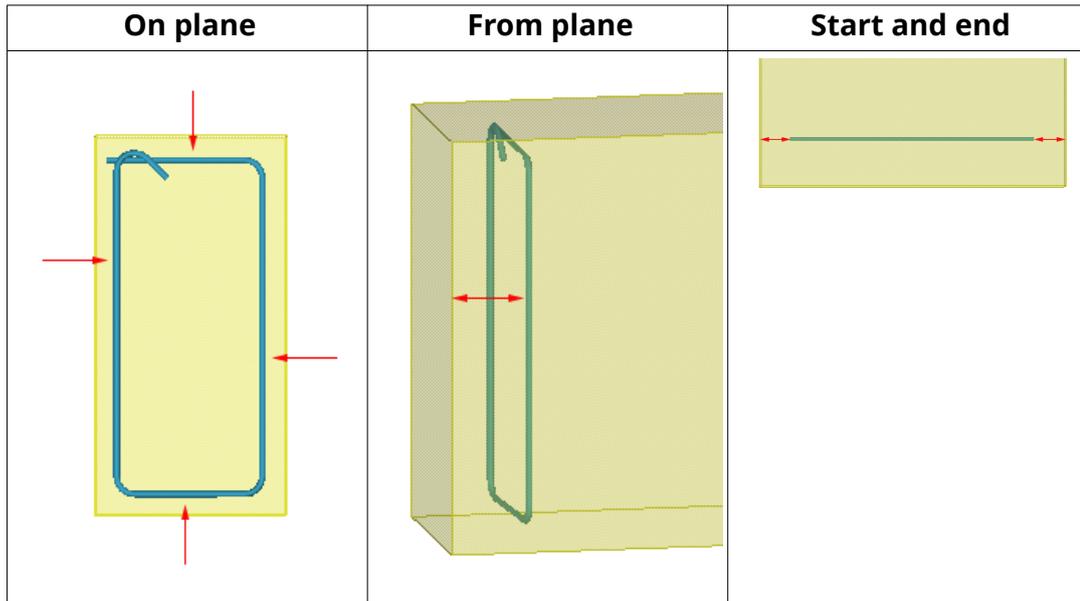
Define the reinforcement cover thickness

Reinforcing bars need a concrete cover to protect them against harmful elements, such as the weather and fire. When you create single bars, Tekla Structures uses the thickness of concrete cover to determine the position of the bar.

To define the reinforcement cover thickness, do one of the following:

To	Do this
Change the cover thickness using direct modification (page 496)	<ol style="list-style-type: none">1. Ensure that the  Direct modification switch is active.2. Select a single reinforcing bar, a reinforcing bar group, or a mesh.3. Drag a line handle to the desired location. 
Change the cover thickness using the Single rebar , Rebar group , or Rebar mesh properties	<ol style="list-style-type: none">1. Select a single reinforcing bar, a reinforcing bar group, or a mesh.2. Double-click the reinforcement to open its properties.3. Define the reinforcing bar cover thickness in the Cover thickness section. <p>The cover thickness can be defined in three directions:</p> <ul style="list-style-type: none">• On plane, that is, the distance from beam's bottom, top, and side surfaces to the bar. <p>You can enter several values. Enter the values in the order you pick the points to create the bar. If you enter less values than there are bar legs, Tekla Structures uses the last value for the remaining legs.</p>

To	Do this
	<ul style="list-style-type: none"> • From plane, that is, the distance from the end surface of the beam to the bar. If the reinforcing bar is outside the part, enter a negative value in the On plane and/or the From plane boxes. • In the longitudinal direction of the bar, that is, start and end. To define the length of an ultimate leg of a bar, use the Leg length option and the Snap to nearest points switch. Then pick anywhere on a part edge or line to indicate the direction for the bar leg. <p>4. Click Modify.</p>
Change the default cover thickness of the rebar sets in a model	<ol style="list-style-type: none"> 1. On the File menu, click Settings --> Options to open the Options dialog box. 2. Go to the Reinforcement settings. 3. Modify the settings and click OK. 4. To apply the changes to all or selected existing rebar sets in the model, on the Concrete tab, click Rebar set --> Regenerate rebar sets.
Change the cover thickness of the rebar sets in an individual concrete part	<ol style="list-style-type: none"> 1. Double-click a concrete part to access its properties. 2. In the Custom properties section, click the More button. 3. Go to the Concrete cover tab. 4. Define the cover thickness at the top, bottom, and side faces of the part. 5. Click Modify.



See also

[Create a reinforcing bar group \(page 452\)](#)

[Create a reinforcing bar group using Rebar shape catalog \(page 454\)](#)

[Create a rebar set \(page 428\)](#)

[Modify reinforcement \(page 477\)](#)

Use adaptivity to modify a reinforcement

Reinforcement follows the shape of the part also when the reinforcement handles are located on the face or edge of the part.

The following types of adaptivity are available:

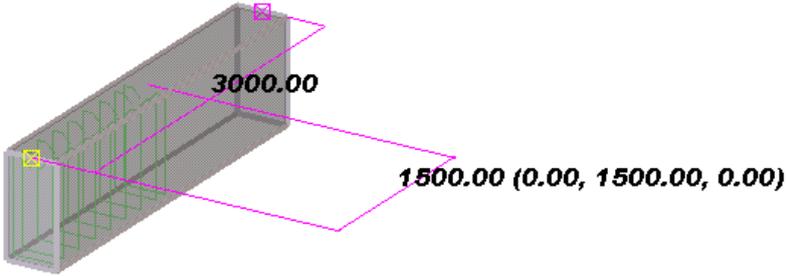
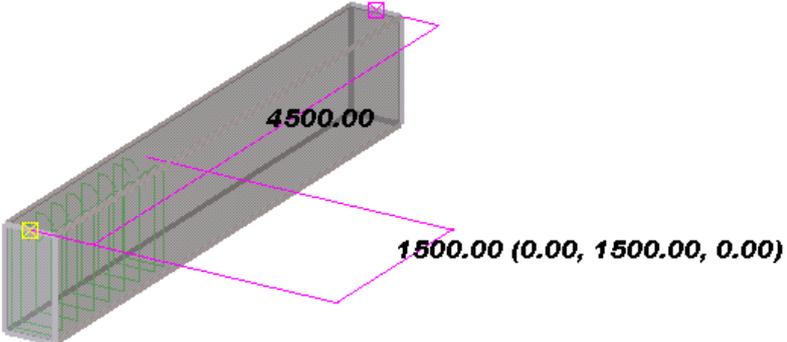
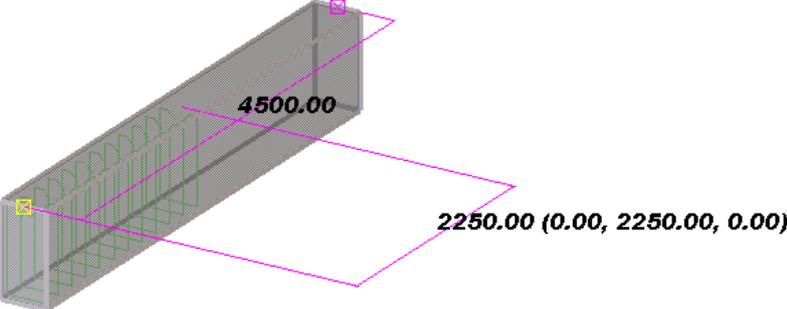
- Fixed adaptivity: handles retain their absolute distances to the nearest part faces.
 - Relative adaptivity: handles retain their relative distances to the nearest part faces in relation to the part's overall size.
1. Select a reinforcement.
 2. Right-click and select **Adaptivity** and then one of the adaptivity options from the pop-up menu.

When a part is modified, Tekla Structures handles the reinforcement according to the adaptivity selection.

TIP To modify the general adaptivity settings, click **File menu** --> **Settings** --> **Options** --> **General**.

You can also modify the adaptivity settings for each part separately. These modifications override the general settings in the **Options** dialog box.

Adaptivity examples

Reinforcing bars in their original position	
Fixed adaptivity	
Relative adaptivity	

See also

[Check the validity of reinforcement geometry \(page 514\)](#)

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

Attach a reinforcement to a concrete part

When you create a reinforcement, Tekla Structures automatically attaches the reinforcement to the part for which you are creating the reinforcement. If needed, you can attach a reinforcement to a concrete part also manually. The attached reinforcing bars follow the part or cast unit when it is moved, copied, or deleted.

NOTE You must attach reinforcement to a part to have Tekla Structures merge automatic reinforcing bar marks in drawings.

1. On the **Concrete** tab, click **Rebar** and select **Attach to part**.



2. Select the reinforcement that you want to attach.
3. Select the part to which you want to attach the reinforcement.
The reinforcement is attached to the part.

Detach a reinforcement from a concrete part

If needed, you can detach a reinforcement from a concrete part.

1. On the **Concrete** tab, click **Rebar** and select **Detach from part**.
2. Select the reinforcement that you want to detach.
The reinforcement is detached from the part.

TIP Alternatively, you can use the pop-up menu. This is how you attach and detach rebar sets, or bars in rebar sets, for example.

1. Select the reinforcement that you want to attach or detach.
 2. Right-click and select **Attach to Part** or **Detach from Part**.
 3. If you are attaching the reinforcement, select the part to which you want to attach the reinforcement.
-

See also

[Modify reinforcement \(page 477\)](#)

Check the validity of reinforcement geometry

Reinforcement creation or modification can result in invalid reinforcement geometry. For example, too big bending radius can cause invalid reinforcement geometry. If a model contains a reinforcement with invalid

geometry, the reinforcement is not shown in the drawings. The reinforcement becomes visible and the drawings are updated when the geometry is corrected.

NOTE Reinforcement geometry validity check does not work with [circular \(page 463\)](#) or [curved \(page 461\)](#) reinforcing bar groups.

1. On the **File** menu, click **Diagnose & Repair** and in the **Model** area, click **Diagnose**.

2. Check the results.

If there are inconsistencies in the geometry, Tekla Structures displays a warning message, and draws a thin line between the reinforcement handles to show the invalid geometry.

You can correct the reinforcement geometry by selecting the line and modifying the reinforcement properties.

See also

[Modify a single reinforcing bar, bar group, or mesh \(page 496\)](#)

Split and splice a reinforcement

You can split long reinforcing bars and bar groups that exceed the stock length, and create splices in split locations.

Use the **Automatic splicing tool** macro to split and splice reinforcement that exceeds the stock length. You can first check the length of the reinforcing bars in the model according to the manufacturer. You can then define the portion of the reinforcement to be split and spliced in the same cross section, and the location, symmetry, type, and length of the splices.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Automatic splicing tool** to start the macro.
4. In the **Automatic Splicing Tool** dialog box:

- a. Select the manufacturer of the reinforcement.

The maximum bar lengths and lap lengths are then listed by the grade and size of the bar.

If needed, you can define the length information in the `AutomaticSplicingTool_Manufacturers.dat` file. You can copy the default file from `..\ProgramData\Trimble\Tekla`

Structures\<>version>\environments\common\system , edit it, and save it to your project or firm folder.

- b. For the bar grades and sizes that are not listed in the AutomaticSplicingTool_Manufacturers.dat file, use the **Maximum length for unspecified stock** box to define the maximum reinforcing bar length after which the bars are split and spliced.
- c. To check if the length of the reinforcing bars exceeds the maximum length, click one of the buttons next to **Perform check to:**
 - To check all reinforcement in the model, click **All**.
 - To check specific reinforcement, select the reinforcement in the model using the **Select objects in components** switch , and then click **Selected**.

Tekla Structures lists the reinforcing bars that are longer than the maximum length under **Longer rebars** on the right side of the dialog box.

When you select a row in the **Longer rebars** list, Tekla Structures highlights the corresponding reinforcement in the model.

- d. Define which proportion of the reinforcement can be spliced in the same cross section.
- e. Define the symmetry that is applied when the reinforcing bars are spliced.
- f. Define the offset of the splice center point.
- g. Define the minimum longitudinal distance between two parallel bar splices.
- h. Select the splice type.

You can create lap splices, coupler splices, or welded splices.
- i. For lap splices, define the default lap length in relation to the nominal bar diameter.

This value will be used if there is no lap length defined for a bar grade and size in the AutomaticSplicingTool_Manufacturers.dat file.
- j. For lap slices, define whether the lapping bars are on top of each other or parallel to each other.
- k. To split and splice reinforcement, click one of the buttons next to **Perform split and splice to:**
 - To splice all reinforcement in the model, click **All**.
 - To splice specific reinforcement, select the reinforcement in the **Longer rebars** list, or in the model using the **Select objects in components** switch , and then click **Selected**.

See also

[Create a reinforcement splice \(page 476\)](#)

Assign running numbers to reinforcement

You can assign running numbers to reinforcement in cast units. You can then use the running numbers in addition to or instead of the position numbers in reinforcement marks and tables in drawings, and in reports.

Use the **Rebar sequence numbering** macro to assign cast unit specific running numbers (1, 2, 3...) to the reinforcement in the model. Running numbers are unique inside each cast unit. The macro does the following:

- Updates the position numbers of the modified model objects using the **Number modified objects** command in **Drawings & reports** --> **Perform numbering**.
- Assigns running numbers to the reinforcing bars, reinforcing bar groups, and reinforcement meshes in the model.
- Saves a running number as the user-defined attribute **Rebar sequence number** (`REBAR_SEQ_NO`) of each bar, group, or mesh.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Rebar sequence numbering** to start the macro.
4. To show the running numbers in drawings and reports, use the user-defined attribute `REBAR_SEQ_NO`.

See also

[Number reinforcement \(page 639\)](#)

Classify reinforcement to layers

To be able to show in drawings what is the order of different reinforcement layers near a surface of a concrete part, you need to classify reinforcement in the model. You can do this by using the **Rebar classification** macro.

Rebar classification classifies the reinforcing bars and reinforcement meshes by their order of depth in concrete slabs and panels. The reinforcing bars and

meshes get an attribute indicating the layer where they are placed inside the concrete part.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Rebar classification** to start the macro.
4. In the **Rebar classification** dialog box:
 - a. Enter the prefixes that you want to use for the reinforcement layers near the top, bottom, front, and back surfaces of the concrete parts.
 - b. Select whether you want to classify **All objects** or **Selected objects**.
If you select **Selected objects**, select the reinforcement or the concrete parts containing the reinforcement that you want to classify.
 - c. Click **Preview** to view the properties of the reinforcement in each layer.
The layers are named using the relevant surface prefix and numbered starting from the surface.
 - d. If you do not want to classify a reinforcement, select it from the list and click **Delete item**.
 - e. To save the classification attributes of the reinforcement, do one of the following:
 - Click **Modify** to also keep the **Rebar classification** dialog box open.
 - Click **OK** to also close the **Rebar classification** dialog box.
5. In a drawing, run the **Rebar layering marker** macro to create layer-specific markers for reinforcement.

How to calculate the reinforcing bar length

You have three options how to calculate the reinforcing bar length in Tekla Structures:

- Along center line, which is the default method
- As a sum of leg lengths
- Using a formula

Along center line

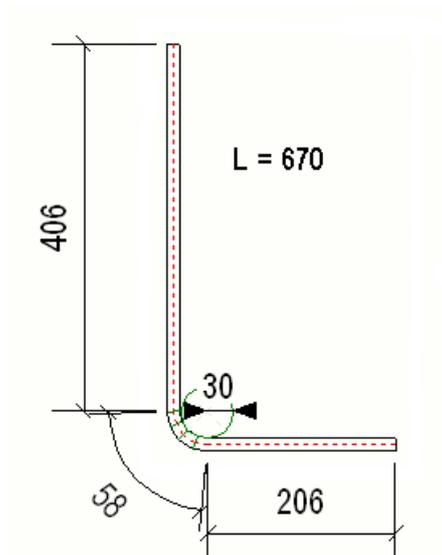
The center line length calculation is used by default when `XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT` is set to `FALSE` in **File menu --> Settings --> Advanced options**.

Center line length calculation uses the actual reinforcing bar diameter by default.

In the example below, the center line length is calculated as follows: $450 - (30 + 14) + 2 * 3.14 * (30 + 14 / 2) * 1 / 4 + 250 - (30 + 14) = 670.1$

where

- 30 = bending radius
- 14 = actual diameter (12 is nominal)

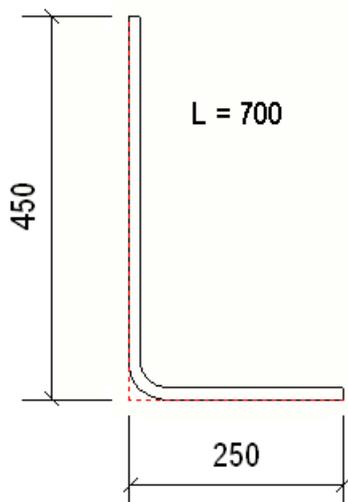


Sum of leg lengths (SLL)

The sum of leg length calculation is based on the dimensions of the straight legs and it does not take the bending radius into account.

This calculation is used when `XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT` and `XS_USE_USER_DEFINED_REBARSHAPERULES` are set to `TRUE` in **File menu --> Settings --> Advanced options**.

In the example below, the reinforcing bar length is $450 + 250 = 700$



If the length value is shown as zero in reports and inquiries, you need to define the length in **Rebar Shape Manager** for each shape.

To define the length in **Rebar Shape Manager**:

1. In **Bending schedule fields**, right-click in the **L** cell and select **SLL (Sum of leg lengths)** from the pop-up menu.
2. Click **Update**.
3. Click **Save**.

Using a formula

You can also use a formula in **Rebar Shape Manager** to calculate the reinforcing bar total length.

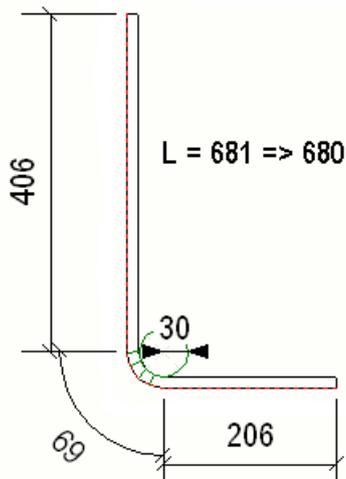
You need to set `XS_USE_USER_DEFINED_REBAR_LENGTH_AND_WEIGHT` and `XS_USE_USER_DEFINED_REBARSHAPERULES` to `TRUE` **File menu** --> **Settings** --> **Advanced options**.

For example, to take the bending radius into account and to calculate the length along the reinforcing bar outer surface, do the following:

1. In **Bending schedule fields**, right-click in the **L** cell and select **(formula)** from the pop-up menu.
2. Enter the following formula for the length calculation: $S1 + S2 + 2 * 3.14 * (RS + DIA) * 1/4$

where

- $S1$ = straight leg length 1 (406)
- $S2$ = straight leg length 2 (206)
- RS = rounding radius (30)
- DIA = actual diameter (14)



Accuracy

The accuracy of reinforcing bar length is defined in the `rebar_config.inp` file. The values can vary in each environment.

For example, the values shown below are from a `rebar_config.inp` file. In the default environment the file is located in `.. \ProgramData\Trimble \Tekla Structures\<version>\Environments\default\system\` folder.

The following settings define the accuracy and rounding for the leg lengths:

- `ScheduleDimensionRoundingAccuracy=1.0`
- `ScheduleDimensionRoundingDirection="DOWN"`

The following settings define the accuracy and rounding for the total reinforcing bar length:

- `ScheduleTotalLengthRoundingAccuracy=10.0`
- `ScheduleTotalLengthRoundingDirection="DOWN"`

Note that also `XS_USE_ONLY_NOMINAL_REBAR_DIAMETER` affects the reinforcing bar length calculation.

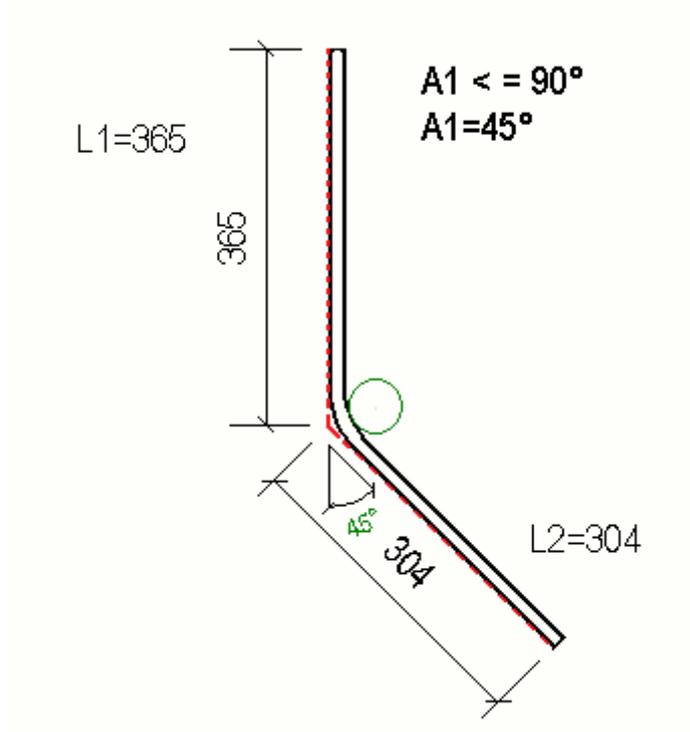
See also

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

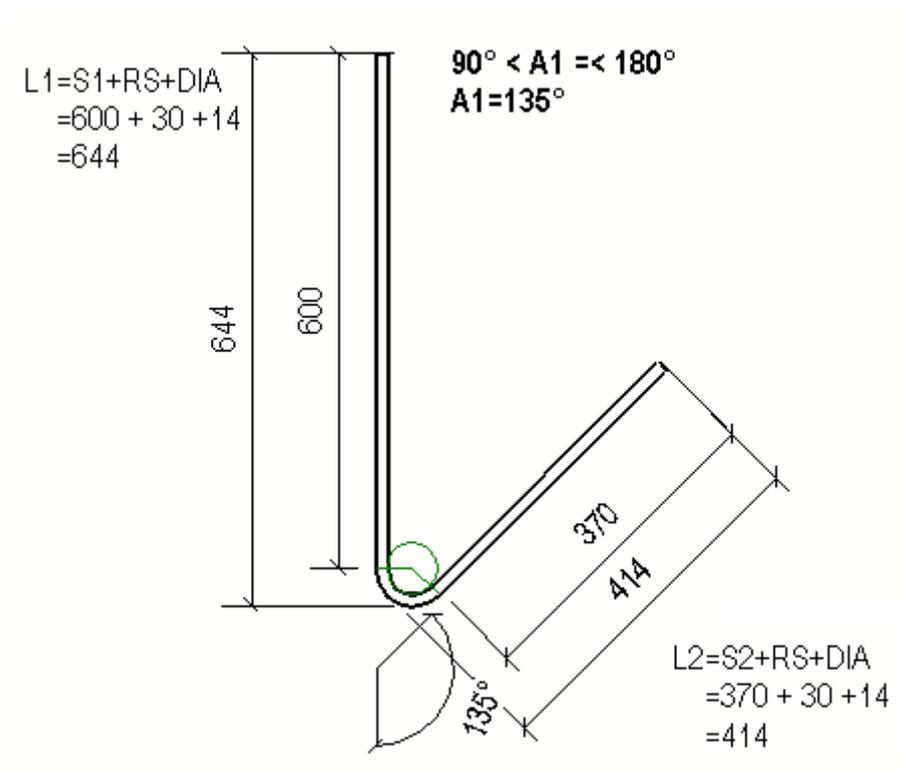
How to calculate the reinforcing bar leg length

How the reinforcing bar leg length is calculated depends on the angle between the bar legs.

- When the angle is $\leq 90^\circ$, the length is measured to the extension of a leg along the outer edge



- When the angle is $> 90^\circ$ and $\leq 180^\circ$, the tangential length is used



The leg lengths are calculated using **Rebar Shape Manager**, where

- S1 = straight portion of a bar for the first segment
- S2 = straight portion of a bar for the second segment
- A1 = bending angle measured between the extension of the first leg and the second leg. The angle is 0° if the second segment continues to the same direction as the first segment (the bar is straight)
- L1 = leg length for the first reinforcing bar segment
- L2 = leg length for the second reinforcing bar segment
- RS = bending radius
- DIA = actual diameter of the reinforcing bar

See also

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

[Reinforcing bar and bar group properties \(page 927\)](#)

Reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns shape codes to them. Tekla Structures then uses the shape and dimension information in bending schedules, pull-out pictures, templates and reports.

Tekla Structures includes two methods for shape recognition.

<p>User-defined bending shape definitions.</p>	<p>These definitions are created with Rebar shape manager (page 524) and saved in the RebarShapeRules.xml file.</p> <p>The file is located in . . . \ProgramData\Trimble\Tekla Structures\<<version> \environments\<<environment> \system folder.</p>
<p>Tekla Structures internal, hard-coded bending type definitions.</p>	<p>These internal bending types (page 535) of reinforcing bars are mapped to area-specific reinforcing bar bending type codes in the rebar_schedule_config.inp file.</p> <p>The file is located in . . . \ProgramData\Trimble\Tekla Structures\<<version> \environments\common\system folder.</p>

See also

[Reinforcement in templates \(page 559\)](#)

Rebar shape manager in reinforcement shape recognition

You can define your own bar bending shapes and assign shape codes with **Rebar shape manager**, and thus increase the amount of recognized bar shapes. User-defined bending shapes are useful when Tekla Structures does not recognize the bending shape and assigns the UNKNOWN bending type to the shape.

Rebar shape manager is meant for users who need to customize the bending shapes according to company or project requirements.

With **Rebar shape manager** you can:

- Customize the existing bending shapes and [create new bending shapes \(page 524\)](#).
- [Establish your own rules \(page 527\)](#) for defining the bending shapes.
- Customize your own dimension mappings that are used in [templates and reports \(page 533\)](#).
- Import and export user-defined bending shapes.
- Use user-defined bending shapes in bending schedules and pull-out pictures.

NOTE **Rebar shape manager** is a tool for recognizing reinforcing bar shapes. You cannot control the reinforcing bar creation properties, such as cover thickness, reinforcing bar grade, or size, with this tool.

See also

[Tips for reinforcement shape recognition in Rebar shape manager \(page 534\)](#)

Define reinforcing bar bending shapes in Rebar shape manager

With **Rebar shape manager** you can establish your own rules for defining the bending shapes. When you define your own reinforcing bar bending shapes and shape codes, an .xml file called `RebarShapeRules.xml` is created in the current model folder.

In addition, Tekla Structures installation contains by default another .xml file called `RebarShapeRules.xml`. This file contains the most typical bending shapes in your environment, and it is located in the `.. \ProgramData \Trimble \Tekla Structures \<version> \environments \<environment> \system folder`.

When you define new shapes, the shapes in the default `RebarShapeRules.xml` rule file can be appended to your own shapes. Tekla Structures reads valid `RebarShapeRules.xml` rule files in the model, project, firm, and system folders in that order. When the shape codes and report field

values are applied, Tekla Structures uses the first matching shape in a `RebarShapeRules.xml` file which is found first based on the search order. All the found bending shapes are displayed in **Rebar shape manager**.

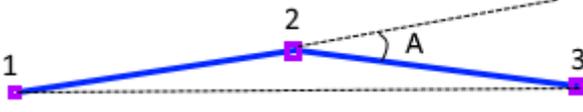
1. Select reinforcing bars in the model.
2. On the **File** menu, click **Editors** --> **Rebar shape manager**.

Rebar shape manager opens and lists the selected reinforcing bars in the **Model rebars** list.

Alternatively, you can first open **Rebar shape manager** and then select reinforcing bars in the model. Click **Get selected** to add the reinforcing bars to the **Model rebars** list.

- The **Model rebars** list shows the ID and the shape code of the selected reinforcing bars.
- The **Shape catalog** list shows the shapes that exist in the default `RebarShapeRules.xml` rule file.
- The **Tolerances** tab shows the tolerances used when the bending shape rules are compared.

3. Select one unknown shape from the **Model rebars** list.
4. To define the needed information for a bending shape, do the following:

To define	Do this
Tolerances	<p>Enter tolerance values for the following measurements:</p> <ul style="list-style-type: none"> • Dimension (leg lengths and other distances) • Angle (bending and twist angles) • Radius (bending radiuses) • Extra point shortening • Extra point max angle <p>The tolerances Extra point shortening and Extra point max angle are used together for defining whether two bar legs ('1-2' and '2-3' in the image below) can be considered as one leg ('1-3').</p>  <p>In the Extra point shortening box, define the maximum allowed difference between the distances '1-3' and '1-2' + '2-3'.</p>

To define	Do this
	<p>In the Extra point max angle box, define the maximum allowed angle ('A' in the image) between the two bar legs.</p> <p>Note that the tolerance values are stored along with the rule file <code>RebarShapeRules.xml</code>, and thus the tolerances are specific to each rule file.</p>
Shape code	<p>Enter a shape code for an unknown shape.</p> <p>Note that multiple reinforcement shapes that are variants of the same shape can have the same Shape code but different Bending shape rules.</p>
Bending shape rules	<p>If the bending shape rules that Rebar shape manager defines automatically are not sufficient to distinguish certain bending shapes, you can manually add (page 527) new bending shape rules.</p> <p>Add or delete a bending shape rule by clicking the Add and Delete buttons on the right.</p> <p>Use the Reset button to restore the original values.</p>
Check hooks	<p>Select the check box if you want to define different shape codes or bending schedule fields for two bars that otherwise have exactly the same geometry, but one bar has hooks and the other one does not.</p> <p>If you select the check box, hooks are considered as hooks. If you clear the check box, hooks are considered as normal legs.</p> <p>Note that the Check hooks option works independently from the advanced option <code>XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION</code>, and it allows bars that have different hooks to have different shape code or schedule fields regardless of the value of the advanced option.</p>
Update	<p>Update the existing shape code definition of the selected reinforcing bar.</p> <p>You can update the definition if you have modified the shape code, bending shape rules or the contents of bending schedule fields.</p>
Bending schedule fields	<p>Define the content (page 533) for a bending schedule. Right-click a field to select a bending shape property or to enter a formula.</p> <p>The names of the Bending schedule fields (A, B, and so on) are used in templates and reports. To make sure that old reports also work correctly, we recommend that you use the same <code>DIM_XX</code> fields as in the <code>rebar_schedule_config.inp</code> file.</p>

To define	Do this
Schedule fields...	<p>Click the Schedule fields button to add, remove or change the order of the available schedule fields. If needed, you can reset the schedule fields to original defaults.</p> <p>If you change the set of available schedule fields and update an existing shape, the old schedule fields which do not exist anymore will be cleared. Therefore we recommend that you do not to remove any of the default schedule fields unless you are sure they have not been used in any of the existing shapes.</p> <p>You can change the names of the existing schedule fields or, if you add new fields, give names to them. To use the fields in templates and reports, use the <code>DIM_XX</code> or <code>ANG_XX</code> fields, and replace <code>xx</code> with the schedule field name.</p>

- When you have finished defining the new shape, click **Add** to add the bending shape definition to the `RebarShapeRules.xml` file.

To enable the **Add** button you need to change the bending shape rule, enter a shape code or select the **Check hooks** check box.

- Click **Save** to save the `RebarShapeRules.xml` file.

By default, the file is located in the current model folder.

When you create, for example, a bending schedule, Tekla Structures uses the updated bending shape information, recognizes the added bending shape and assigns a correct shape code to it.

NOTE Rebar shape manager is able to recognize bending shapes regardless of the modeling direction of the bars. This means that the modeling direction has no effect on the shape definition and the shape code.

When defining the bending shapes, the start or the end of modeling direction is always sorted based first on bending angles, then on twist angles, and finally on leg lengths. However, bending radius is not taken into account in the sorting. This means that radius 1 may not always be less than radius 2 or vice versa.

See also

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

Add new bending shape rules manually in Rebar shape manager

In some cases, the bending shape rules that are defined in **Rebar shape manager** are not sufficient to distinguish certain bending shapes. If needed, you can manually add new bending shape rules for reinforcing bars in **Rebar shape manager**.

1. In **Rebar shape manager**, click **Add** next to the **Bending shape rules** list.
2. In the **New bending rule** dialog box, select the options from the lists to define the new rule.

The content of the lists depends on the shape and the bending of the reinforcing bar.

3. Click **OK** to add the new rule to the **Bending shape rules** list.

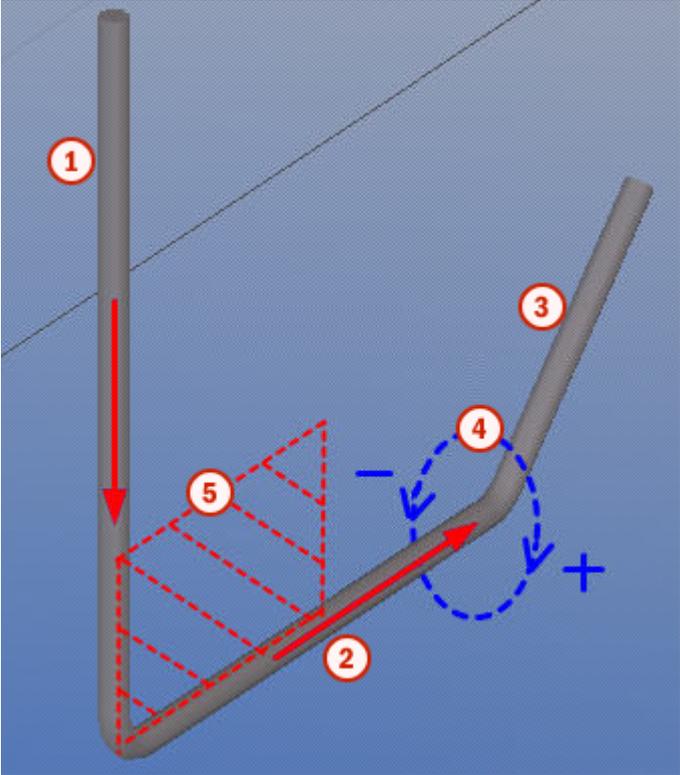
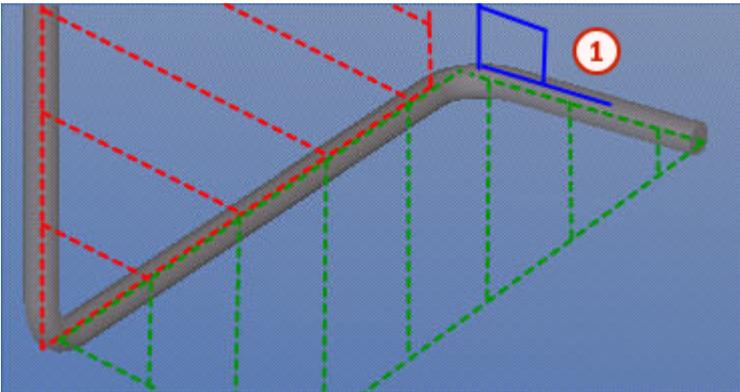
The **OK** button is enabled only when the rule is valid.

Bending shape rule settings

All the rule options are available in the **New bending rule** dialog box, even though only certain selections are valid, depending on the type of the conditions used. The left and right condition of a rule need to be of the same type. The values in the parentheses are the values that were used to create the bar shape.

Use the **New bending rule** dialog box in **Rebar shape manager** to manually define rules for reinforcing bar bending shapes.

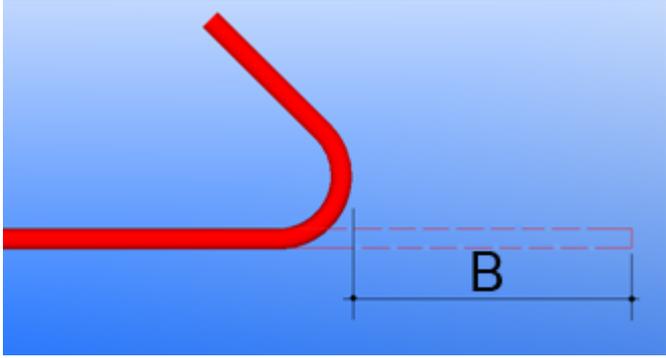
Option	Description
Angle (A)	Bending angle between the legs. Bending angle is always between 0 and +180 degrees. The angle cannot be negative.
Twist angle (T)	Rotation angle of a plane that has been created by two bars of consecutive lengths. The plane is rotated around the axis of the last bar creating the plane. For bars where all the legs lie in the same plane, the twist angle is either 0 degrees or +180 degrees. If the bar twists out of the plane, i.e. the bar is in 3D, the twist angle is between -180 and +180 degrees.

Option	Description
	 <p>The diagram shows a 3D view of a bent reinforcement bar with three legs. Leg 1 is vertical, Leg 2 is horizontal, and Leg 3 is angled upwards. A red arrow on Leg 1 points downwards. A red arrow on Leg 2 points to the right. A blue dashed circle with a minus sign indicates a negative twist angle at the bend between Leg 2 and Leg 3. A blue dashed circle with a plus sign indicates a positive twist angle at the bend between Leg 1 and Leg 2. Red dashed lines form a plane through Leg 1 and Leg 2, labeled with a circled 5. A circled 4 is placed at the bend between Leg 2 and Leg 3.</p> <ol style="list-style-type: none"> 1. Leg1 2. Leg 2 3. Leg 3 4. Twist angle direction 5. Plane created by legs 1 and 2
Twist angle example	<p>The twist angle between two planes is +90 degrees. The planes are created by legs 1 and 2, and legs 2 and 3.</p>  <p>The diagram shows the same bent reinforcement bar. Red dashed lines form a plane through Leg 1 and Leg 2. Green dashed lines form a plane through Leg 2 and Leg 3. A blue square with a circled 1 is drawn at the bend between Leg 2 and Leg 3, indicating a 90-degree angle between the two planes.</p> <ol style="list-style-type: none"> 1. Twist angle: +90 degrees

Option	Description
Radius (R), (RX)	Bending radius of the bending. (RX) Radius * is the value of the bending radius when all the bendings have equal radius. Otherwise the value is zero (0). Radius * = Radius 1 ensures that all the bendings have been created using the same radius.
Bending length (BL)	Center line length of the bending.
Straight length (S)	Straight length between the start and the end of adjacent bendings. The rule is generated only when there is no straight part, for example, Straight length 2 = 0.
Leg length (L)	Length of the leg.
Leg (V)	Leg direction as a vector value.
Bar diameter (DIA), (DIAX)	Diameter of the reinforcing bar.
Nominal diameter (NDIA), (NDIAX)	Nominal diameter of the reinforcing bar.
Center line length (CLL)	Leg length according to the center line.
Sum of leg lengths (SLL)	Sum of all leg lengths.
Reversed	Reversed reinforcing bar. You can use Reversed to have additional bending shape rules and/or formulas for the schedule fields. When used in a rule, you can have separate definitions in shape code and/or schedule fields for reinforcing bars that have different modeling order of the points. When used as a part of a formula, you can eliminate the automatic normalization of the modeling order of the points. For example, a formula <code>if (REVERSED) then L2 else L3 endif</code> forces the content of the field to show the desired leg length depending on the order of the points or legs.
Arc inner radius (RI)	Inner radius of the arc.
Arc outer radius (RO)	Outer radius of the arc.
Arc angle (AA)	Angle of the arc.
Arc length (AL)	Length of the arc.

Option	Description
Curve width (CW)	Extreme width of the curved bar.
Curved height (CH)	Extreme height of the curved bar.
RFACTOR	Relative radius.
LFACTOR	Relative length.
Spiral rounds (SR)	Rounds of the spiral bar.
Spiral pitch (SP)	Pitch of the spiral bar.
Spiral length (SL)	Distance between the reference points of the spiral bar.
Spiral total length (STL)	Total length of the spiral bar when the bar is installed on site.
Standard radius (RS)	Standard minimum bending radius. The bending radius depends on the size and the grade of the bar.
Weight per length (WPL)	Weight per leg length.
Leg distance from leg (D)	Similar to Point/arc distance off from leg (H) . The difference is that Point/arc distance off from leg (H) considers the bending radius, whereas Leg distance from leg (D) is measured from the sharp corner. When the legs are parallel, both Leg distance from leg (D) and Point/arc distance off from leg (H) give the same result.
Point/arc distance along leg (K)	Distance parallel to a leg from outer edge to outer edge, or tangential to the bending. The distances are positive or negative depending on the leg direction. Example:

Option	Description
<p>Point/arc distance off from leg (H)</p>	<p>Distance perpendicular to a leg from outer edge to outer edge, or tangential to the bending.</p> <p>The distances are positive or negative depending on the leg direction.</p> <p>Example:</p>
<p>SH SHA SHR SHS SHLA SHLB EH EHA EHR EHS</p>	<p>Start and end hook properties.</p> <p>Use method A or B for the hook length calculation:</p>

Option	Description
EHLA EHLB	
Constant angle	Constant value of the angle. Enter the value in the rightmost box.
Constant radius	Constant value of the radius. Enter the value in the rightmost box.
Custom properties, template attributes, user-defined attributes	The custom properties, template attributes, and user-defined attributes defined in the <code>RebarShapeManager.CustomProperties.dat</code> file appear at the end of the list and can be used like any other option.

See also

[Define content for templates and reports in Rebar shape manager \(page 533\)](#)

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

Define content for templates and reports in Rebar shape manager

Use the **Bending schedule fields** in **Rebar shape manager** to define the content for templates and reports. Each of the **Bending schedule fields** cells can contain a shape property or a formula.

When you right-click in a **Bending schedule fields** cell, you can:

- Select a shape property from the list. The content of the list depends on the reinforcing bar geometry.
- Select the **(empty)** option to clear the content of the current cell.
- Select the **(formula)** option to enter a formula. The variables in the formula can either be the shape properties visible in the pop-up menu, or direct references to other non-empty bending schedule field cells.

You can use the same functions in the formulas as in custom components:

- Mathematical functions
- Statistical functions
- String operation

- Trigonometric functions

When you map angles and trigonometric functions in the **Schedule field formula** dialog box, enter the functions (sin, cos, tan) in small letters, for example `sin(A1)`. Capital letters are not recognized, and a blank space will appear in reports.

If you have angles in a formula, the formula needs to be in radians. For example, if you want to subtract 180 degrees from angle A1, enter `A1-PI` in capital letters. If you enter `A1-180` or `A1-pi`, the formula does not work.

Bending schedule fields cell shows the result of a valid formula. If the formula is not valid, a question mark and text describing the error is shown.

NOTE Use the fields **S**, **T**, **U** or **V** to report angles. If you do not use these fields, you need to override the default unit settings in the **Template Editor**.

Example

Formula is `L1+L3+L5-2*DIA`

- L1, L3 and L5 are the leg lengths measured from outer edge to outer edge
- H1 is the total width
- to achieve H1: `L1+L3+L5 minus 2*bar diameter`

See also

[Add new bending shape rules manually in Rebar shape manager \(page 527\)](#)

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

Tips for reinforcement shape recognition in Rebar shape manager

The reinforcing bar shape recognition is based on each shape's bending shape rules. The shapes and their rules are listed in the `RebarShapeRules.xml` file, located in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\system` folder by default. Sometimes a shape matches the rules of two shapes, and Tekla Structures does not recognize the reinforcing bar shape as expected.

NOTE The most convenient way to ensure that a shape is recognized correctly is to modify the shape definition by [adding more rules \(page 527\)](#) to the shape in **Rebar shape manager**.

However, if necessary, you can manually modify the `RebarShapeRules.xml` file and thus affect the shape recognition. When Tekla Structures recognizes the shape, the order of the shapes in the `RebarShapeRules.xml` is important:

- The first shape that matches the rules is the one that Tekla Structures recognizes as the shape. If you need to change the order of the shapes to change how Tekla Structures recognizes the shape, you can do it by manually modifying the `RebarShapeRules.xml` file. If you modify the file and change the order of the shapes, make sure that the structure of the file stays valid.
- You can divide the shape definitions in several `RebarShapeRules.xml` files and different folders. Tekla Structures searches the `RebarShapeRules.xml` file in the model, project, firm, and system folders in that order. Tekla Structures uses the first matching shape in the first `RebarShapeRules.xml` file which is found based on the search order.

See also

[Rebar shape manager in reinforcement shape recognition \(page 524\)](#)

Hard-coded bending type identifiers in reinforcement shape recognition

Tekla Structures recognizes different reinforcing bar bending shapes and assigns bending type identifiers to them.

The bending type identifiers in the table below are internal, hard-coded types of Tekla Structures. The leg dimensions ($D1$, $D2$, and so on) and bending angles ($A1$, $A2$, and so on) of reinforcing bars are Tekla Structures internal dimensions and angles. You can map the Tekla Structures internal bending types, for example, to country- or project-specific bending types, and the Tekla Structures internal dimensions and angles to specific template attributes. You can do this in the `rebar_schedule_config.inp` file.

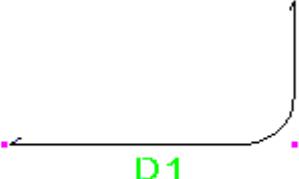
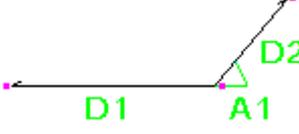
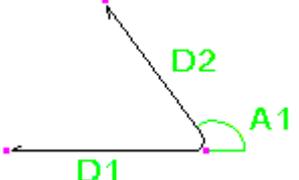
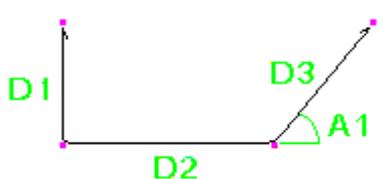
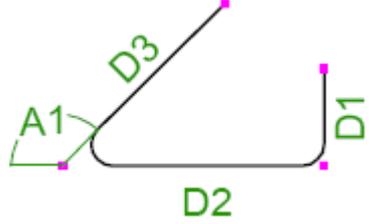
Reinforcing bar bending dimensions are calculated so that the leg dimensions ($D1$, $D2$, and so on) follow the outer edge, or the edge extension, of the reinforcing bar. The total length is calculated according to the center line of the reinforcing bar.

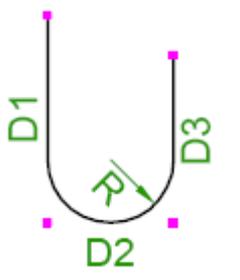
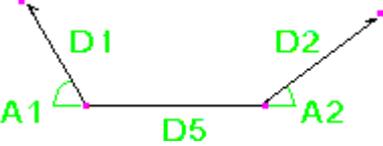
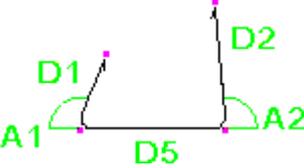
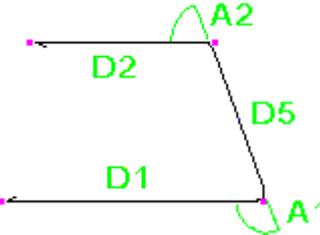
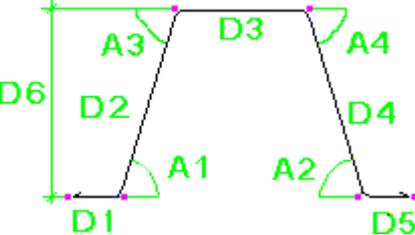
If Tekla Structures does not recognize the shape of a reinforcing bar, it assigns the UNKNOWN bending type to it.

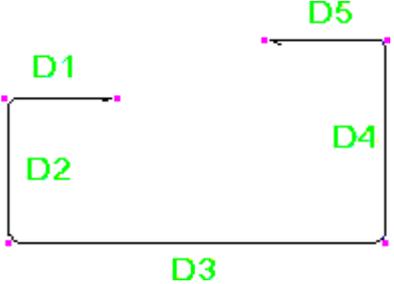
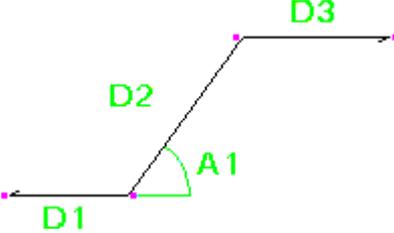
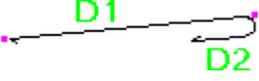
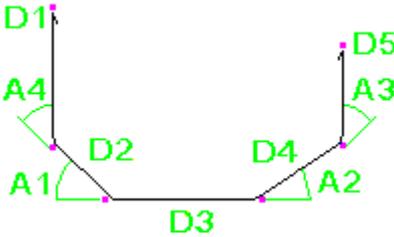
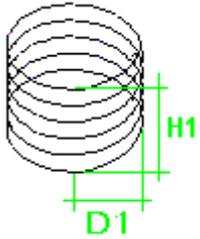
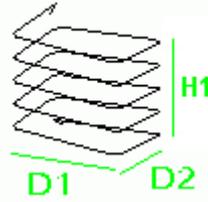
The magenta points in the images in the table below represent the points you pick in the model when you create reinforcing bars.

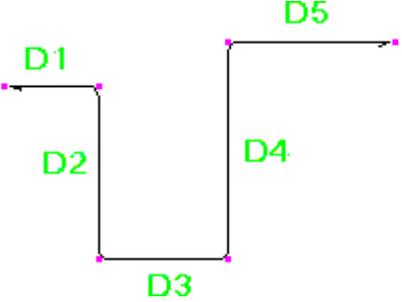
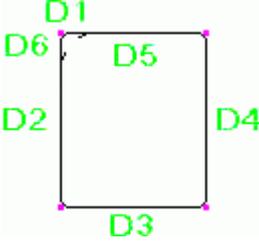
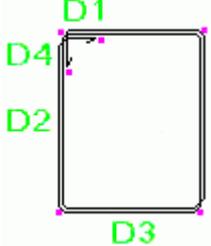
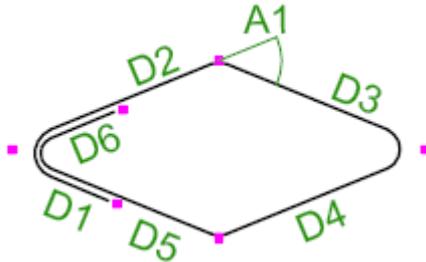
NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use [Rebar shape manager \(page 524\)](#).

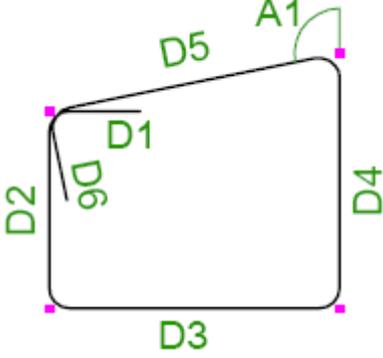
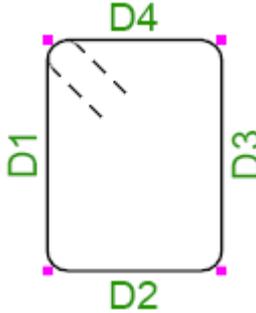
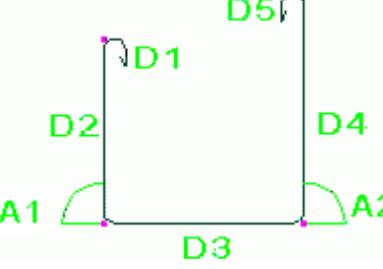
Bending type identifier	Bending shape
1	

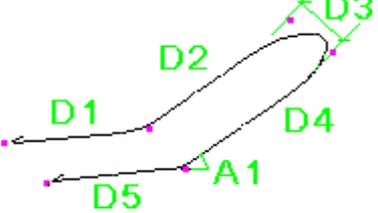
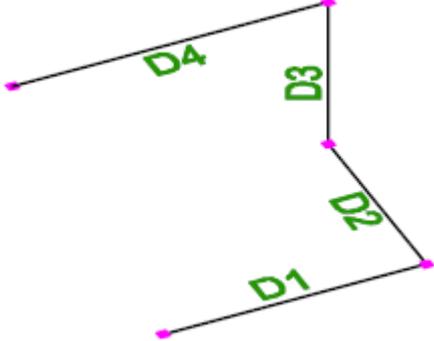
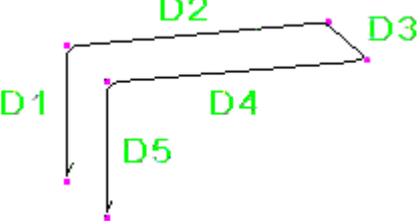
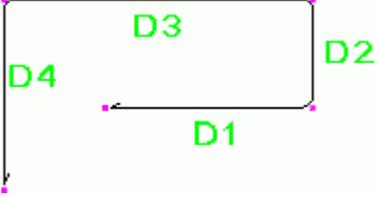
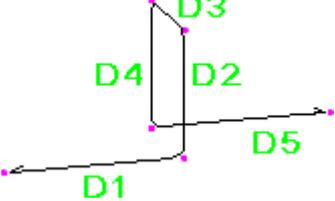
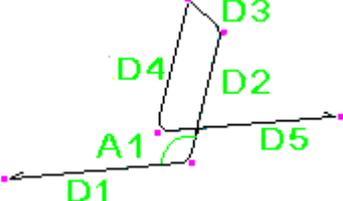
Bending type identifier	Bending shape
2_1	 <p data-bbox="475 555 949 584">Requires standard bending radius.</p>
2_2	 <p data-bbox="475 808 890 837">Non-standard bending radius.</p>
3_1	
3_2	
4	
4_2	
4_3	

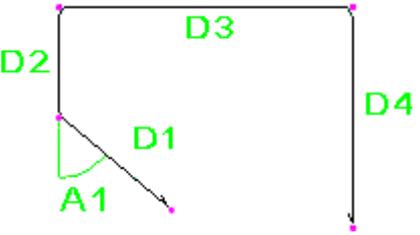
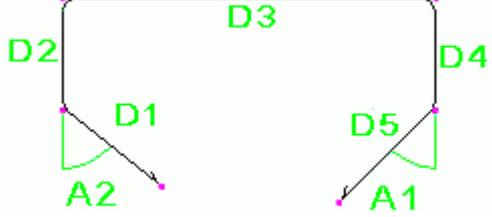
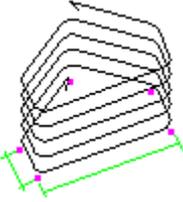
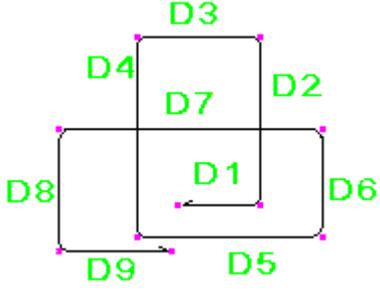
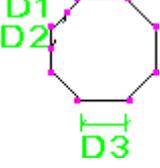
Bending type identifier	Bending shape
4_4	
5_1	
5_2	
5_3	
6_1	
6_2	

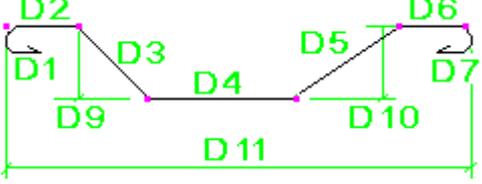
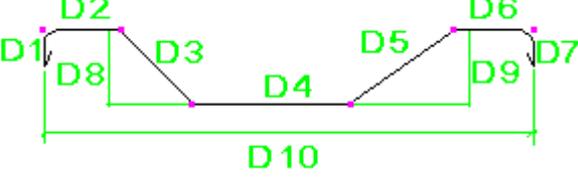
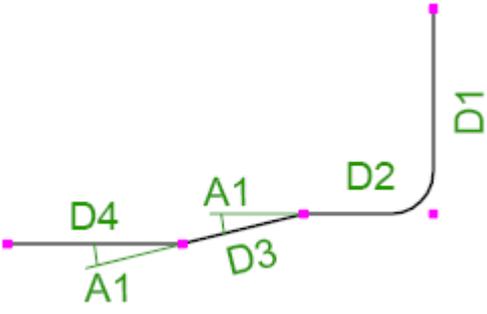
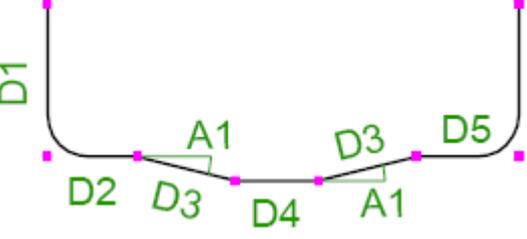
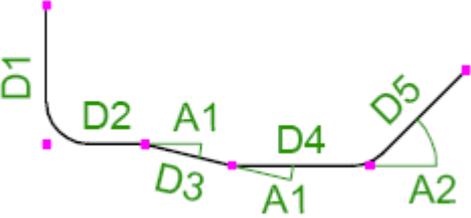
Bending type identifier	Bending shape
7	
8	
9	 <p data-bbox="475 992 842 1025">Requires 180 degree hook.</p>
10	
11	 <p data-bbox="475 1585 1359 1653">D1 = Radius from center of circle to the center line of reinforcing bar.</p>
12	

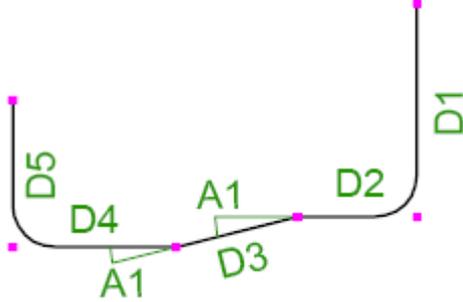
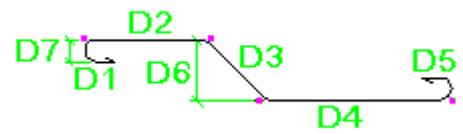
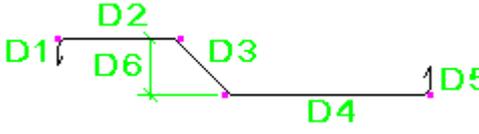
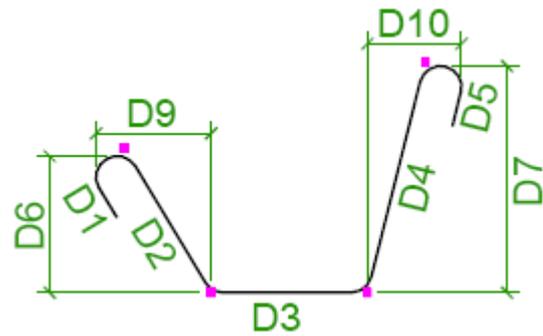
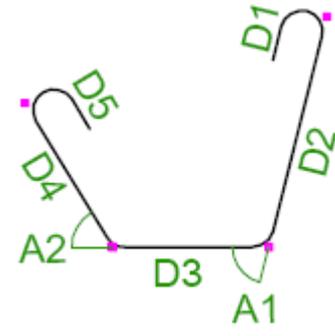
Bending type identifier	Bending shape
13	 <p data-bbox="475 667 1380 734">Can also be modeled using hooks at both ends (i.e. model D1 and D5 using 90 degree hooks).</p>
14	 <p data-bbox="475 1012 1380 1048">Requires 90 degree hooks at both ends.</p>
14_2	
14_3	

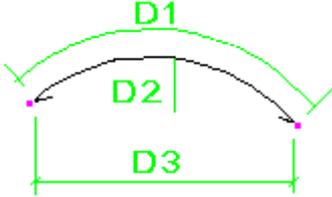
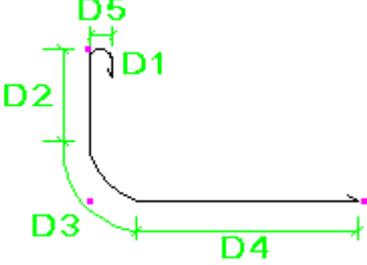
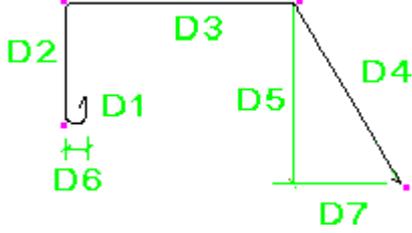
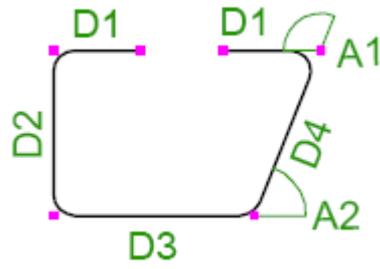
Bending type identifier	Bending shape
14_4	 <p data-bbox="475 739 1021 772">Requires 90 degree hooks at both ends.</p>
14_5	 <p data-bbox="475 1137 1340 1205">Recognized when the start point and end point are in the same location and no hooks are used.</p> <p data-bbox="475 1227 1324 1332">If XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION is set to FALSE, reinforcing bars with hooks (types 14 and 48) are recognized as 14_5.</p>
15	 <p data-bbox="475 1639 877 1673">Requires hooks at both ends.</p>
16_1	

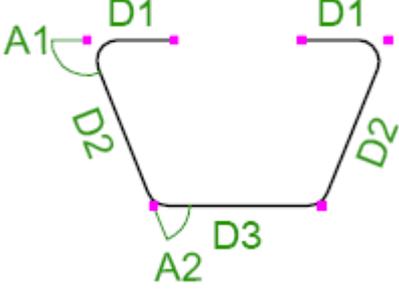
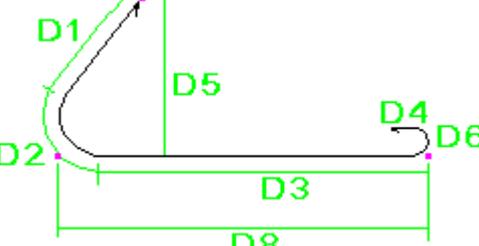
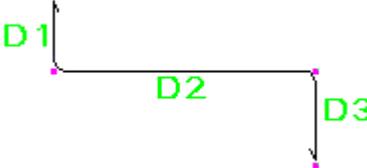
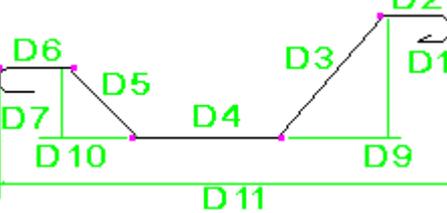
Bending type identifier	Bending shape
16_2	
17	
18	
19	
20_1	
20_2	

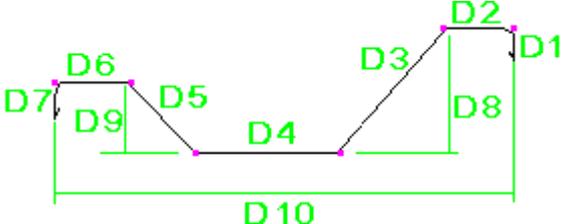
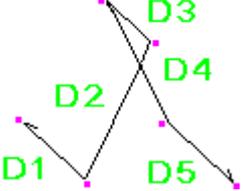
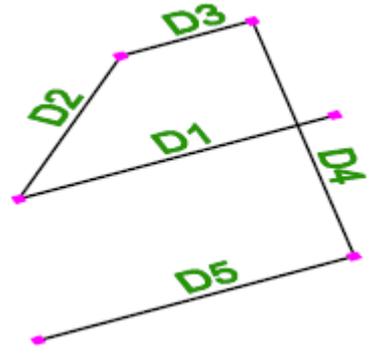
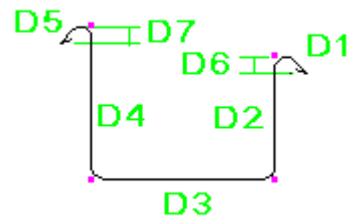
Bending type identifier	Bending shape
21	
22	
23	
24	
25	
26	 <p data-bbox="475 1704 1038 1738">Requires 180 degree hooks at both ends.</p>
27	 <p data-bbox="475 1850 1023 1883">Requires 90 degree hooks at both ends.</p>

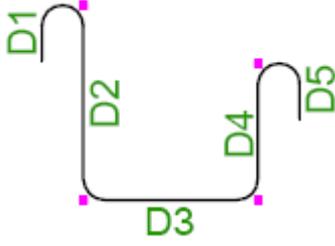
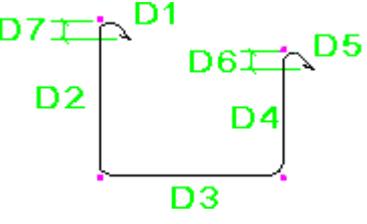
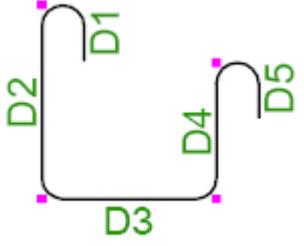
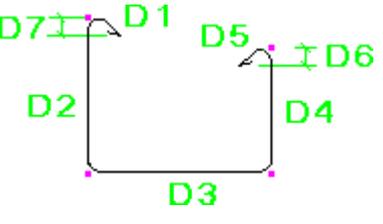
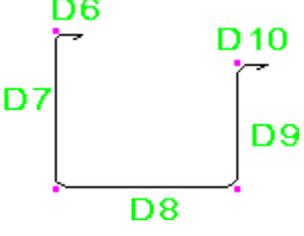
Bending type identifier	Bending shape
28	 <p data-bbox="475 555 1037 589">Requires 180 degree hooks at both ends.</p>
29	 <p data-bbox="475 813 1021 846">Requires 90 degree hooks at both ends.</p>
29_2	
29_3	
29_4	

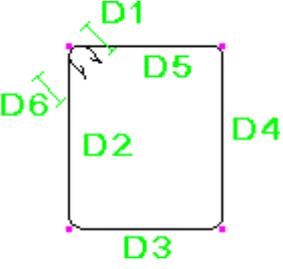
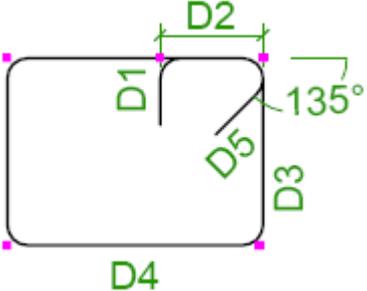
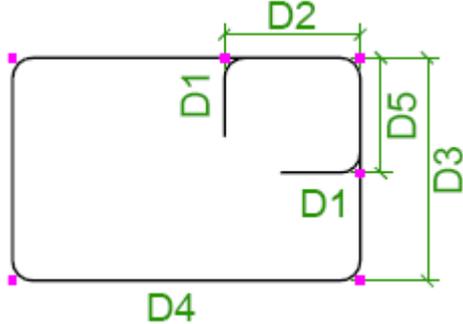
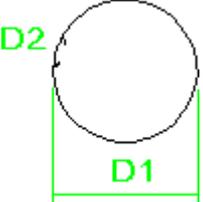
Bending type identifier	Bending shape
29_5	
30	 <p data-bbox="475 851 1037 884">Requires 180 degree hooks at both ends.</p>
31	 <p data-bbox="475 1064 1021 1097">Requires 90 degree hooks at both ends.</p>
32	 <p data-bbox="475 1489 1037 1534">Requires 180 degree hooks at both ends.</p>
32_2	

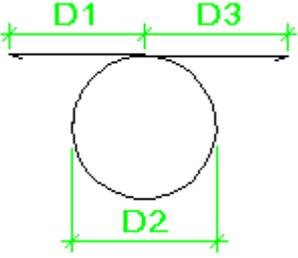
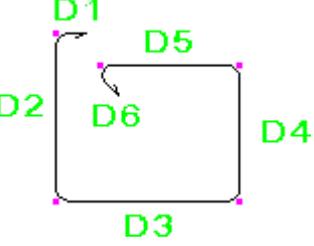
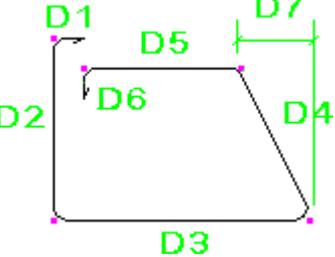
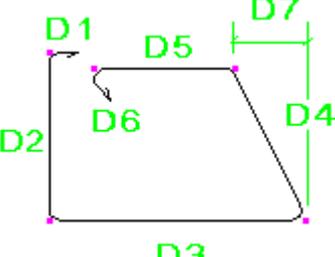
Bending type identifier	Bending shape
33	 <p>Requires 90 degree hooks at both ends.</p>
34	
35	 <p>Requires 180 degree hook.</p>
36	 <p>Requires 180 degree hook.</p>
36_2	 <p>Can also be modeled using hooks at both ends.</p>

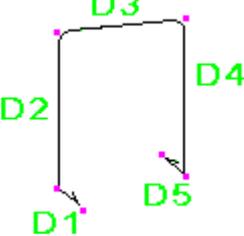
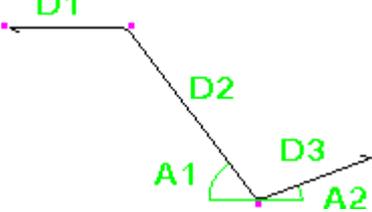
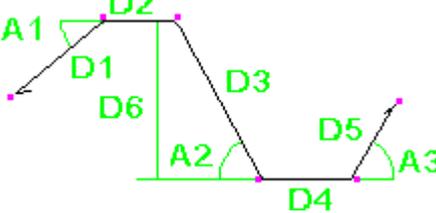
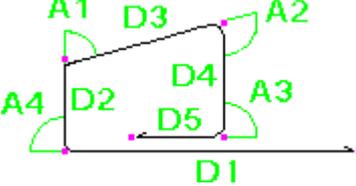
Bending type identifier	Bending shape
36_3	 <p>Can also be modeled using hooks at both ends.</p>
37	 <p>Requires 180 degree hook.</p>
38	 <p>Requires 180 degree hook at one end and 90 degree hook at the other end.</p>
38_2	
39	
40	 <p>Requires 180 degree hooks at both ends.</p>

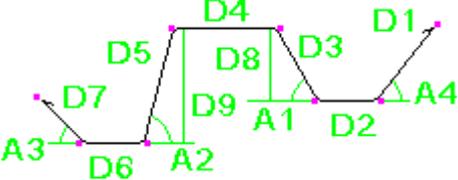
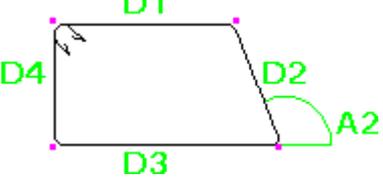
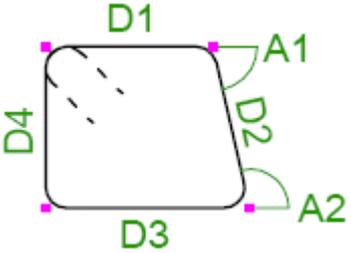
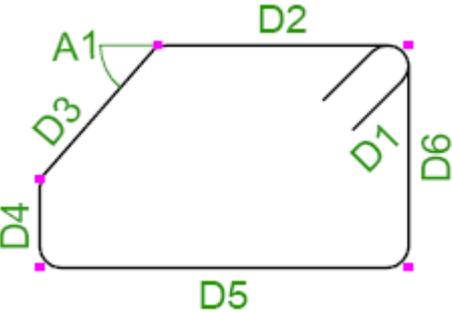
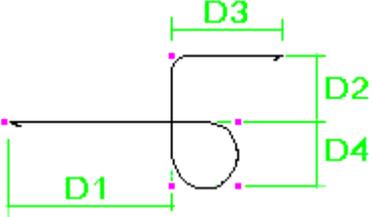
Bending type identifier	Bending shape
41	 <p data-bbox="475 593 1021 627">Requires 90 degree hooks at both ends.</p>
42	 <p data-bbox="475 761 1037 795">Requires 180 degree hooks at both ends.</p>
43	
43_2	
44	 <p data-bbox="475 1668 877 1702">Requires hooks at both ends.</p>

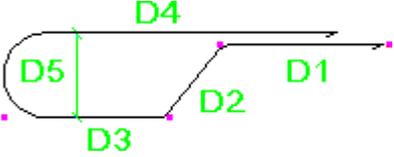
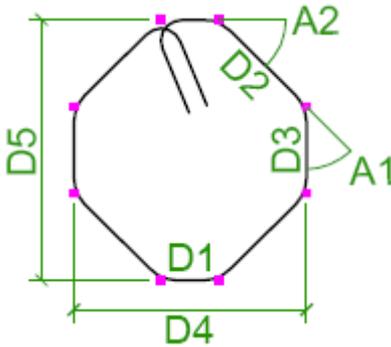
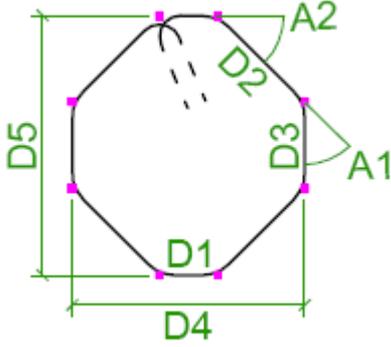
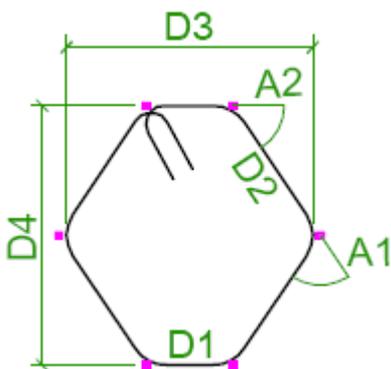
Bending type identifier	Bending shape
44_2	 <p>Requires 180 degree hooks at both ends.</p>
45	 <p>Requires hooks at both ends.</p>
45_2	 <p>Requires 180 hooks at both ends.</p>
46	 <p>Requires hooks at both ends.</p>
47	 <p>Requires 90 degree hooks at both ends.</p>

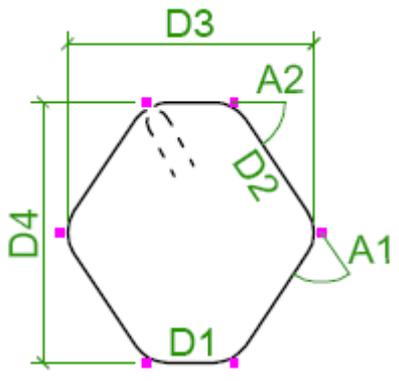
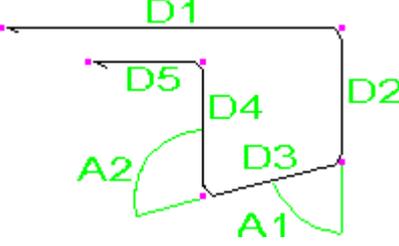
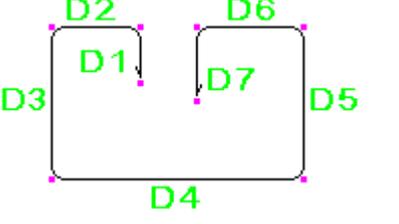
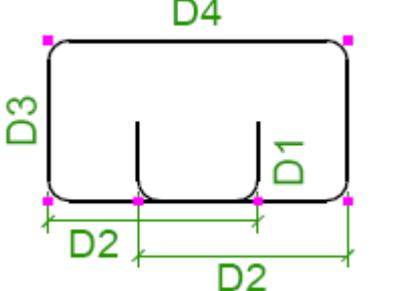
Bending type identifier	Bending shape
48	 <p data-bbox="475 627 877 660">Requires hooks at both ends.</p>
48_2	 <p data-bbox="475 1008 877 1041">Requires hooks at both ends.</p>
48_3	
49	 <p data-bbox="475 1646 1053 1680">D1 = Reinforcing bar center line diameter.</p>

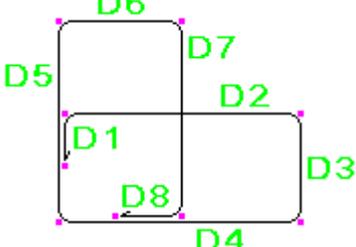
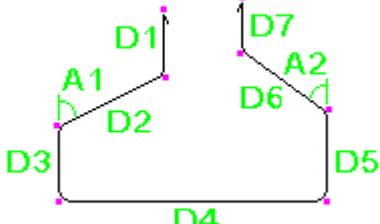
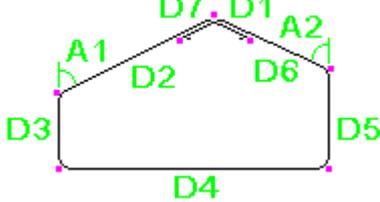
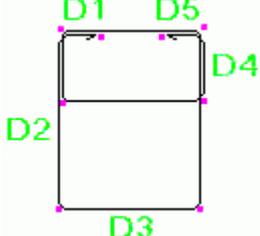
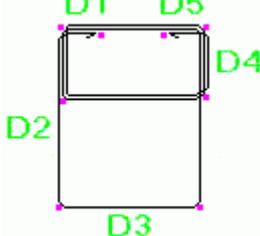
Bending type identifier	Bending shape
49_2	 <p>A circular shape with a horizontal diameter dimensioned as D1 and a vertical diameter dimensioned as D2. A horizontal dimension D3 is shown above the circle, spanning the width of the circle.</p>
50	 <p>A square shape with dimensions D1 (top), D2 (left), D3 (bottom), and D4 (right). An internal dimension D5 is shown at the top, and D6 is shown at the top-left corner. A hook is indicated at the top-left corner.</p> <p>Requires hooks at both ends.</p>
51	 <p>A trapezoidal shape with dimensions D1 (top-left), D2 (left), D3 (bottom), D4 (right), D5 (top), and D6 (top-left internal). A hook is indicated at the top-left corner. Dimension D7 is shown at the top-right corner.</p> <p>Requires 90 degree hooks at both ends.</p>
52	 <p>A trapezoidal shape with dimensions D1 (top-left), D2 (left), D3 (bottom), D4 (right), D5 (top), and D6 (top-left internal). A hook is indicated at the top-left corner. Dimension D7 is shown at the top-right corner.</p> <p>Requires hooks at both ends.</p>
53	 <p>A horizontal line with a hook at the left end (D1) and a hook at the right end (D3). The length of the line is dimensioned as D2.</p> <p>Requires hooks at both ends.</p>

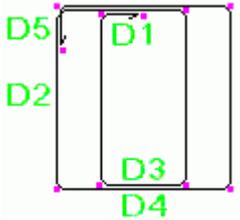
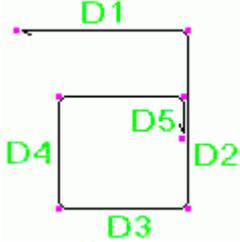
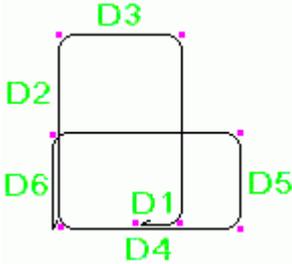
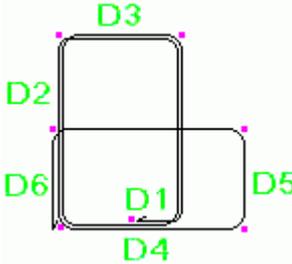
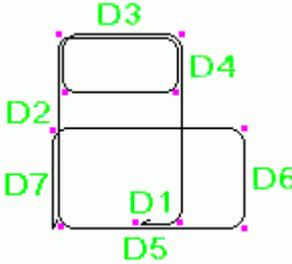
Bending type identifier	Bending shape
54	 <p data-bbox="475 577 863 611">Requires hooks at both ends.</p>
55	
56	
57	
58	
59	

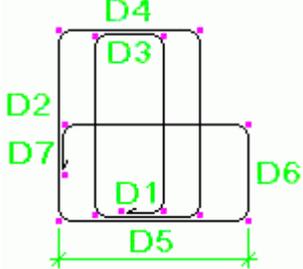
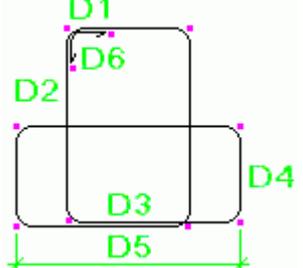
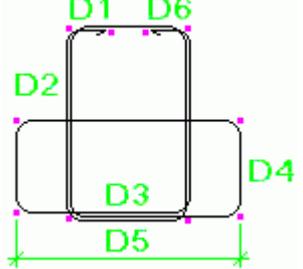
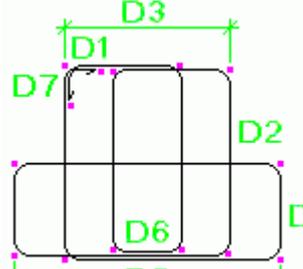
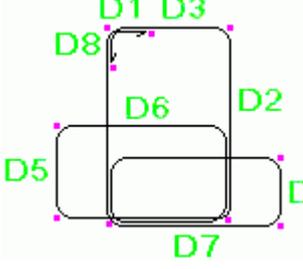
Bending type identifier	Bending shape
60	
61	 <p data-bbox="475 763 874 792">Requires hooks at both ends.</p>
61_2	 <p data-bbox="475 1099 1380 1167">Recognized if XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION is set to FALSE.</p>
61_3	 <p data-bbox="475 1536 874 1570">Requires hooks at both ends.</p>
62	 <p data-bbox="475 1832 683 1865">Requires hook.</p>

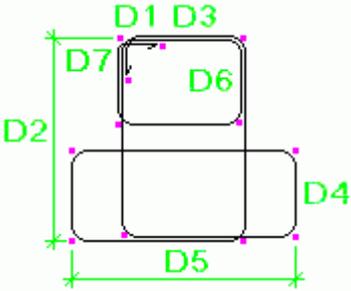
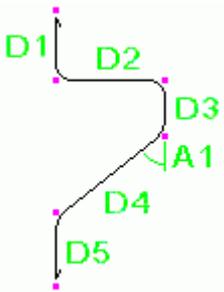
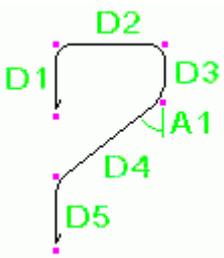
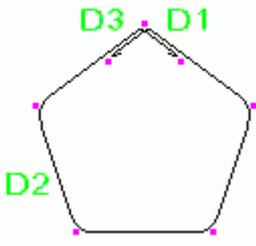
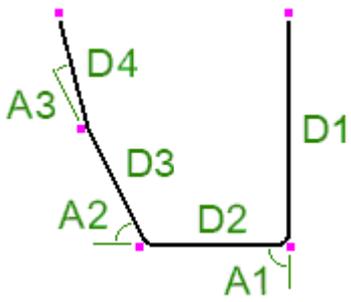
Bending type identifier	Bending shape
63	 <p data-bbox="475 521 683 555">Requires hook.</p>
64	 <p data-bbox="475 947 874 981">Requires hooks at both ends.</p>
64_2	 <p data-bbox="475 1384 1380 1451">Recognized if XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION is set to FALSE.</p>
65	 <p data-bbox="475 1877 874 1910">Requires hooks at both ends.</p>

Bending type identifier	Bending shape
65_2	 <p>Recognized if XS_REBAR_RECOGNITION_HOOKS_CONSIDERATION is set to FALSE.</p>
66	
67	
67_2	

Bending type identifier	Bending shape
68	
69_1	
69_2	
70_1	
70_2	

Bending type identifier	Bending shape
71	
72	
73_1	
73_2	
73_3	

Bending type identifier	Bending shape
74	
75_1	
75_2	
76	
77	

Bending type identifier	Bending shape
78	 <p>A diagram of a stepped bending shape. It consists of a lower horizontal segment and an upper horizontal segment. The lower segment has a width dimension D5 and a height dimension D2. The upper segment has a width dimension D3 and a height dimension D6. The transition between segments is a vertical step with a width dimension D7. The total width of the lower segment is D5, and the total width of the upper segment is D3. The height of the lower segment is D2, and the height of the upper segment is D6. The dimensions are labeled in green text with corresponding dimension lines.</p>
79_1	 <p>A diagram of a stepped bending shape. It starts with a vertical segment of height D1, followed by a horizontal segment of length D2. The next segment is a diagonal line with a slope angle A1, having a vertical projection of D3 and a horizontal projection of D4. The final segment is a vertical segment of height D5. The dimensions and angle are labeled in green text with corresponding dimension lines.</p>
79_2	 <p>A diagram of a stepped bending shape. It starts with a vertical segment of height D1, followed by a horizontal segment of length D2. The next segment is a diagonal line with a slope angle A1, having a vertical projection of D3 and a horizontal projection of D4. The final segment is a vertical segment of height D5. The dimensions and angle are labeled in green text with corresponding dimension lines.</p>
80	 <p>A diagram of a pentagonal bending shape. The top-left side is labeled D3, the top-right side is labeled D1, and the left side is labeled D2. The bottom side is horizontal. The dimensions are labeled in green text with corresponding dimension lines.</p>
UNKNOWN	<p>For example:</p>  <p>A diagram of a U-shaped bending shape. The left side is a curved segment with a slope angle A3, having a vertical projection of D4. The bottom side is a horizontal segment of length D2, with a slope angle A2 at the left end and a slope angle A1 at the right end. The right side is a vertical segment of height D1. The dimensions and angles are labeled in green text with corresponding dimension lines.</p>

See also

[Reinforcement in templates \(page 559\)](#)

[Reinforcement shape recognition \(page 523\)](#)

Reinforcement in templates

Sometimes you need to localize reinforcing bar bending types or to create templates for reinforcing bar bending schedules.

NOTE If you want to customize the hard-coded bending shapes or define new bending shapes, use **Rebar shape catalog**. See [Define reinforcing bar bending shapes in Rebar shape manager \(page 524\)](#).

Reinforcement templates

You can show dimensions, bending angles, and bending types of reinforcing bars in drawings and reports by including reinforcement-specific attributes, such as `DIM_A`, `ANG_S`, `SHAPE`, and `SHAPE_INTERNAL`, in template fields. For more information on creating templates, see the Template Editor (TplEd) help.

Mapping dimensions

Use the `rebar_schedule_config.inp` file in the `..\ProgramData\Trimble\Tekla Structures\<version>\environments\<environment>\system` folder to map

- Tekla Structures internal reinforcing bar dimensions and angles with specific template attributes
- Tekla Structures internal reinforcing bar bending types with specific bending types

These mappings are environment-specific by default. You can modify them to suit your company or project needs.

You can use equations, functions, and `if` statements to calculate the dimensions and angles you need to show.

Use any standard text editor (for example, Microsoft Notepad) to edit the `rebar_schedule_config.inp` file.

Examples

The following example of the `rebar_schedule_config.inp` file maps the internal bending type `5_1` to the bending type identifier `E`, and the leg dimensions and bending angles to specific template attributes.

rebar_schedule_config.inp	
<pre> BEND_TYPE_5_1[1]="E" BEND_TYPE_5_1[2]="DIM_A=D1" BEND_TYPE_5_1[3]="DIM_B=D5" BEND_TYPE_5_1[4]="DIM_C=D2" BEND_TYPE_5_1[5]="DIM_TD=TD" BEND_TYPE_5_1[6]="ANG_U=A1" BEND_TYPE_5_1[7]="ANG_V=A2" </pre>	

With this mapping, the internal bending type 6_2 becomes XY, and the template attributes DIM_B and DIM_C will show the horizontal and vertical dimensions of the second leg D2, and DIM_E and DIM_F the horizontal and vertical dimensions of the fourth leg D4.

rebar_schedule_config.inp	
<pre> BEND_TYPE_6_2[1]="XY" BEND_TYPE_6_2[2]="DIM_A=D1" BEND_TYPE_6_2[3]="DIM_B=D2*COS(A2*PI/180)" BEND_TYPE_6_2[4]="DIM_C=D2*SIN(A2*PI/180)" BEND_TYPE_6_2[5]="DIM_D=D3" BEND_TYPE_6_2[6]="DIM_E=D4*COS(A1*PI/180)" BEND_TYPE_6_2[7]="DIM_F=D4*SIN(A1*PI/180)" BEND_TYPE_6_2[8]="DIM_G=D5" BEND_TYPE_6_2[9]="DIM_TD=TD" </pre>	

The following example maps the internal bending type 4 to the bending type identifier A **if** the dimensions D1 and D3 are the same. Otherwise it maps 4 to B.

rebar_schedule_config.inp	
<pre> BEND_TYPE_4[1]=if (D1==D3) then ("A") else ("B") endif BEND_TYPE_4[2]="DIM_A=D1" BEND_TYPE_4[3]="DIM_B=D2" BEND_TYPE_4[4]="DIM_C=D3" BEND_TYPE_4[5]="DIM_TD=TD" </pre>	

If Tekla Structures does not recognize a reinforcing bar bending shape, it uses the internal bending type UNKNOWN for it. In the rebar_schedule_config.inp file you can also define how unknown bending types appear in drawings and reports. For example, you may just want to use the bending type identifier ???, and list all leg dimensions and bending angles.

rebar_schedule_config.inp

<pre> BEND_TYPE_UNKNOWN[1]="???" BEND_TYPE_UNKNOWN[2]="DIM_A=D1" BEND_TYPE_UNKNOWN[3]="DIM_B=D2" BEND_TYPE_UNKNOWN[4]="DIM_C=D3" BEND_TYPE_UNKNOWN[5]="DIM_D=D4" BEND_TYPE_UNKNOWN[6]="DIM_E=D5" BEND_TYPE_UNKNOWN[7]="DIM_F=D6" BEND_TYPE_UNKNOWN[8]="ANG_S=A1" BEND_TYPE_UNKNOWN[9]="ANG_T=A2" BEND_TYPE_UNKNOWN[10]="ANG_U=A3" BEND_TYPE_UNKNOWN[11]="ANG_V=A4" BEND_TYPE_UNKNOWN[12]="DIM_TD=TD" </pre>	
---	--

No.	Grade	Size	Mark	Length	Type	A	B	C	D	E	F	S	T	U	V	TD
1	A615-40	#4	R/5	1930	???	740	420	430	380			90	65	15		76

See also

[Hard-coded bending type identifiers in reinforcement shape recognition \(page 535\)](#)

[Reinforcement shape recognition \(page 523\)](#)

2.10 Create construction objects and points

Points and construction objects help you to place other objects in the model.

When you want to place objects to a position where no lines or objects intersect in the model, you can create [construction lines \(page 562\)](#), [planes \(page 562\)](#), [circles \(page 563\)](#), [arcs \(page 564\)](#), and [polycurves \(page 565\)](#). For example, you can easily [pick \(page 77\)](#) the points at intersections of construction lines and circles. The [snap priority \(page 78\)](#) of construction objects is the same as with the other lines.

Construction objects remain in the model when you update or redraw views and windows. They do not appear in drawings.

You can also create magnetic construction lines or planes to bind and move groups of objects. For example, rather than binding lots of handles and chamfers to part faces, simply create a construction plane that goes through all the handles and chamfers. Then make this plane magnetic and bind the plane to the appropriate face. When you move the plane, the attached handles and chamfers move with it.

See also

[Create a construction line \(page 562\)](#)

[Create a construction plane \(page 562\)](#)

[Create a construction circle \(page 563\)](#)

- [Create a construction arc \(page 564\)](#)
- [Create a construction polycurve \(page 565\)](#)
- [Copy a construction object with offset \(page 566\)](#)
- [Modify a construction object \(page 567\)](#)
- [Create points \(page 570\)](#)

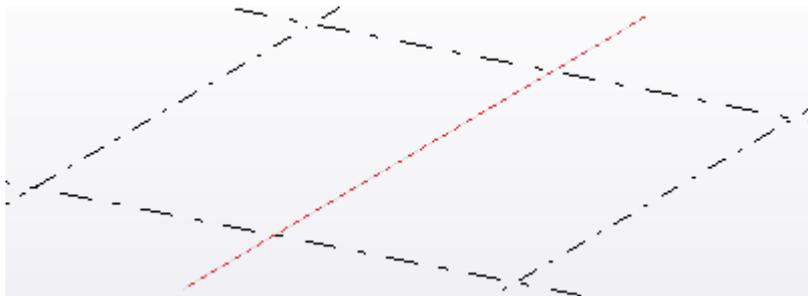
Create a construction line

1. On the **Edit** tab, click **Construction object** --> **Line** .
2. Pick the start point of the construction line.
3. Pick the end point of the construction line.
4. To end the command, press **Esc**.
5. To modify the construction line properties, double-click the line in the model.

The line properties are shown in the property pane.

- a. If you want to make the line magnetic, select **Yes** in the **Magnetic** list.
- b. Select a color for the line.
- c. Define how far the line extends beyond the picked points.
- d. Select a line type for the line.
- e. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.



See also

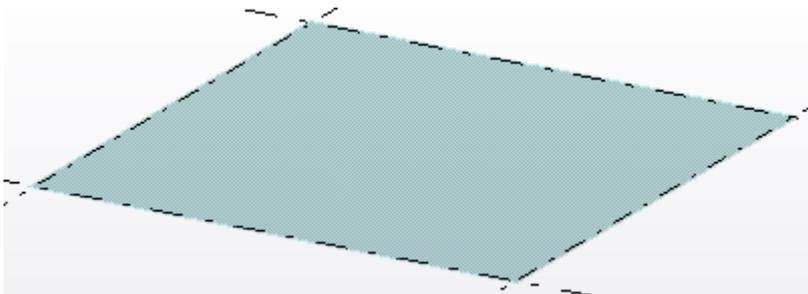
- [Create construction objects and points \(page 561\)](#)
- [Copy a construction object with offset \(page 566\)](#)
- [Modify a construction object \(page 567\)](#)

Create a construction plane

1. On the **Edit** tab, click **Construction object --> Plane** .
2. Pick three points.
3. Click the middle mouse button.
Tekla Structures draws the plane.
4. To end the command, press **Esc**.
5. To modify the construction plane properties, double-click the plane in the model.

The plane properties are shown in the property pane.

- a. Enter a name for the plane.
- b. If you want to make the construction plane magnetic, select **Yes** in the **Magnetic** list.
- c. Click **Modify**.



See also

[Create construction objects and points \(page 561\)](#)

[Modify a construction object \(page 567\)](#)

Create a construction circle

You can create construction circles by picking three points in the 3D space in the model.

1. On the **Edit** tab, click **Construction object --> Circle** .
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick.
 - Click  , and then pick three points: the center point, a point to define the radius, and a point to define the plane of the circle.

- Click , and then pick three points along the arc of the circle.

Tekla Structures creates the circle using the points you picked and using the current properties. Tekla Structures also indicates the center point of the circle with an X in the model.

3. To end the command, press **Esc**.
4. To modify the construction circle properties, double-click the circle in the model.

The circle properties are shown in the property pane.

- a. Select a color for the circle.
- b. Select a line type for the circle.
- c. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

See also

[Create construction objects and points \(page 561\)](#)

[Copy a construction object with offset \(page 566\)](#)

[Modify a construction object \(page 567\)](#)

Create a construction arc

You can create construction arcs by picking three points in the 3D space in the model.

1. On the **Edit** tab, click **Construction object --> Arc**.
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick:

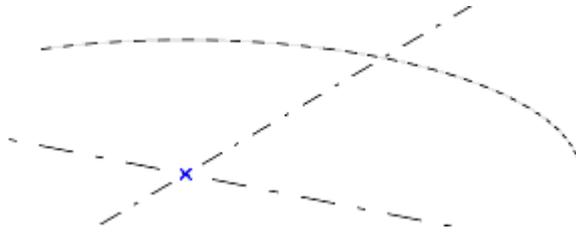


- Click , and then pick three points: the center point, start point, and end point of the arc.

You can also define the angle or the length of the arc.

- Click , and then pick the start point, end point, and an optional point along the arc.
- Click , and then pick a point to define a tangent, and two points along the arc.

Tekla Structures creates the arc using the points you picked and using the current properties. Tekla Structures also indicates the center point of the arc with an X in the model.



3. To end the command, press **Esc**.
4. To modify the construction arc properties, double-click the arc in the model.

The arc properties are shown in the property pane.

- a. Select a color for the arc.
- b. Select a line type for the arc.
- c. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

See also

[Create construction objects and points \(page 561\)](#)

[Copy a construction object with offset \(page 566\)](#)

[Modify a construction object \(page 567\)](#)

Create a construction polycurve

You can create 3D construction polycurves that pass through the points you pick and that can have straight and curved segments.

1. On the **Edit** tab, click **Construction object --> Polycurve**.
2. On the contextual toolbar that appears, click a button to specify which set of points you want to pick to create a polycurve segment.

You can switch between these picking modes each time you complete a segment.



- For a straight segment, click the first button  **Create line** and then pick the start point and the end point of the segment.

- For a curved segment, click  and then pick three points along the segment.
- For a curved tangential segment, click  and then pick a point on the tangent line, the start point, and the end point of the segment.
- For a straight segment that is tangential to the previous segment, click the last button  **Create tangent line** and then pick a point on the tangent line.

Tekla Structures creates a polycurve segment.

3. Repeat step 2 for each polycurve segment that you want to create, but skip picking the first point of the segment as it is the same as the last point of the previous segment.
4. Click the middle mouse button to finish picking.

Tekla Structures creates the polycurve through the points you picked, using the polycurve's current properties.

5. To end the command, press **Esc**.
6. To modify the construction polycurve properties, double-click the polycurve in the model.

The polycurve properties are shown in the property pane.

- a. Select a color for the polycurve.
- b. Select a line type for the polycurve.
- c. Click **Modify** to apply the changes.

Tekla Structures uses the new properties the next time you create an object of this type.

See also

[Create construction objects and points \(page 561\)](#)

[Copy a construction object with offset \(page 566\)](#)

[Modify a construction object \(page 567\)](#)

Copy a construction object with offset

You can copy construction lines, circles, arcs, and polycurves in the direction that you indicate, and using the offset values that you specify. For example, you can create new circles and arcs centered in the same location as the original circle or arc, and adjust the radiuses using the offset values.

1. On the **Edit** tab, click **Construction object --> Copy with offset** .

2. Select the construction object that you want to copy.
You can copy [lines \(page 562\)](#), [circles \(page 563\)](#), [arcs \(page 564\)](#), and [polycurves \(page 565\)](#).
3. Enter the offset values in the box that appears, and then press **Enter**.
If you only enter one offset value, Tekla Structures creates one copy of the object.
To create several copies, enter multiple offset values. For example, 500 1000 1500 or 3*500.
4. Click in the direction where you want to copy the object.

Tekla Structures copies the selected object in the direction you indicated.

For example, if you selected a line, Tekla Structures makes a new copy of the line in the specified location. If you selected a circle or arc, Tekla Structures creates a new object that is centered in the same location as the original object, and adjusts the radius using the offset value that you specified.

See also

[Create construction objects and points \(page 561\)](#)

[Modify a construction object \(page 567\)](#)

Modify a construction object

You can modify construction points, lines, circles, arcs, polycurves, and planes using direct modification.

Before you start:

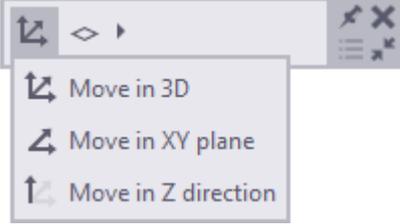
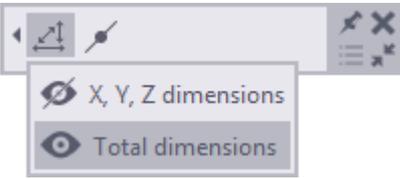
- Ensure that the  **Direct modification** switch is active.
- Select the construction object.

Tekla Structures displays the handles and dimensions that you can use to modify the construction object.

When you select a handle and move the mouse pointer over , Tekla Structures displays a toolbar with more modification options. The available options depend on the type of the construction object you are modifying.

To modify a construction object, do any of the following:

To	Do this	Available for
Set a reference point to move in one, two, or any direction	1. Select the handle in the reference point.	Construction points, lines, circle center points,

To	Do this	Available for
	<p>2. To define in which directions the handle can move, select an option from the list on the toolbar:</p>  <p>You can also press Tab to cycle through the options.</p> <p>3. To move the handle parallel to a certain plane only, click  and select the plane.</p>	planes ((arc center points??))
Move a point, a point on a line, circle, arc or polycurve, or a plane corner	Drag the handle in the reference point to a new location.	All construction objects
Move a circle or arc	Drag the handle in the center point to a new location.	Construction circles, arcs
Move a line or a plane edge	Drag the line handle to a new location.	Construction lines, planes
Move a plane	Drag the plane to a new location.	Construction planes
Show or hide diagonal dimensions	<p>1. Select a handle.</p> <p>2. On the toolbar, click .</p> <p>3. Click the eye button to show or hide orthogonal and total dimensions:</p> 	Construction lines, planes
Change a dimension	<p>Drag a dimension arrowhead to a new location, or:</p> <p>1. Select the dimension arrowhead which you want to move.</p>	Construction lines, circles, planes Arcs (numeric input only)

To	Do this	Available for
	<p>To change the dimension at both ends, select both arrowheads.</p> <p>To change the radius of a circle or arc, select the outer arrowhead.</p> <p>2. Using the keyboard, enter the value with which you want the dimension to change.</p> <p>To start with the negative sign (-), use the numeric keypad.</p> <p>To enter an absolute value for the dimension, first enter \$, then the value.</p> <p>3. Press Enter, or click OK in the Enter a Numeric Location dialog box.</p>	
Chamfer a polycurve corner	<p>1. Select a corner handle.</p> <p>2. On the toolbar:</p> <ul style="list-style-type: none"> • Click  to create a rounded chamfer, and then enter the chamfer radius. • Click  to create a straight chamfer, and then enter the chamfer dimensions X and Y. <p>3. Press Enter to confirm the chamfer dimensions.</p>	Polycurves
Change an arc to a line Make a curved segment straight	<p>Select the arc or segment midpoint handle (with an arc symbol)  and press Delete.</p>	Arcs, curved polycurve segments
Change a line to an arc Make a straight segment curved	<p>Drag the arc symbol  at the midpoint of the line or segment.</p>	Lines, straight polycurve segments
Add a corner point and an intermediate segment to a polycurve	<p>Drag a segment midpoint handle to a new location.</p>	Polycurves
Remove a corner point and the two	<p>Select the corner point handle and press Delete.</p>	Polycurves

To	Do this	Available for
connected segments		
Remove the last polycurve segment	Select the end point handle and press Delete .	Polycurves
Change the radius of an arc and keep the end point locations	Click the radius dimension, enter a new value, and press Enter .	Arcs
Change the radius of a polycurve segment and keep the end point locations	Drag the arc symbol  at the segment midpoint handle.	Curved polycurve segments
Change the angle or length of an arc	Drag the start or end point to a new location.	Arcs
Copy a construction object using offset	See Copy a construction object with offset (page 566) .	Lines, circles, arcs, polycurves

See also

[Create construction objects and points \(page 561\)](#)

[Create points \(page 570\)](#)

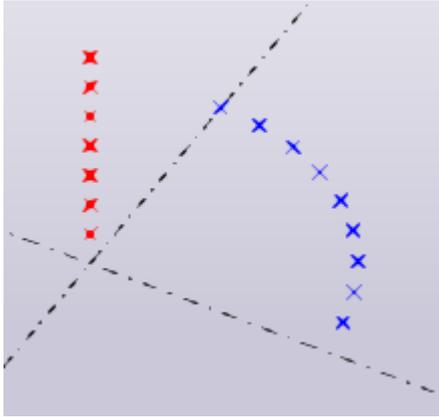
[Resize and reshape model objects \(page 103\)](#)

Create points

You can create points to make it easier to place model objects at positions where no lines or objects intersect.

There are many ways to create points in Tekla Structures. Which method is the most convenient at each time depends on what you have already created in the model and which locations you can easily pick.

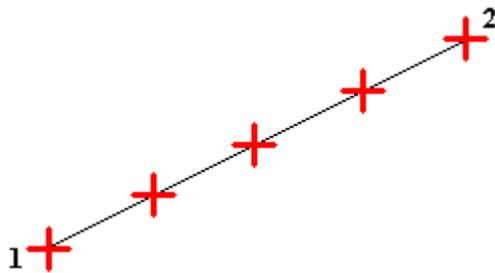
When you create points, Tekla Structures always places them according to the work plane coordinate system. Points located on the view plane are by default blue and points outside the view plane are by default red. You can change the color of points in the point properties.



Create points on a line

You can create points at equal intervals along a line that is defined by two points.

1. On the **Edit** tab, click **Points** --> **On line** .
The **Divided Line Points** dialog box appears.
2. Define the number of points to be created.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).



Create points on a plane

You can create several equally spaced points in the desired area in the model. The points are created in relation to the picked origin position.

A point array consists of several points in a rectangular xy(z) pattern relative to the current work plane. The x, y, and z coordinates of the points define the array pattern. The x and y coordinates are relative distances between the

points on the work plane. The z coordinates are absolute distances perpendicular to the work plane.

1. On the **Edit** tab, click **Points --> On plane** .
The **Point Array** dialog box appears.
2. Define the array point coordinates.
Use positive or negative values to define the direction of the array.
Use a zero at the beginning of the row to represent a point in the array origin. Separate multiple values with spaces.
3. Pick the origin of the array in the view.
Alternatively, you can define the origin in the **Point Array** dialog box.
4. Click **OK**.

Create points parallel to two points

You can create offset points that are parallel to a line between two points you have picked.

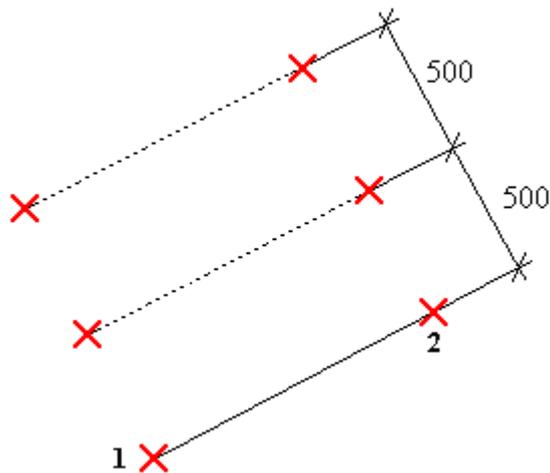
1. On the **Edit** tab, click **Points --> Parallel to two points** .
The **Point Input** dialog box appears.
2. Define the distances at which the points are created.
If you want to create multiple pairs of offset points, enter multiple values separated with spaces.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).

The picking order of the start point and the end point defines the offset direction of the new points.

When you look from the start point to the end point, Tekla Structures creates the new points to the left of the picked points. If you enter negative values to the **Point Input** dialog box, Tekla Structures creates points to the right of the picked points.

When you pick points, Tekla Structures uses arrows to indicate the offset direction.

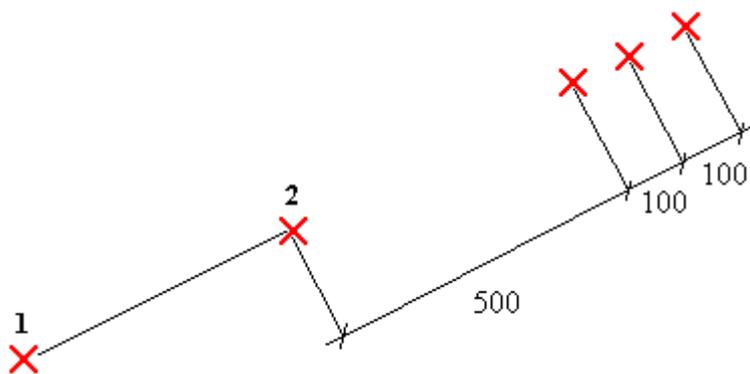
For example, if you enter 500 500 to the **Point Input** dialog box, the first pair of new points is created at a 500 mm distance from the picked points, and the second pair of points is created at a 500 mm distance from the first pair of points.



Create points along the extension line of two points

1. On the **Edit** tab, click **Points** --> **Along extension of two points** .
The **Point Input** dialog box opens.
2. Define the distances at which the points are created.
Separate multiple values with spaces.
3. Click **OK**.
4. Pick the start point of the line (1).
5. Pick the end point of the line (2).

For example, if you enter 500 100 100 to the **Point Input** dialog box, the first point is created at 500 mm distance from the end point of the line, and the second and the third point are each created at 100 mm distance from the previous point.

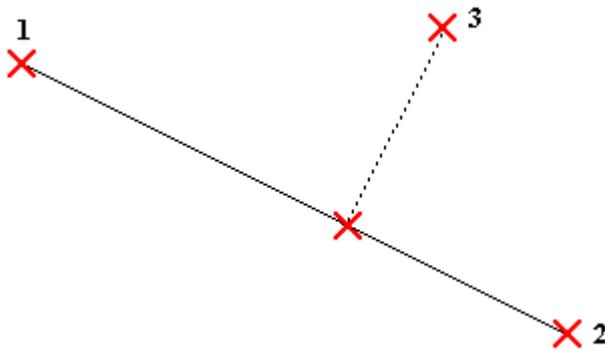


TIP Enter a negative value to the **Point Input** dialog box to create a point between the start point and the end point.

Create projected points on a line

You can project a point onto a selected line or its extension.

1. On the **Edit** tab, click **Points** --> **Projected points on line** .
2. Pick the first point on the line (1).
3. Pick the second point on the line (2).
4. Pick the point to be projected (3).

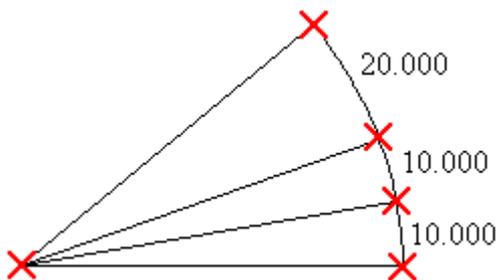


Create points along an arc using center and arc points

You can create points along an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using center and arc points** .
The **Arc Points** dialog box appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.
Give the angle values in degrees.
Separate multiple angle and distance values with spaces.
3. Click **OK**.
4. Pick the center point.
5. Pick the start point of the arc.

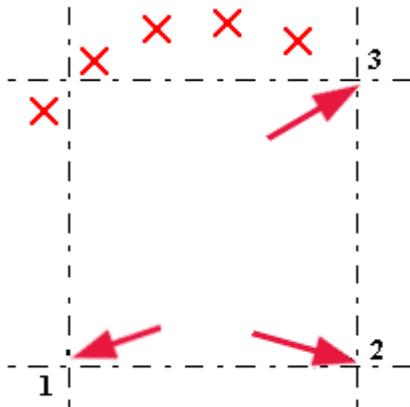
Tekla Structures creates the arc points counterclockwise from the start point.



Create points along an arc using three arc points

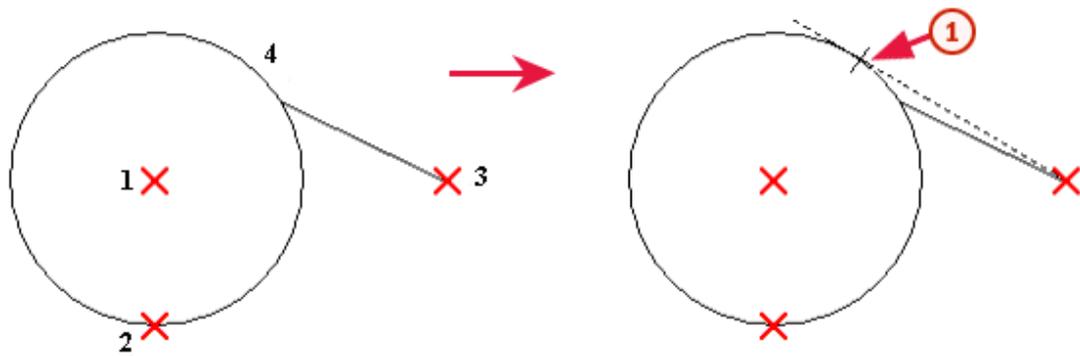
You can create points as an extension of an arc.

1. On the **Edit** tab, click **Points** --> **Along arc using three arc points** .
The **Arc Points** dialog box appears.
2. Select either **Angles** or **Distances** and enter the angles or distances between the points along the arc.
Give the angle values in degrees.
Separate multiple angle and distance values with spaces.
3. Click **OK**.
4. Pick three points along the arc (1-3).



Create points tangent to a circle

1. On the **Edit** tab, click **Points** --> **Tangent to circle** .
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the end point of the tangent (3).
5. Pick a side to indicate the side on which Tekla Structures creates the tangent point (4).



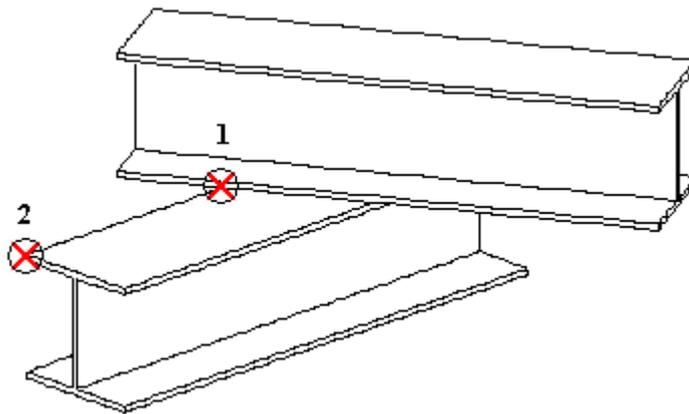
① Tangent point

Create points at any position

NOTE [Snap switches \(page 79\)](#) determine the positions you can pick.

You can also use temporary reference points and numeric snapping to create a point, for example, to a certain distance from an existing corner or a point.

1. On the **Edit** tab, click **Points** --> **At any position**.
2. Pick the intersection of two part edges (1), or the corner of a part (2).



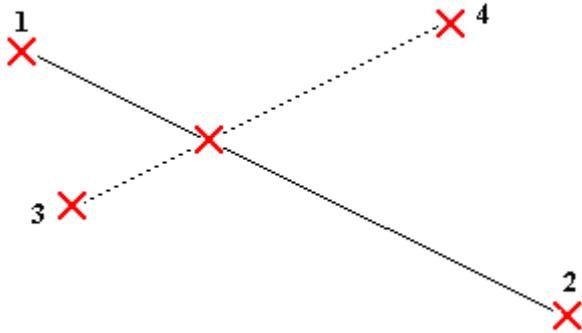
Create bolt points

You can create points on the view plane at the center points of single bolts and bolt group bolts.

1. On the **Edit** tab, click **Points** --> **Bolt points**.
2. Select a bolt or a bolt group.

Create points at the intersection of two lines

1. On the **Edit** tab, click **Points** --> **At intersection of two lines** .
2. Pick the start point of the first line (1).
3. Pick the end point of the first line (2).
4. Pick the start point of the second line (3).
5. Pick the end point of the second line (4).



Create points at the intersection of a plane and a line

1. On the **Edit** tab, click **Points** --> **At intersection of plane and line** .
2. Pick three points to define the plane.
3. Pick the first point of the line.
4. Pick the second point of the line.

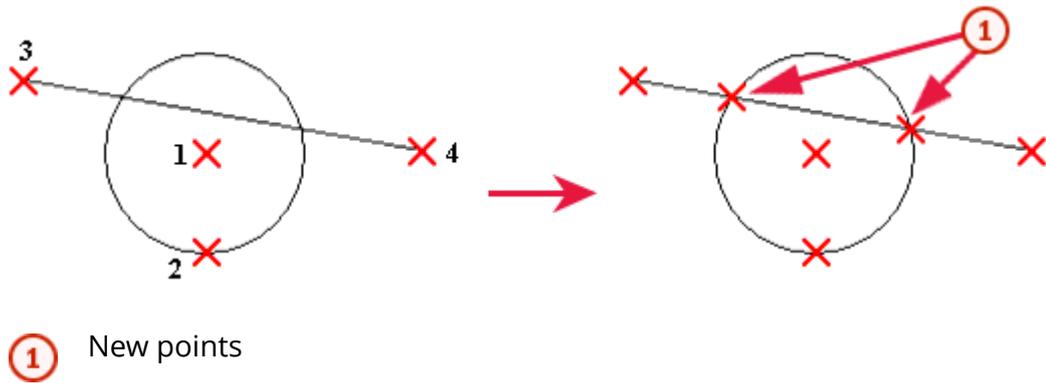
Create points at the intersection of a part and a line

You can create points where a line intersects with the surface of a part.

1. On the **Edit** tab, click **Points** --> **At intersection of part and line** .
2. Select the part.
3. Pick the first point of the line.
4. Pick the second point of the line.

Create points at the intersection of a circle and a line

1. On the **Edit** tab, click **Points** --> **At intersection of circle and line** .
2. Pick the center point of the circle (1).
3. Pick a point on the circle to define the radius (2).
4. Pick the first point on the line (3).
5. Pick the second point on the line (4).

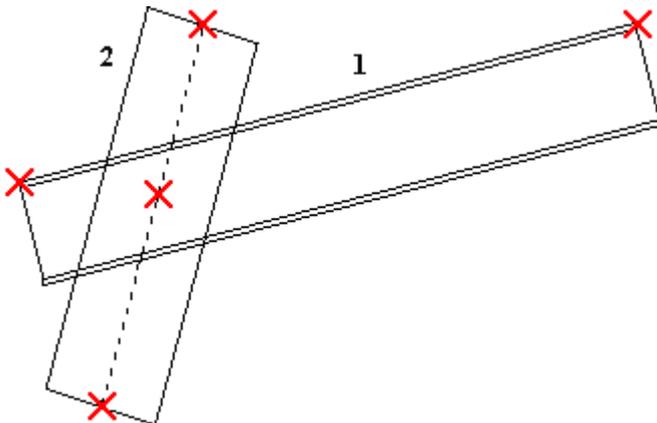


Create points at the intersection of two part axes

You can create points where the axes of two parts intersect, and project the points onto the axis of the part that you select first.

1. On the **Edit** tab, click **Points** --> **At intersection of two part axes** .
2. Select the first part (1).
3. Select the second part (2).

Tekla Structures projects the point onto the axis of the first part.



Import points

NOTE This is for advanced users.

You can import points to specific locations in an open Tekla Structures model using the **Point Creation Import (8)** component. You need to specify the point coordinates in a text file. In some cases this file is generated by another software package.

1. Create a point import file.
 - a. Create a text file that consists of single lines for each point.

Use commas or tabs as delimiters for the three point coordinates on a line. For example:

100, 500, 1000

300, 700, 1500

- b. Save the file.

NOTE During the import process, Tekla Structures ignores all lines in the import file which do not consist of valid values delimited by tabs or commas.

2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Enter `point` in the **Search...** box.
4. Click **Point Creation Import (8)**.
5. Enter the ASCII file name.
Include the full path and the file name extension (for example `.txt`). If you do not specify the path, Tekla Structures looks for the file in the current model folder.
6. Define the origin of the imported points by entering the coordinates.
7. Click **Create**.

Point properties

Use the **Point** properties to view and modify the properties of a point.

If you have [customized \(page 213\)](#) the property pane layout, the list of properties may be different.

Setting	Description
General	
Color	Change the color of the points. Note that if you change the point color, the changed color is not applied the next time you create points. The points are created using the default point color.
Location	
X	The local (work plane) and global x, y, and z coordinates of a point. Indicates the correct location of the point.
Y	
Z	

TIP You can change the point size in [Display settings \(page 918\)](#).

3 Change how model objects are displayed

To change the display of model objects:

- You can change the general display settings to [Set the visibility and appearance of model objects \(page 582\)](#) and [Change the rendering of parts and components \(page 584\)](#).
- You can temporarily hide selected objects as explained in [Hide model objects \(page 587\)](#) or do the opposite and show only the selected objects as explained in [Show only selected model objects \(page 588\)](#).
- You can temporarily show hidden assembly and component objects as explained in [Temporarily display assembly and component objects \(page 589\)](#).
- To show all the detailing that belongs to a part, see [Display detailing of a part \(page 590\)](#).
- To show a part in a selected view angle, see [Show parts in a selected view angle \(page 591\)](#).
- You can group model objects using various criteria (for example, profile) to handle them as a single unit when you define display settings, see [Create object groups \(page 591\)](#).
- Detailed instructions for changing model object color and transparency can be found in the section [Change the color and transparency of model objects \(page 593\)](#).

3.1 Show and hide model objects

This section explains how to control the visibility and appearance of parts and other model objects.

Click the links below to find out more:

[Set the visibility and appearance of model objects \(page 582\)](#)

[Change the rendering of parts and components \(page 584\)](#)

[Hide model objects \(page 587\)](#)

[Show only selected model objects \(page 588\)](#)

[Temporarily display assembly and component objects \(page 589\)](#)

[Display detailing of a part \(page 590\)](#)

[Show parts in a selected view angle \(page 591\)](#)

Set the visibility and appearance of model objects

Modify the display settings to define how parts and other model objects appear in a model view.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** to open the **Display** dialog box.
3. Select or clear check boxes to specify which objects are visible in the view.
4. Select a representation option for parts, bolts, holes, welds, construction planes and reinforcing bars.

You have the following options:

- **Fast**
 - **Exact**
 - **Reference line** (only for parts)
 - **Exact slotted holes** (only for holes)
 - **Exact - no weld mark** (only for welds)
5. If you are working with [cast-in-place \(page 404\)](#) concrete structures, and the pour management functionality is [enabled \(page 402\)](#):
 - a. In the **Cast in place** list, select whether you want to show the structures as **Parts** or as **Pours**.
 - b. If you selected **Parts** for the cast-in-place concrete structures, select whether you want to show the parts as **Merged** or as **Separated**.
 6. Ensure that the view is selected.
 7. Click **Modify** to apply the changes.

See also

[Display settings \(page 918\)](#)

[Show parts with exact lines \(page 583\)](#)

[Show parts with high accuracy \(page 583\)](#)

[Show part handles and part reference lines in a model view \(page 303\)](#)

[Set the visibility and appearance of welds \(page 358\)](#)

[Set the visibility of pour breaks \(page 417\)](#)

[Change the rendering of parts and components \(page 584\)](#)

[Change the color and transparency of model objects \(page 593\)](#)

Show parts with exact lines

Use the **Show part with exact lines** command to temporarily display a part with exact lines even if you are using the **Fast** representation option for parts.

1. Select the part.
2. Go to **Quick Launch**, start typing `show part with exact lines`, and select the **Show part with exact lines** command from the list that appears.
3. Click the view in which you want to display exact lines.
4. To clear the exact lines effect, on the **View** tab, click .

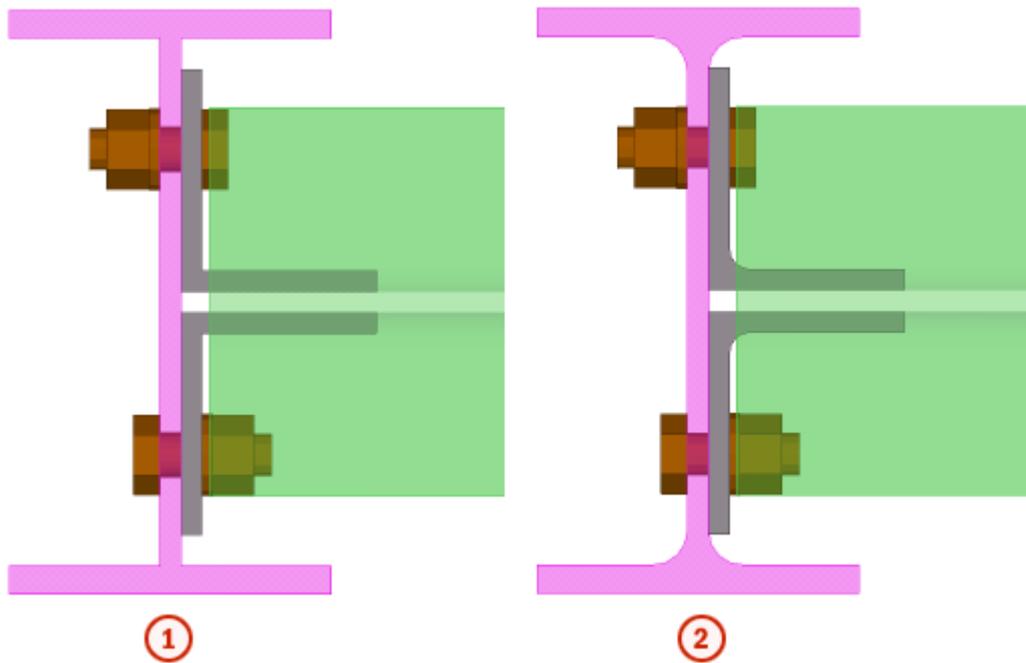
See also

[Show and hide model objects \(page 581\)](#)

Show parts with high accuracy

You can temporarily display parts with the highest possible level of accuracy. This can be useful, for example, when checking a large model, because the entire model can still be displayed in the **Fast** or **Exact** representation mode but the individual part can be shown in more detail.

1. Select the parts.
2. Right-click, and then hold down the **Shift** key while selecting **Show with Exact Lines**.
Tekla Structures displays the selected parts with the highest possible level of accuracy.
3. To clear the high accuracy effect, right-click and select **Show with Exact Lines**.



- ① Normal display mode
- ② High accuracy mode

See also

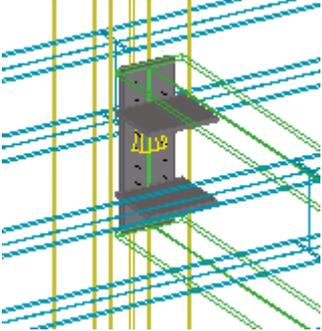
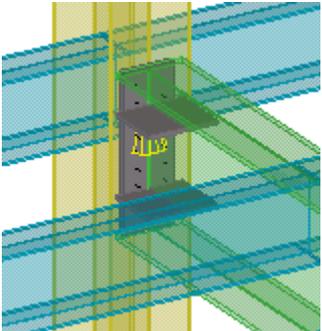
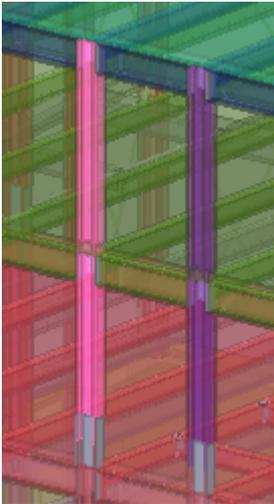
[Show and hide model objects \(page 581\)](#)

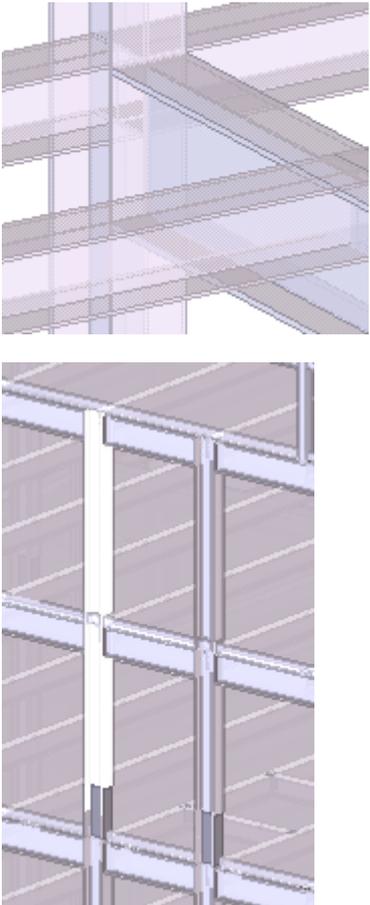
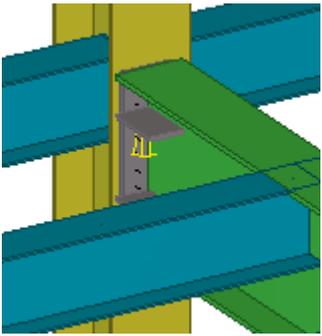
[Display settings \(page 918\)](#)

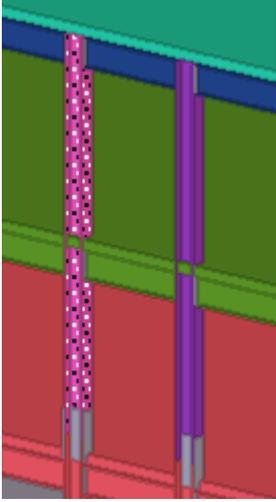
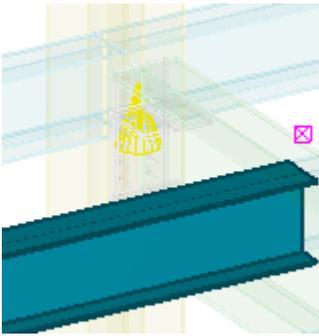
Change the rendering of parts and components

You can easily change the rendering of parts and components in model views.

1. On the **View** tab, click **Rendering**.
2. Select one of the rendering options for parts or components:

Option	Description	Example
Parts wireframe/ Components wireframe	Object outlines are displayed, surfaces are not. Objects are transparent.	 <p data-bbox="959 629 1326 734">In this wireframe example, component objects are displayed as rendered.</p>
Parts shaded wireframe/ Components shaded wireframe	<p data-bbox="523 1055 935 1160">Object outlines are displayed. Objects are transparent, and their surfaces are rendered.</p> <p data-bbox="523 1173 935 1413">If you use the DirectX rendering and have set <code>XS_HATCH_OVERLAPPING_FACES_IN_DX</code> to <code>TRUE</code>, the overlapping surfaces are visualized in rendered wireframe views.</p>	 <p data-bbox="959 1104 1358 1209">In this rendered wireframe example, component objects are displayed as rendered.</p>  <p data-bbox="959 1753 1310 1859">In this DirectX rendering example, the overlapping surfaces are visualized.</p>

Option	Description	Example
<p>Parts grayscale/ Components grayscale</p>	<p>Objects are shown in grayscale.</p> <p>If you use the DirectX rendering and have set <code>XS_HATCH_OVERLAPPING_FACES_IN_DX</code> to <code>TRUE</code>, the overlapping surfaces are visualized in grayscale views.</p>	 <p>In this DirectX rendering example, the overlapping surfaces are visualized.</p>
<p>Parts rendered/ Components rendered</p>	<p>Object surfaces are displayed. Objects are not transparent.</p> <p>If you use the DirectX rendering and have set <code>XS_HATCH_OVERLAPPING_FACES_IN_DX</code> to <code>TRUE</code>, the overlapping surfaces are visualized with a hatch.</p>	

Option	Description	Example
		 <p data-bbox="959 797 1362 936">In this DirectX rendering example, the overlapping surfaces are visualized with a hatch.</p>
<p data-bbox="309 1256 507 1429">Show only selected part/Show only selected component</p>	<p data-bbox="525 1256 916 1395">Selected objects are displayed. Other objects are almost completely transparent.</p> <p data-bbox="525 1413 943 1509">This option is useful, for example, when viewing clash check results in a large model.</p>	

TIP Alternatively, you can use the keyboard shortcuts **Ctrl+1...5** for parts and **Shift+1...5** for components to switch between the rendering options.

See also

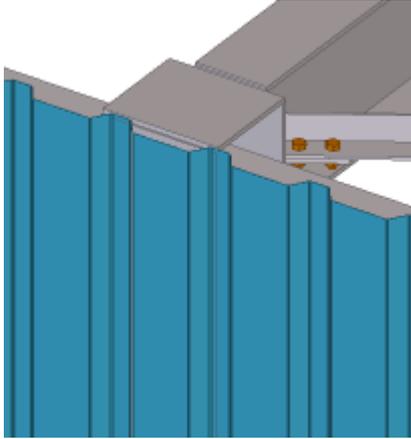
[Set the visibility and appearance of model objects \(page 582\)](#)

[Change the rendering of the model \(page 67\)](#)

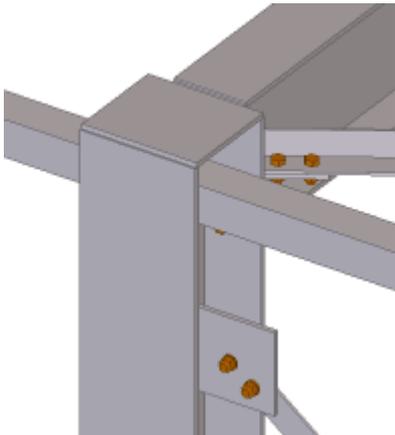
Hide model objects

You can quickly hide selected parts or other objects in a model view. This can be useful, for example, when you want to temporarily hide parts to see the parts behind them.

1. Select the objects you want to hide.



2. Right-click and select **Hide**.
The selected objects become invisible.



3. To make the objects visible again, click  on the **View** tab.

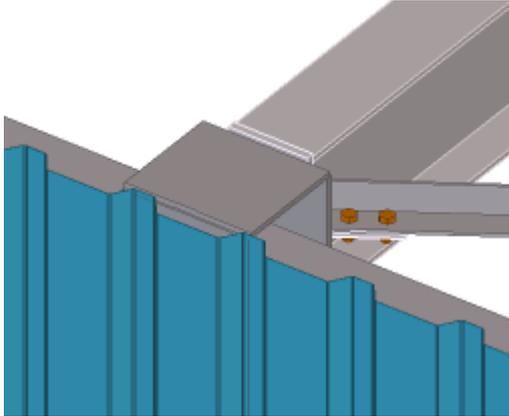
See also

[Show only selected model objects \(page 588\)](#)

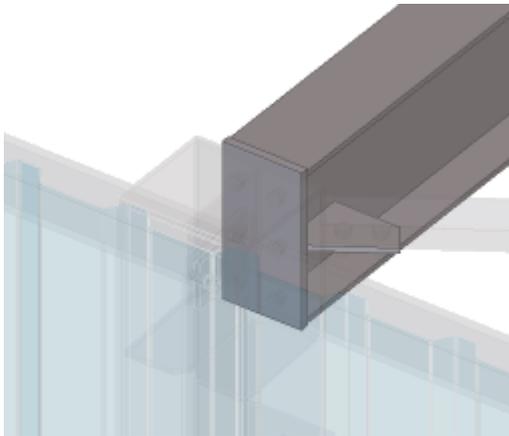
Show only selected model objects

As an alternative to hiding single objects in a model view, you can define which objects you want to keep visible. All of the other, unselected objects will be hidden.

1. Select the objects that you want to keep visible.



2. Right-click and select **Show Only Selected**.
The unselected objects become almost transparent.



TIP To completely hide the unselected objects, hold down the **Shift** key when selecting the command.

To show the unselected parts as sticks, hold down the **Ctrl** key when selecting the command.

3. To make the objects visible again, click  on the **View** tab.

See also

[Hide model objects \(page 587\)](#)

Temporarily display assembly and component objects

You can temporarily show the contents of an assembly or a component even if some of the assembly or component objects are not visible in a model view.

To	Do this
Show the contents of an assembly	<ol style="list-style-type: none"> 1. Right-click an assembly, or a part in the assembly. 2. Select Assembly --> Show Assembly. For a concrete part, select Show Assembly. <p>Tekla Structures shows an orange box around the assembly, and displays all parts, bolts, welds, and other details (not cuts or fittings) belonging to the assembly, even if you had defined them as hidden in the display settings (page 918).</p> <p>For concrete parts, Tekla Structures displays reinforcement and surface treatment (not surfaces), even if you had defined them as hidden in the display settings.</p>
Show the contents of a component	<ol style="list-style-type: none"> 1. On the View tab, click Rendering --> Show component content . 2. Select a component. <p>Tekla Structures displays all bolts, welds, and other details belonging to the component, even if you had defined them as hidden in the display settings (page 918).</p>
Reapply the display settings (page 918) and make assembly or component objects hidden again	<p>On the View tab, click  .</p>

See also

[Display detailing of a part \(page 590\)](#)

[Set the visibility and appearance of model objects \(page 582\)](#)

Display detailing of a part

In some cases it is useful to see in the model all the objects that are connected to a part, such as components, welds, fittings, reinforcement, and surfaces. You can then examine, for example, whether parts are welded correctly.

1. Select a part.
2. Click  **Display detailing** on the contextual toolbar.
Alternatively, you can press **Alt+D**, or use **Quick Launch**.

Tekla Structures displays all bolts, welds, cuts, fittings, and other details belonging to the part, even if you had defined them as hidden in the [display settings \(page 918\)](#). For concrete parts, Tekla Structures displays also reinforcement, surface treatment, and surfaces.

See also

[Temporarily display assembly and component objects \(page 589\)](#)

[Set the visibility and appearance of model objects \(page 582\)](#)

Show parts in a selected view angle

In some cases it is useful to examine parts in a selected view angle. For example, when you reinforce concrete elements, you can easily check the distances between the reinforcing bars.

1. Select a part.
2. Click  **View angle** on the contextual toolbar.
3. Select a top, back, right, bottom, front, or left view.

Tekla Structures displays the part in the selected view angle. The view angle is based on the selected part's coordinate system so that the top view is looking to the negative z-direction. For example, if you select the top view, Tekla Structures sets the current view angle from up to down in the coordinate system.

4. To return to the original 3D view, click the button in the middle of the view angle options.

NOTE Tekla Structures shows the part in the current view, which is the view where your mouse has last been. If the contextual toolbar is located, for example, on top of two views, the part is shown in the view where your mouse has last been and not in the view where the part was selected.

See also

[Show and hide model objects \(page 581\)](#)

3.2 Create object groups

You can group parts and other objects based on their properties. Use object groups to control the color and transparency of parts in the model. Object groups are also needed in model view filters, selection filters, **Organizer** filters, and with the **Project Status Visualization** tool.

Create an object group

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Click **Object group...** to open the **Object Group - Representation** dialog box.
3. Select an existing Object Group from the **Save/Load** list to create a modified version or click **New filter** to start without existing settings.
4. Click **Add row** or continue to modify the settings on an existing row.
5. Select options from the **Category**, **Property**, and **Condition** lists.
You can use the same [object properties \(page 169\)](#) and [techniques \(page 166\)](#) as in filtering.
6. In the **Value** list, enter a value or select one from the model.
The values can be complete strings, such as the profile name UC310*97. You can also use incomplete strings together with [wildcards \(page 186\)](#). For example, the value UC* will match with all parts whose profile name begins with the characters UC*. Empty values are matched to empty object properties.
If you use multiple values, separate the strings with blank spaces (for example, 12 5). If a value consists of multiple strings, enclose the entire value in quotation marks (for example, "custom panel"), or use a question mark (for example, custom?panel) to replace the space.
7. Use the [And/Or options and brackets \(page 166\)](#) to define how multiple rows work together.
8. To temporarily disable rules without deleting them, you can deselect the check boxes in the first column of the row. Select the check box to re-enable the rule.
9. Enter a unique name in the box next to the **Save as** button.
10. Click **Save as** to save the object group.

Copy an object group to another model

1. Select the object group you want to copy.
The object groups you have created are located in the model's `\attributes` folder, and they have the file name extension `.PObjGrp`.
2. Select where you want to copy the object group.
 - To make an object group available in another model, copy the file to the `\attributes` folder of the destination model.

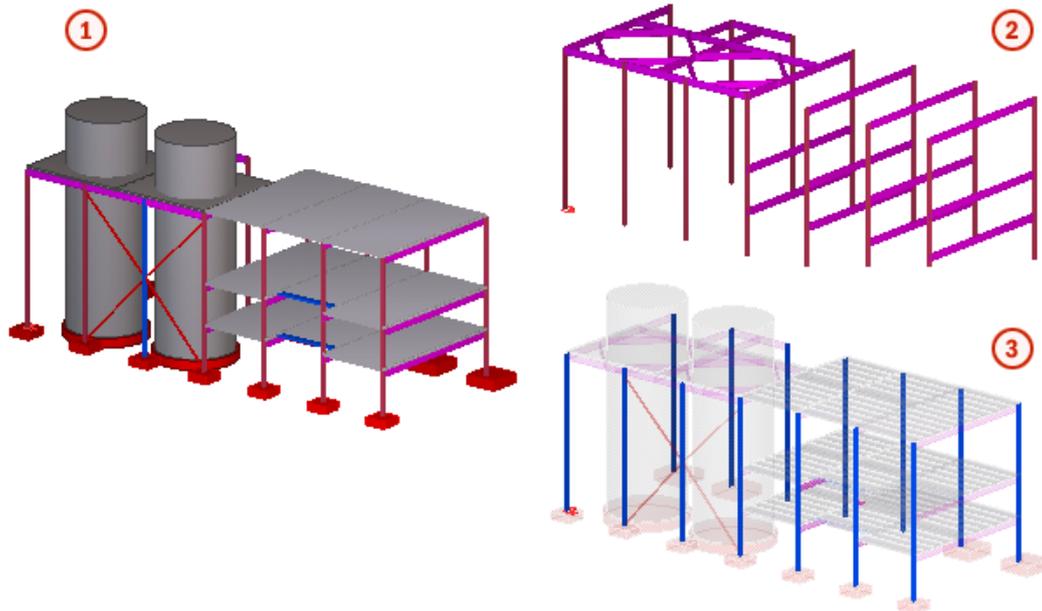
- To make an object group available in all models, copy the file to the project or firm folder, defined by the advanced option XS_PROJECT or XS_FIRM.
3. Restart Tekla Structures.

Delete an object group

1. Delete the object group file located in the model's \attributes folder. Object groups have the file name extension *.PObjGrp.
2. Restart Tekla Structures.

3.3 Change the color and transparency of model objects

You can modify the color and transparency of model objects and create customized presentations of the model. The following images show the same model with different transparency settings:



1. Standard color and transparency settings
2. Only parts whose profile name starts with IPE* or HEA* are visible
3. Parts whose user-defined attribute **Planned erection date** is set to a certain date are displayed in blue, while all other parts are 90% transparent

Click the links below to find out more:

[Change the color of a model object \(page 594\)](#)

[Change the color of an object group \(page 595\)](#)

[Define color and transparency settings \(page 597\)](#)

[Copy color and transparency settings to another model \(page 598\)](#)

[Delete color and transparency settings \(page 598\)](#)

Change the color of a model object

You can change the color of individual objects in a model by modifying their class. Alternatively, you can use the object representation settings to specify colors for entire object groups.

The possible class numbers range between 0 and 14, and result in different colors as follows:

	Class 0
	Class 1
	Class 2
	Class 3
	Class 4
	Class 5
	Class 6
	Class 7
	Class 8
	Class 9
	Class 10
	Class 11
	Class 12
	Class 13
	Class 14

Class numbers above 14 produce the same colors as 1...14. For example, class numbers 2, 16, 30, 44, and so on, all result in red.

You can also use class numbers to define the default color of pour objects and pour breaks.

To change the color and class of a part or reinforcement:

To	Do this
Change object color on the contextual toolbar	<ol style="list-style-type: none">1. Select a model object.2. On the contextual toolbar, select a new class.
Change object color in the object properties	<ol style="list-style-type: none">1. Double-click a model object to open its properties.

To	Do this
	2. In the Class box, select a new class. 3. Click Modify .

See also

[Change the color of an object group \(page 595\)](#)

[Define your own colors for object groups \(page 595\)](#)

Change the color of an object group

You can customize the color of model objects by selecting a specific color for each object group. You can use an unlimited number of colors. This option gives you more freedom in visualizing the different types of objects in the model.

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Select an object group from the **Object group** list.
3. In the **Color** list, select a [color \(page 921\)](#).
4. In the **Transparency** list, select the desired [transparency \(page 922\)](#) option.
5. Click **Save** to save the changes.
6. Click **Modify** to change the color of objects in the model.

See also

[Create object groups \(page 591\)](#)

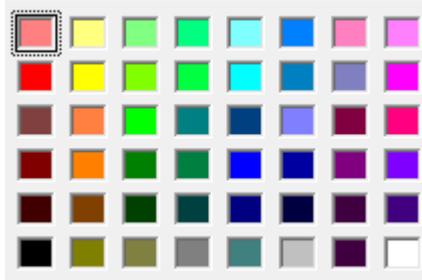
[Define your own colors for object groups \(page 595\)](#)

[Change the color of a model object \(page 594\)](#)

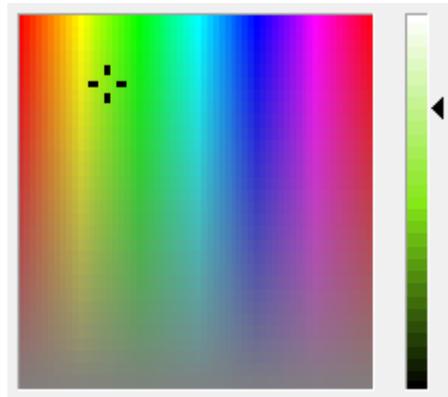
Define your own colors for object groups

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Select an object group from the **Object group** list.
3. In the **Color** list, select **Choose color...**
4. Do one of the following:

- Click a color in the **Basic colors** palette.



- Click **Define Custom Colors** and create a color of your own:
 - a. Click a color in the color window.

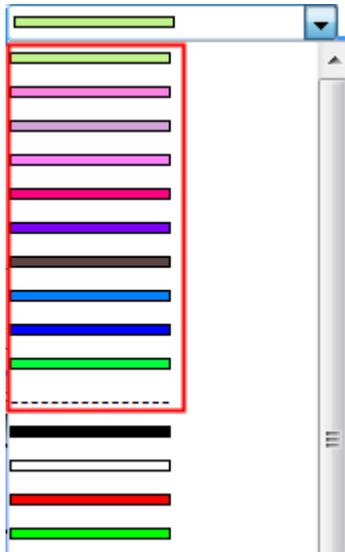


- b. Define the color depth by using the color bar on the right, or enter the exact RGB values.
- c. Click **Add to Custom Colors**.
- d. Click the color in the **Custom colors** palette to select it.



5. Click **OK**.
6. Click **Save** to save the changes.

When you open the **Object Representation** dialog box the next time, the **Color** list shows a maximum of 10 colors that you last defined. Customized colors are shown above the dashed line:



Information about the colors that you define for the object groups is stored in the `used_custom_colors.clr` file, which is located in the `\attributes` folder under the model folder. Information about the colors that you add in the **Custom colors** palette is stored in the `xs_user.xxx` file in the model folder (`xxx` is your user name).

See also

[Change the color of an object group \(page 595\)](#)

Define color and transparency settings

You can define color and transparency settings for parts and other model objects.

1. On the **View** tab, click **Representation** to open the **Object Representation** dialog box.
2. Click **Add row**.
3. Select an object group from the **Object group** list.
4. Use the **Color** list to define the color of the objects.
5. Use the **Transparency** list to define the transparency of the objects.
6. Repeat steps 3–5 for each row you add.

7. Use the **Move up** and **Move down** buttons to change the order of the rows.
If an object belongs to several object groups, the color and transparency setting defined on the top row is applied to that object.
8. Enter a unique name in the box next to the **Save as** button.
9. Click **Save as** to save the settings.

NOTE If your setting does not contain the group **All**, Tekla Structures adds that row to the bottom of the list when you click **Modify**, **Apply** or **OK**.

See also

[Change the color and transparency of model objects \(page 593\)](#)

[Color settings for object groups \(page 921\)](#)

[Transparency settings for object groups \(page 922\)](#)

[Define your own colors for object groups \(page 595\)](#)

Copy color and transparency settings to another model

1. Select the settings you want to copy.
The settings you have created are located in the model's `\attributes` folder, and they have the file name extension `.rep`.
2. Select where you want to copy the settings.
 - To make the settings available in another model, copy them to the `\attributes` folder of the destination model.
 - To make the settings available in all models, copy them to the project or firm folder, defined by the advanced option `XS_PROJECT` or `XS_FIRM`.
3. Restart Tekla Structures.

See also

[Change the color and transparency of model objects \(page 593\)](#)

Delete color and transparency settings

1. Delete the `.rep` file located in the model's `\attributes` folder.
2. Restart Tekla Structures.

See also

[Change the color and transparency of model objects \(page 593\)](#)

4 Check the model

This section describes a variety of tools you can use to ensure the model does not contain errors.

Click the links below to find out more:

[Inquire object properties \(page 599\)](#)

[Measure objects \(page 605\)](#)

[Compare parts or assemblies \(page 607\)](#)

[Create a clip plane \(page 608\)](#)

[Fly through the model \(page 609\)](#)

[Detect clashes \(page 610\)](#)

[View solid errors \(page 623\)](#)

[Diagnose and repair the model \(page 624\)](#)

[Find distant objects \(page 626\)](#)

4.1 Inquire object properties

Use the **Inquire** commands to get information about a particular object, or a group of objects, within the model.

Do any of the following:

To inquire	Do this
Object properties	<ol style="list-style-type: none">1. On the ribbon, click  Object.2. Select an object. Tekla Structures displays the object properties in a separate window.

To inquire	Do this
Point coordinates	<ol style="list-style-type: none"> On the ribbon, click the down arrow next to , and then select Point coordinates. The Inquire point coordinates dialog box is displayed. Click Pick, and then pick a point in the model to see the coordinates of the point in: <ul style="list-style-type: none"> Local coordinates Model coordinates (global) Project base point coordinates Current base point coordinates
Center of gravity	<ol style="list-style-type: none"> On the ribbon, click the down arrow next to , and then select Center of gravity. Select one or more parts. Tekla Structures creates a point at the center of gravity of each selected part, and displays information about the center of gravity in a separate window.
Object properties using custom reports	See Custom inquiry (page 602) .
Welded parts	<ol style="list-style-type: none"> On the ribbon, click the down arrow next to , and then select Welded parts. Select a part. Tekla Structures highlights the selected part and all the parts that are welded to it.
Primary welded parts	<ol style="list-style-type: none"> On the ribbon, click the down arrow next to , and then select Primary welded part. Select a part. Tekla Structures highlights the primary part when you select a secondary part.
Assembly or cast unit objects	See Check and highlight objects in an assembly (page 393) or Check and highlight objects in a cast unit (page 398) .
Component objects	<ol style="list-style-type: none"> On the ribbon, click the down arrow next to , and then select Component objects.

To inquire	Do this
	<p>2. Select a component.</p> <p>Tekla Structures highlights all objects belonging to the selected component.</p>
Phases	<p>On the ribbon, click the down arrow next to , and then select Phases.</p> <p>Tekla Structures displays information about objects in different phases in a separate window.</p>
Model size	<p>On the ribbon, click the down arrow next to , and then select Model size.</p> <p>Tekla Structures displays the quantity of all objects in the current model in a separate window.</p>

See also

[Object property report templates \(page 601\)](#)

Object property report templates

When you view object properties using the **Inquire object** command, Tekla Structures uses the following report templates, available in the folder `.. \ProgramData\Trimble\Tekla Structures\<version>\environments \common\system:`

Object type	Template
Assemblies	TS_Report_Inquire_Assembly.rpt
Bolts	TS_Report_Inquire_Bolt.rpt
Cast units	TS_Report_Inquire_Cast_Unit.rpt
Parts	TS_Report_Inquire_Part.rpt
Pour breaks	TS_Report_Inquire_Pour_Break.rpt
Pour objects	TS_Report_Inquire_Pour_Object.rpt
Reinforcement meshes	TS_Report_Inquire_Rebar_Mesh.rpt
Reinforcement strands	TS_Report_Inquire_Rebar_Strand.rpt
Reference models	TS_Report_Inquire_Reference.rpt
Reinforcement	TS_Report_Inquire_Reinforcement.rpt
Surfaces	TS_Report_Inquire_Surface.rpt
Welds	TS_Report_Inquire_Welding.rpt

You can modify these templates according to your needs. For more information on how to use templates, see the Template Editor user documentation.

You can also create a custom template for connections and details by saving the template with the name `TS_Report_Inquire_Connection.rpt`.

See also

[Inquire object properties \(page 599\)](#)

Custom inquiry

You can use the **Custom inquiry** command to display information about the selected model object in the side pane. You can define what information you want to display.

Use the Custom Inquiry tool

1. Click the **Custom Inquiry** button  in the side pane.

Alternatively, you can click the down arrow next to  on the ribbon, and then select **Custom inquiry**.

The **Custom Inquiry** window opens in the side pane.

2. In the **Report type** list, select the report template you want to use for showing the object information.
3. Select a model object.

Tekla Structures displays the object properties in the side pane.

If you select several objects or object types, for example parts, bolts, and reinforcing bars, Tekla Structures displays the quantity of all the selected objects, regardless of the object types, or the report template used. For the object properties that differ, Tekla Structures shows **Varies**.

Define what information is shown by Custom Inquiry tool

You can define what information is displayed in the **Custom Inquiry** side pane window. You can add and modify report templates and the attributes in them.

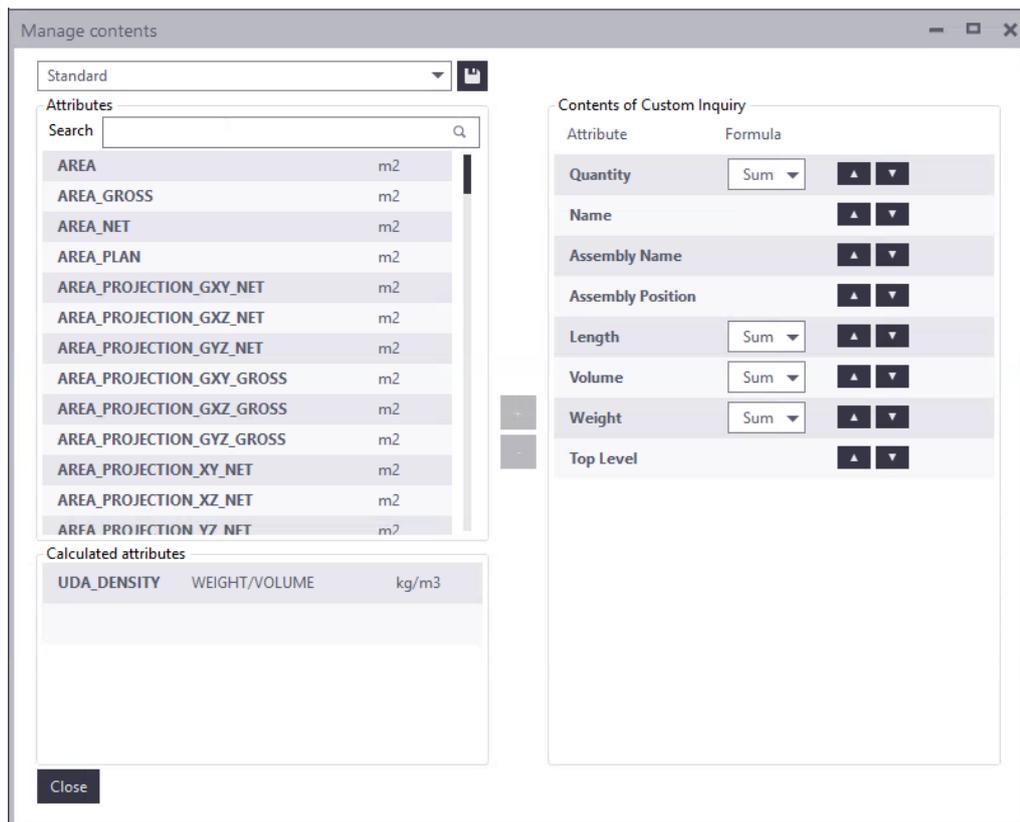
1. Click the **Custom Inquiry** button  in the side pane.

Alternatively, you can click the down arrow next to  on the ribbon, and then select **Custom inquiry**.

The **Custom Inquiry** window opens in the side pane.

2. Click the  button.

The **Manage contents** dialog box appears.



The **Attributes** list contains the attributes that are available by default. In the **Calculated attributes** area you can create your own attribute formulas. The **Contents of Custom Inquiry** list contains the attributes whose values will be displayed in the side pane.

3. Define which report templates and attributes are available.
 - To modify an existing report template, select it from the top left-hand list in the **Manage contents** dialog box.
 - To create a new report template, enter a name in the box next to the  button, and then click .
 - To modify the default attributes, edit the `InquiryTool.config` file.
 - To create or modify a calculated attribute, double-click a cell in the **Calculated attributes** area. In the first cell, type the name of the

- attribute. In the second cell, use attribute names and standard mathematical symbols (+, -, *, and /) to form equations.
4. Define which attributes are shown in the **Custom Inquiry** side pane window.
 - To add more attributes to the side pane, select an attribute in the **Attributes** list and then click the  button.
 - To remove attributes from the side pane, select an attribute in the **Contents of Custom Inquiry** list and then click the  button.
 - To change the order of the attributes, use the   buttons.
 - To change the formula of an attribute, click the down arrow and select a different formula (**Sum**, **Average**, **Max**, or **Min**) from the list.
 5. Click  to save the changes.

Modify the default attributes in InquiryTool.config file

Use the `InquiryTool.config` file to control which attributes are shown as default attributes in the **Manage contents** dialog box in the **Custom Inquiry** tool.

NOTE This section is for advanced users.

Tekla Structures searches for the `InquiryTool.config` file in the following folders in the following order:

1. `\attributes` folder in the model folder
2. `\CustomInquiry` subfolder in the folder defined by `XS_PROJECT`
3. `\CustomInquiry` subfolder in the folder defined by `XS_FIRM`
4. `\CustomInquiry` subfolder in the folder defined by `XS_SYSTEM`

If you specify several folders that have `\CustomInquiry` as a subfolder, Tekla Structures uses the first folder it finds.

To add new attributes to the `InquiryTool.config` file:

1. Open the `InquiryTool.config` file in any standard text editor.
2. Copy the entire contents of `[ATTR_CONTENT_??]` to the end of the file.
3. Change the position number of the new attribute.
For example, change `[ATTR_CONTENT_??]` to `[ATTR_CONTENT_66]`.
4. Modify the `NAME`, `DISPLAY_NAME`, `DATATYPE`, `UNIT`, and `DECIMAL` values of the new attribute. Use the attribute names and definitions that are

included in the `contentattributes_global.lst` or `contentattributes_userdefined.lst` file.

5. Change the `TOTAL_ATTR_CONTENT` value to reflect the total number of attributes in the file.

For example, change `TOTAL_ATTR_CONTENT=65` to `TOTAL_ATTR_CONTENT=66`.

6. Save the file.

4.2 Measure objects

Use the **Measure** commands to measure angles, arcs, the distance between two points and between bolts in the model.

All measurements are temporary. The measurements appear in the model view window until you [update or redraw \(page 43\)](#) the window.

The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

Measure distances

You can measure horizontal, vertical and user-defined distances in the model.

1. Press **Ctrl+P** to switch to the plane view.
2. On the **Edit** tab, click **Measure** and select one of the following commands:
 - **Distance**

This command measures the distance between any two points. Use this command to measure inclined or aligned distances. By default, the results contain the distance and the coordinates.
 - **Horizontal distance**

This command measures the distance between two points in the direction of the view plane x axis.
 - **Vertical distance**

This command measures the distance between two points in the direction of the view plane y axis.
3. Pick the start point.
4. Pick the end point.

- Pick a point to indicate on which side of the dimension line you want the measurement to appear.

The measurement is displayed until the next window [update or redraw](#) (page 43).

Measure angles

You can measure angles in the model.

- On the **Edit** tab, click **Measure** --> **Angle** .
- Pick the center point.
- Pick the start point.
- Pick the end point.

The measurement is displayed until the next window [update or redraw](#) (page 43).

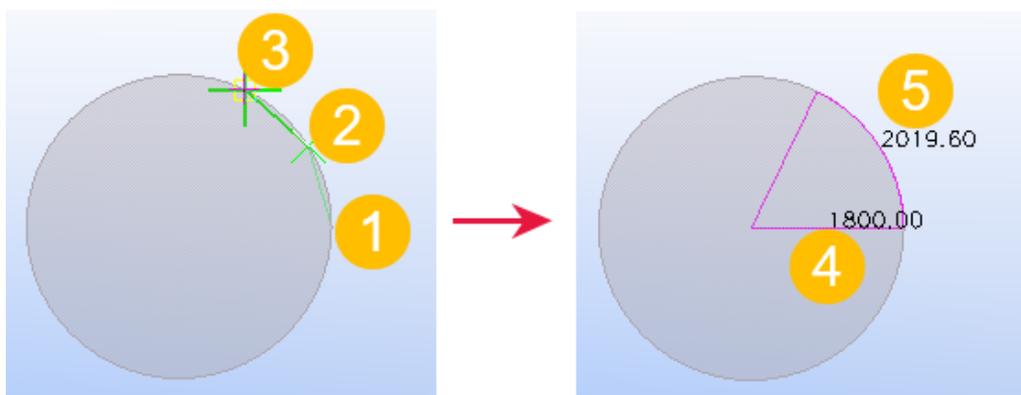
Measure arcs

You can measure the radius and length of an arc in the model.

- On the **Edit** tab, click **Measure** --> **Arc** .
- Pick the start point.
- Pick the middle point.
- Pick the end point.

This can be any point along the arc between the start and end points.

The measurement is displayed until the next window [update or redraw](#) (page 43).



- (1) Start point
- (2) Middle point
- (3) End point

(4) Arc radius

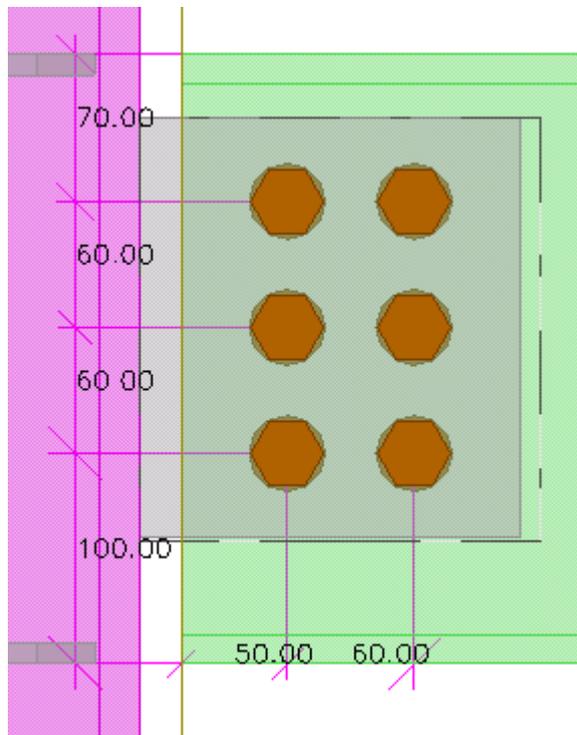
(5) Arc length

Measure bolt spacing

You can measure distances between bolts in a bolt group. Tekla Structures also gives you the edge distances between the bolts and a selected part.

1. On the **Edit** tab, click **Measure** --> **Bolt spacing**.
2. Select a bolt group.
3. Select a part.

The measurement is displayed until the next window [update or redraw \(page 43\)](#).



4.3 Compare parts or assemblies

You can compare two selected parts or assemblies.

1. Select the objects you want to compare.
 - To compare parts, select two parts in the model.
 - To compare assemblies, select a part in each assembly.

2. On the **Edit** tab, click **Compare**, and then select either **Compare parts** or **Compare assemblies**.

Tekla Structures displays the results on the status bar.

See also

[Check the model \(page 599\)](#)

4.4 Create a clip plane

Clip planes enable you to focus on the required detail in the model. You can create up to six clip planes in any model view that shows the object faces.

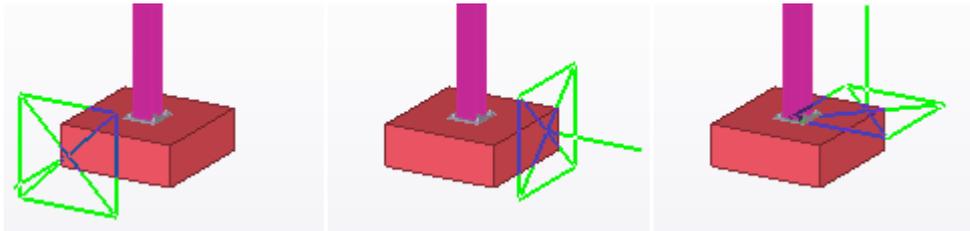
1. When you create clip planes, ensure that you are using a model view that shows object faces.

On the **View** tab, click **Rendering**, and use any of the following options:

- **Parts grayscale** (Ctrl+3)
- **Parts rendered** (Ctrl+4)
- **Components grayscale** (Shift+3)
- **Components rendered** (Shift+3)

2. On the **View** tab, click **Clip plane** .
3. Move the mouse pointer over the model objects.

A green symbol indicates the object faces that you can select and align the clip plane with. The green line indicates the side that will be clipped out. For example:



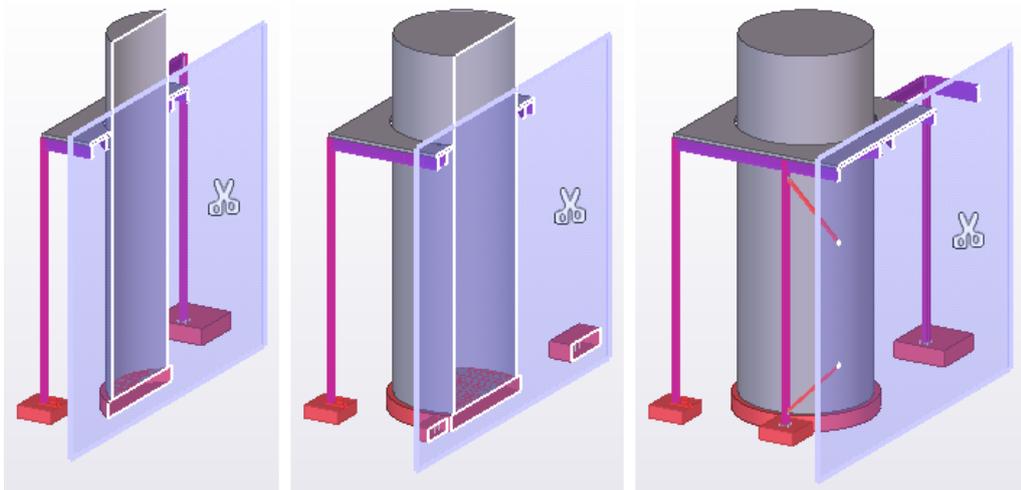
Note that the [snap depth \(page 78\)](#) setting on the **Snapping** toolbar affects the object faces you can select. Set snap depth to **3D** or **Auto** to select object faces in the entire 3D space.

4. Select an object face.

The clip plane symbol appears in the model:



5. Repeat step 4 to create as many clip planes as needed.
6. To finish creating clip planes, press **Esc**.
7. To move a clip plane, select the clip plane scissor symbol and drag it to a new location.



8. If you want to move the clip plane scissor symbol to a new location on a clip plane, hold down **Shift** and drag the symbol.
This does not move the clip plane, only the scissor symbol.
9. You can delete the clip planes when you do not need them anymore.
 - To delete a single clip plane, select the clip plane symbol and press **Delete**.
 - To delete all the clip planes in all the open views, click **View --> Delete all clip planes**.

4.5 Fly through the model

Using the **Fly** command, you can travel through a model, changing direction and varying the speed as you go. You can also adjust the field of view setting, which can be useful when flying in a tight space.

1. Set view projection to **Perspective**.
 - a. Double-click the view to open the **View Properties** dialog box.
 - b. In the **Projection** list, select **Perspective**.
 - c. Click **Modify**.

2. If needed, adjust the field of view setting.

The bigger the value, the more distance there is between the parts when you fly through the model.

- a. On the **File** menu, click **Settings** --> **Advanced options** , and go to the **Model View** category.
- b. Modify the advanced option `XS_RENDERED_FIELD_OF_VIEW`.
- c. Click **OK**.

3. On the **View** tab, click **Fly**.

4. Select a view.

The mouse pointer changes into an arrow and a cross. The arrow indicates the current flying direction.



5. Drag the mouse to move around in the model.

- To fly forward, move the mouse forward.
- To change the flying direction, drag the mouse in the desired direction.

The flying speed grows exponentially when you are approaching the model from a distance.

- To move up or down, hold down **Ctrl** and drag the mouse forward or backward.
- To change the camera angle, scroll with the mouse wheel.
- To fly in the in the direction of the camera angle, hold down **Shift** and scroll forward or backward.

6. To stop flying, press **Esc**.

4.6 Detect clashes

Use the **Clash check manager** tool to find parts, bolts, reinforcement, or reference model objects that collide. Clashes of objects that only touch one another are not included in the clash check results.

Flag	Number	Type	Status	Priority	Date Modified	Object ID	Assembly ID	Object Name
	1	Clash	Assigned	High	11.6.2015 10:31	269; 417	274; 419	BEAM (2)
	2	Clash	Ignored	Low	11.6.2015 10:30	269; 346	274; 352	BEAM; BEAM1
✓	3	Clash			11.6.2015 10:25	269; 298	274; 299	BEAM (2)
	4	Clash		Medium	11.6.2015 10:42	269; 523	274; 526	BEAM (2)
⚠	5	Is inside			11.6.2015 10:29	417; 523	419; 526	BEAM; BEAM2
?	6	Clash	Fixed		11.6.2015 10:31	269; 590	274; 594	BEAM (2)
✨	7	Clash		Medium	11.6.2015 10:43	269; 633	274; 637	BEAM; BEAM1

Use the clash check settings to define clearances between different model objects.

You can also use the sections and floors created in **Organizer** to run a controlled clash check.

If you want to use another Tekla Structures model as a reference model, you must export it in IFC format to be able to use it in clash checking. The following reference model file types are supported in clash checking:

- IFC
- DWG
- DGN

See also

[Find clashes in a model \(page 611\)](#)

Find clashes in a model

1. On the **Manage** tab, click **Clash check** .
2. In the model, select the objects you want to include in the clash check.
3. Click  to check the objects.

You can continue working during the clash check. When the clash check is complete, the status bar message changes from **Clash checking in progress – Press Esc to cancel** to **Ready**.

4. To highlight a clash in the model, select a row in the list of clashes.
The related model objects are selected.

5. To zoom the active view so that the selected objects are shown in the center of the view, double-click a row.
6. If you want to include more objects in the clash check, select the desired model objects and re-run the clash check.
New clashes are appended to the end of the list.
7. After removing or modifying objects, re-run clash check to see if the clash still exists.
 - a. Select the desired rows in the list of clashes.
 - b. Click  to re-run clash check.

NOTE For best results, run the clash check only for appropriate sections and floors, not for the entire model. Use **Organizer** to select the sections and floors for which you want to run the clash check. Right-click and select **Select in the model**.

NOTE If you cannot find the clashes in the model, change the rendering of objects to **Show only selected part (Ctrl+5)** for better visibility.

See also

[Manage the list of clashes \(page 616\)](#)

Manage clash check results

This section describes how to interpret the symbols and clash types used in clash checking and how to change the status or priority of the clashes.

Click the links below to find out more:

[Symbols used in clash checking \(page 612\)](#)

[About clash types \(page 613\)](#)

[Manage the list of clashes \(page 616\)](#)

[Search for clashes \(page 616\)](#)

[Change the status of clashes \(page 617\)](#)

[Change the priority of clashes \(page 617\)](#)

Symbols used in clash checking

Clash Check Manager uses the following flags to indicate the state of clashes:

Flag	State	Description
(none)	Active	The default state. The clash is not new, modified, resolved, or missing.
	New	All clashes are marked as new when they are found for the first time.
	Modified	If the object has been modified (for example, if the profile has been changed), the state changes to modified when you re-run the clash check. Only certain object properties affect this flag. To see which properties have an effect, right-click one of the column headings. Both visible and hidden properties affect the flag.
	Resolved	If the objects no longer clash, the state changes to resolved when you re-run the clash check.
	Missing	If one or both of the clashing objects have been removed from the model, the state changes to missing when you re-run the clash check.

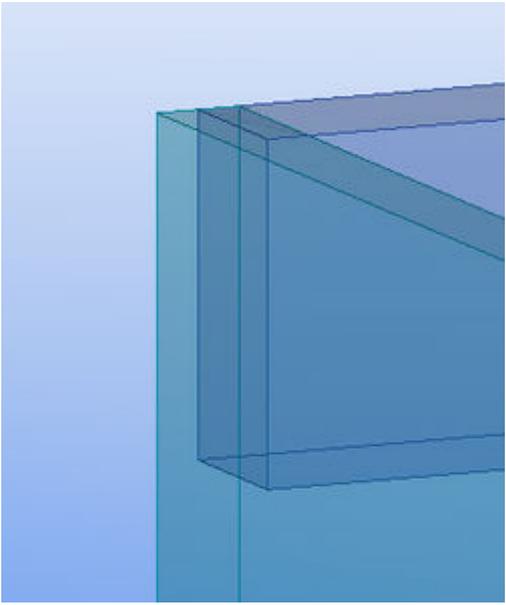
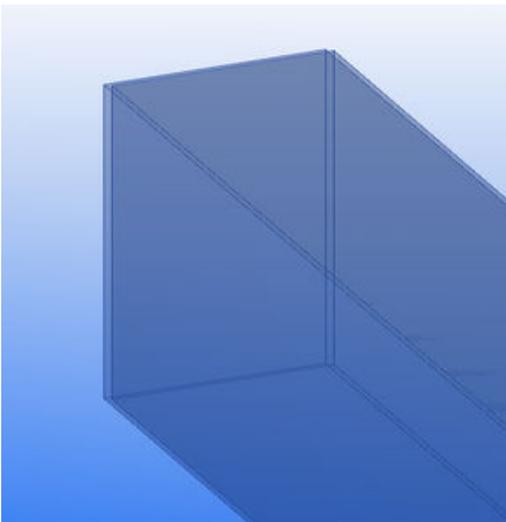
See also

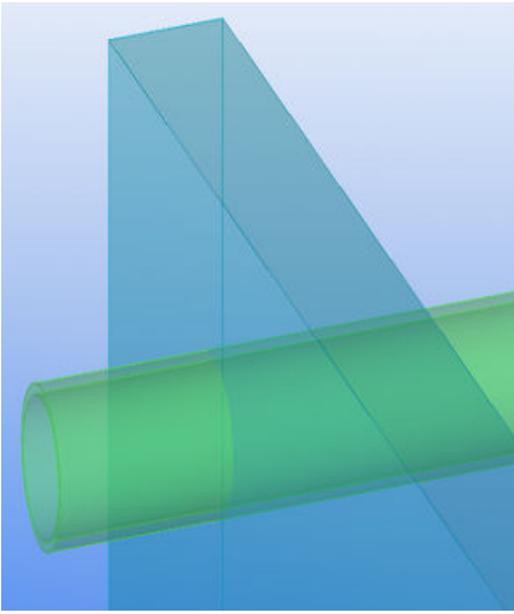
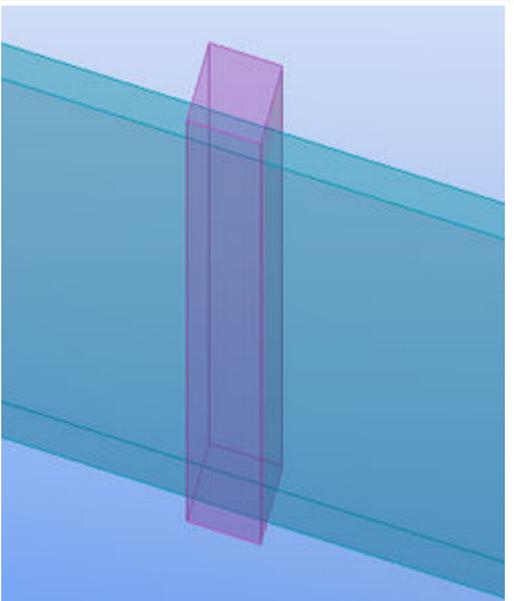
[Find clashes in a model \(page 611\)](#)

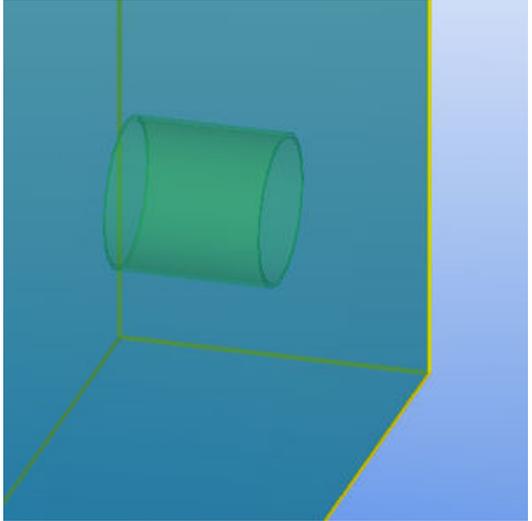
About clash types

Tekla Structures shows the type of each clash in the **Type** column in the **Clash Check Manager** dialog box.

The following clash types can occur:

Type	Description	Example
Clash	The object partly overlaps with another object.	
Clash	Two identical objects completely overlap.	

Type	Description	Example
Clash	Objects intersect with each other in several locations.	
Clash	The object cuts through another object.	

Type	Description	Example
Is inside	The object is inside another object.	

See also

[Find clashes in a model \(page 611\)](#)

[Manage clash check results \(page 612\)](#)

Manage the list of clashes

To manage the list of clashes in **Clash Check Manager**:

To	Do this
Change the sort order of clash results	Click the heading of the desired column to alternate between the ascending and descending sort order.
Select multiple rows in the list of clashes	Hold down Ctrl or Shift while selecting rows.
Show or hide a column	<ol style="list-style-type: none"> 1. Right-click one of the column headings to open a menu. 2. Click any of the list items to show or hide them. <p>A check mark <input checked="" type="checkbox"/> in front of the item indicates that it is visible.</p>

See also

[Find clashes in a model \(page 611\)](#)

Search for clashes

Use the **Search** box to find clashes based on search terms. The more search terms you enter, the more refined your search will be. For example, if you enter `column 8112`, only the clashes that match both of these terms are displayed.

1. Open the clash check session you want to find clashes from.
2. In the **Search** box, enter the words to search for.
The search results are displayed as you type.
3. To narrow your search, enter more characters.
4. To display all the clashes again, click **×** next to the **Search** box.

See also

[Find clashes in a model \(page 611\)](#)

Change the status of clashes

1. In **Clash Check Manager**, select the clashes whose status you want to change.
2. Right-click one of the selected rows.
3. Select **Status** and then one of the status options:
 - **Assign**
 - **Fix**
 - **Approve**
 - **Ignore**
 - **Re-open**

See also

[Find clashes in a model \(page 611\)](#)

Change the priority of clashes

1. In **Clash Check Manager**, select the clashes whose priority you want to change.
2. Right-click one of the selected rows.
3. Select **Priority** and then one of the status options:
 - **High**
 - **Medium**
 - **Low**

See also

[Find clashes in a model \(page 611\)](#)

Group and ungroup clashes

You can combine several clashes into a group so that the clashes are treated as a single unit.

1. In **Clash Check Manager**, select the clashes you want to group.
2. Right-click and select **Group --> Group** .
3. If you want to add clashes to an already existing group, select the clashes and the group, and then repeat step 2.

NOTE You cannot create nested groups of clashes.

4. If you want to ungroup clashes:
 - a. Select the clash group to ungroup.
 - b. Right-click and select **Group --> Ungroup** .

See also

[Detect clashes \(page 610\)](#)

View the details of a clash

Use the **Clash Information** dialog box to view more detailed information about a clash.

For example, you can view the profile, material and class of the objects that collide. This can be useful especially when you view [clash groups \(page 618\)](#) that contain more than two objects.

1. Select the clash or clash group whose details you want to view.
2. Right-click and select **Clash Information**.

NOTE You can view information on one clash or clash group at a time. If you select more than one clash or clash group, the **Clash Information** option is disabled.

See also

[Add comments to a clash \(page 619\)](#)

[View the history of a clash \(page 620\)](#)

Add comments to a clash

You can add comments to clashes and clash groups. For example, you might use comments as reminders to yourself and other users.

1. Select the clash or clash group you want to comment on.
2. Right-click and select **Clash Information**.
3. Go to the **Comments** tab.
4. Click  to open the **Add comment** dialog box.
5. Enter your comment in the **Comment** box.
6. Modify the author name and the date if needed.
7. Click **OK**.

See also

[Modify a clash comment \(page 619\)](#)

[Remove a clash comment \(page 619\)](#)

Modify a clash comment

1. Select the clash or [clash group \(page 618\)](#) whose comment you want to modify.
2. Right-click and select **Clash Information**.
3. Go to the **Comments** tab.
4. Select the comment you want to modify.
5. Click  to open the **Edit comment** dialog box.
6. Modify the comment.
7. Click **OK**.

See also

[Add comments to a clash \(page 619\)](#)

[Remove a clash comment \(page 619\)](#)

Remove a clash comment

1. Select the clash or [clash group \(page 618\)](#) whose comment you want to remove.
2. Right-click and select **Clash Information**.
3. Go to the **Comments** tab.

4. Select the comment you want to remove.
5. Click .

See also

[Add comments to a clash \(page 619\)](#)

[Modify a clash comment \(page 619\)](#)

View the history of a clash

You can view the history of a particular clash. For example, you can see when and by whom the clash was detected.

1. Select a clash or a [clash group \(page 618\)](#).
2. Right-click and select **Clash Information**.
3. Go to the **History** tab.
The history information of the clash is displayed.

See also

[View the details of a clash \(page 618\)](#)

Print a list of clashes

You can print lists of clashes. You can control the print settings the same way as in any standard Windows application.

1. Open the clash check session you want to print.
2. Click  **Print...**
3. If needed, modify the print settings.
4. Click **Print....**

See also

[Preview a list of clashes before printing \(page 620\)](#)

[Set the paper size, margins and page orientation \(page 621\)](#)

Preview a list of clashes before printing

Use the options in the **Print preview** dialog box to see what a list of clashes will look like when it is printed.

To preview a list of clashes:

To	Do this
Open the Print preview... dialog box	In Clash Check Manager , click the down arrow next to  and select Print preview...
Select how many pages to view at a time	Click one of the Page Layout buttons:  If the list of clashes is very long, it might be spread out on several pages.
Zoom the page in or out	Click the down arrow next to  and select an option from the menu.
Print the current page	Click  .
Close the Print preview... dialog box.	Click Close .

See also

[Print a list of clashes \(page 620\)](#)

[Set the paper size, margins and page orientation \(page 621\)](#)

Set the paper size, margins and page orientation

Before printing a list of clashes, you can set the paper size, margins, and page orientation in the **Page Setup** dialog box.

NOTE The options for paper size and paper source are printer-dependent. To use different paper options, select another printer in the **Print** dialog box and click **Apply**.

1. Click the arrow ▼ next to the  button and select **Page setup**.
2. In the **Size** box, select the paper size you want to use.
3. In the **Source** box, select the correct paper source.
4. Under **Orientation**, select one of the page orientation options.
 - **Portrait:** vertical page orientation
 - **Landscape:** horizontal page orientation
5. Under **Margins**, enter values for the **Left**, **Right**, **Top**, and **Bottom** margins.
6. Click **OK** to save the changes.

See also

[Print a list of clashes \(page 620\)](#)

[Preview a list of clashes before printing \(page 620\)](#)

Open and save clash check sessions

Clash check sessions are saved as XML files in the . .

\TeklaStructuresModels\Clash Check Manager for the first time.

To open or save sessions in **Clash Check Manager**, do any of the following:

To	Do this
Open a session	<ol style="list-style-type: none">1. Click .2. In the Open dialog box, select a session.3. Click Open.
Start a new session	Click  . Clash Check Manager clears the list of clashes without running clash check.
Save the current session	Click  .
Save the current session with a another name or in another location	<ol style="list-style-type: none">1. Click the arrow  next to the  button. A menu appears.2. Click  Save as .3. In the Save as dialog box, browse to the folder where you want to save the session.4. In the File name box, enter a new name.5. Click Save.
Save only selected clashes	<ol style="list-style-type: none">1. In the list of clashes, select the clashes you want to save.2. Click the arrow  next to the  button. A menu appears.3. Click  Save selected .

See also

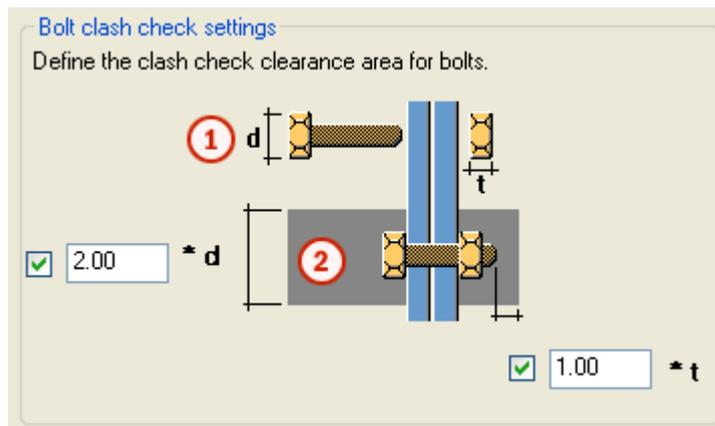
[Detect clashes \(page 610\)](#)

Define a clash check clearance area for bolts

To check if bolts collide with profiles and if there is enough space to fix the bolt, you can define a clash check clearance area for bolts.

1. On the **File** menu, click **Settings --> Options** .
2. In the **Options** dialog box, go to the **Clash check** settings.
3. Modify the bolt clearance values.

If the boxes are empty, Tekla Structures uses the default value 1 . 00.



1. **d** is the larger value of the bolt head or nut diameters
 2. Clash check clearance area
4. Ensure that you have selected the check box in front of each box.
If you clear the check boxes, the clearance will be zero.
 5. Click **Apply** or **OK**.

NOTE If Tekla Structures cannot find the bolt head or nut diameter in the bolt catalog, it uses the shank diameter instead.

See also

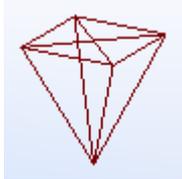
[Detect clashes \(page 610\)](#)

4.7 View solid errors

You can view solid errors in a log file. This may be needed, for example, if overlapping volumes and faces occur for parts and pour objects, and you need to analyze the error.

1. On the **File** menu, click **Logs --> Session history log** .
2. Look for rows that start with the phrase `Solid error`.
3. Click the corresponding `Solid failure position` row to show the solid object error.

A diamond-shaped position locator is displayed in the model to point you to the error.



TIP When you click a `Solid error` row in the log file, hold down the **z** key to center the view to the error location.

4. Redraw the view to hide the position locator.

See also

[Troubleshoot pours \(page 422\)](#)

4.8 Diagnose and repair the model

Use the **Diagnose & repair** commands to check and to repair errors and inconsistencies in the structure of model objects and the library database (`xslib`). By diagnosing and repairing the model you can ensure, for example, that empty assemblies are removed and that unused points and attributes are deleted. Repairing the model also corrects invalid object relations and hierarchies. We recommend you to diagnose and to repair your model regularly to maintain the consistency and integrity of your model databases.

1. On the **File** menu, click **Diagnose & repair**.
2. Select the appropriate diagnose command.

The errors and inconsistencies found in the model are listed in a report. Some of them are automatically corrected, some of them are warnings that you need to correct manually.

If a profile, material grade, bolt element or assembly, or reinforcement seems to be missing from the corresponding catalog, your Tekla Structures environment or a catalog file may not be the same as the model's original one.

The following table lists the most common errors and inconsistencies found when diagnosing the model.

Diagnosis result	Description	Action needed
Empty assembly	The assembly does not contain any objects.	<ol style="list-style-type: none"> a. On the File menu, click Diagnose & repair. b. Under Model, click Repair model to delete the assembly.
Missing assembly	A part is not included in any assembly.	<ol style="list-style-type: none"> a. On the File menu, click Diagnose & repair. b. Under Model, click Repair model to create an assembly and to move the part to it.
Illegal profile	An unknown profile has been found.	<ol style="list-style-type: none"> a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original <code>profdb.bin</code> and <code>profitab.inp</code> files and save them in the model folder. c. Reopen the model.
Illegal material	An unknown material grade has been found.	<ol style="list-style-type: none"> a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original <code>matdb.bin</code> file and save it in the model folder. c. Reopen the model.
Illegal bolt	An unknown bolt element or bolt assembly has been found.	<ol style="list-style-type: none"> a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original <code>screwdb.db</code> and <code>assdb.db</code> files and save them in the model folder. c. Reopen the model.
Illegal reinforcing bar size or grade Illegal reinforcement mesh	Reinforcement with invalid properties has been found.	<ol style="list-style-type: none"> a. Ensure that you are using the correct Tekla Structures environment. b. Use the model's original <code>rebar_database.inp</code> and <code>mesh_database.inp</code> files and save them in the model folder. c. Reopen the model.

Diagnosis result	Description	Action needed
Invalid rebar geometry	Reinforcement with undefined geometry has been found.	See Check the validity of reinforcement geometry (page 514) .

If the model does not contain any errors or inconsistencies, a message is displayed on the status bar.

See also

[Check the model \(page 599\)](#)

4.9 Find distant objects

When the work area is huge, the model may contain some distant objects that are not easy to find. Use the **Find distant objects** command to find these objects.

1. On the **File** menu, click **Diagnose & repair**.
2. Under **Utilities**, click **Find distant objects**.

Tekla Structures displays a list of object GUIDs. At the end of the list, Tekla Structures displays additional six objects that have the biggest and smallest x, y, or z coordinates.

```
Min x: Guid: e32a7a28-40db-4597-b160-031d15c1944a
Max x: Guid: 985a39e2-8097-4a9a-8706-9651d08f61c6
Min y: Guid: 8ccb2748-cfe8-4a97-be80-abf453008567
Max y: Guid: 08c8e02d-6a79-4b7e-be70-5370359a1ff5
Min z: Guid: 95eec6e2-d22b-4ae8-8c31-ee8009c028a6
Max z: Guid: f791c3d0-de62-4ced-8d79-03668296f862
```

3. Select an object in the list.
4. Right-click and select a command.
You can, for example, inquire or delete the object.

See also

[Check the model \(page 599\)](#)

5 Number the model

This section explains how to change numbering settings and apply numbering in Tekla Structures.

Click the links below to find out more:

[What is numbering and how to plan it \(page 627\)](#)

[Adjust the numbering settings \(page 636\)](#)

[Number parts \(page 637\)](#)

[Change existing numbers \(page 640\)](#)

[Clear existing numbers \(page 641\)](#)

[Check the numbering \(page 642\)](#)

[View the numbering history \(page 644\)](#)

[Repair numbering errors \(page 645\)](#)

[Renummer the model \(page 645\)](#)

[Control numbers \(page 646\)](#)

[Number parts by design group \(page 653\)](#)

[Numbering examples \(page 656\)](#)

5.1 What is numbering and how to plan it

Before you can create drawings or accurate reports, you need to number all parts in the model. You do not need to number the model before you create general arrangement drawings.

Numbering is the key to the production output, for example, drawings, reports, and NC files. Numbers are also needed when you export models. Part numbers are vital in the fabrication, shipping, and erection stages of construction. Tekla Structures assigns a mark to each part and assembly/cast unit in a model. The mark includes part or assembly prefix and position

number, and other elements, such as profile or material grade. It is useful to identify the parts with numbers to see which parts are similar and which different. Identical parts within a numbering series have the same number, which makes the planning of the production easier.

We recommend that you plan the numbering in an early phase of the project. If other users are using the same model, it is even more important to make a numbering plan that everybody in the project follows. You should have the numbering ready before you create the first drawings and reports.

When planning the numbering, it can be useful to number the model in phases, for example first floor of the building first, then the second, and so on.

Give start numbers in wide ranges so that you do not run out of numbers within a numbering series, and that any numbering series does not overlap with another. For example, start the first floor with the start number 1000 and the second with start number 2000.

If the numbering of a part or assembly is not up to date, a question mark (?) is displayed in the part label and in the **Inquire object** dialog box, for example:

```
Assembly information
-----
Assembly Pos:      C/0(?)
Main part profile: HEA400
```

See also

[Numbering series \(page 628\)](#)

[Identical parts \(page 632\)](#)

[Identical reinforcement \(page 632\)](#)

[Define what affects numbering \(page 633\)](#)

[User-defined attributes in numbering \(page 634\)](#)

[Family numbers \(page 634\)](#)

[Inquire object properties \(page 599\)](#)

Numbering series

Use numbering series to divide steel parts, assemblies, and cast units into groups. For example, you can allocate a separate numbering series to each phases or part type. Using separate numbering series for different parts speeds up the numbering operation.

The name of a numbering series consists of a *prefix* and a *start number*. You do not always have to define a part prefix (for example, you may want to omit the part prefix for minor parts).

When you run numbering, Tekla Structures compares parts that belong to the same series with each other. All identical parts in the same numbering series are given the same part number.

NOTE Concrete parts are numbered according to the cast unit numbering settings. For example, if the cast unit prefix is **C** and the start number is **1**, concrete parts will get the part prefix **Concrete_C-1**.

This applies also to concrete components whose part position prefix is **Concrete** and start number is **1**.

Example

For example, if you define a numbering series with the prefix P and start number 1001, Tekla Structures numbers that series P1001, P1002, P1003, ...

See also

[Plan your numbering series \(page 629\)](#)

[Assign a numbering series to a part \(page 630\)](#)

[Assign a numbering series to an assembly \(page 630\)](#)

[Overlapping numbering series \(page 631\)](#)

[Family numbers \(page 634\)](#)

Plan your numbering series

Before you start modeling, it is a good idea to plan the numbering prefixes and start numbers you will use for the entire project. Careful planning prevents numbering conflicts.

To save time, include the numbering series in the default part properties for each type of part before you start modeling.

You may want to omit the part prefix for minor parts, such as plates. If you do this, ensure that you assign a **Start number** for that numbering series so that it will not overlap other parts.

Example

One way to plan the numbering series is to create a table:

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Beam	PB	1	AB	1
Vertical brace	PVB	1	AVB	1
Horizontal brace	PHB	1	AHB	1
Rafter	PR	1	AR	1

Part type	Part Prefix	Part Start number	Assembly Prefix	Assembly Start number
Purlin	PP	1	AP	1
Column	PC	1	AC	1
Plate		1001	A	1

See also

[Numbering series \(page 628\)](#)

[Overlapping numbering series \(page 631\)](#)

[Numbering prefixes in US environments](#)

Assign a numbering series to a part

1. Double-click a part to open the part properties in the property pane.
2. If you are modifying the properties of a concrete part, go to the **Cast unit** tab.
3. Under **Numbering series**, define a part prefix and a start number.
4. Click **Modify**.

See also

[Assign a numbering series to an assembly \(page 630\)](#)

[Numbering series \(page 628\)](#)

Assign a numbering series to an assembly

To assign a numbering series to an assembly:

To	Do this
To assign a numbering series to an assembly according to its main part	<ol style="list-style-type: none"> 1. Check what is the main part of an assembly. <ol style="list-style-type: none"> a. On the ribbon, click the down arrow next to , and then select Assembly objects. b. Select the assembly. Tekla Structures highlights the main part in orange. c. Press Esc.

To	Do this
	<ol style="list-style-type: none"> 2. Ensure that the  Select objects in assemblies selection switch (page 122) is active. 3. Double-click the assembly main part to open the part properties in the property pane. 4. Under Numbering series, define an assembly prefix and a start number. 5. Click Modify.
To assign a numbering series to an assembly by using the assembly properties	<ol style="list-style-type: none"> 1. Ensure that the  Select assemblies selection switch (page 122) is active. 2. Double-click an assembly to open the assembly properties in the property pane. 3. In the Assembly numbering boxes, define an assembly prefix and a start number. 4. Click Modify.

Related information

See also

[Assign a numbering series to a part \(page 630\)](#)

[Numbering series \(page 628\)](#)

Overlapping numbering series

When you plan numbering, ensure that you reserve enough numbers for each series. If a series overlaps another, Tekla Structures numbers only one of the objects that would have overlapping numbers and leaves the other object unnumbered.

Tekla Structures warns you about series overlaps. View the numbering history log to check which numbers overlap, and then adjust the numbering prefixes and start numbers so that the series will not overlap anymore.

Related information

See also

[Numbering series \(page 628\)](#)

[View the numbering history \(page 644\)](#)

Identical parts

Tekla Structures gives parts the same number if the parts are identical **in the fabrication or casting**. If a part is deformed after fabrication or casting (for example if the part is cambered, shortened, or warped), the final geometry on site and in the model may be different.

Tekla Structures treats parts as identical and gives them the same number if the following basic part properties are the same:

- Part geometry
- Casting direction
- Numbering series
- Profile
- Material
- Finish
- Shortening

You can set the degree of tolerance for part geometry in the **Numbering Setup** dialog box. If the geometry of parts differs within this degree of tolerance, Tekla Structures treats the parts as identical for numbering purposes.

Class does not affect numbering. Tekla Structures gives the same number to identical parts that belong to different classes.

If you have created NC files, pop marks and contour marking affect numbering.

See also

[Casting direction \(page 399\)](#)

[Define what affects numbering \(page 633\)](#)

[User-defined attributes in numbering \(page 634\)](#)

Identical reinforcement

Tekla Structures treats reinforcing bars as identical, and gives them the same number, if the following properties are the same:

- Bar geometry
- Numbering series
- Size
- Grade

- Bending radius

Class does not affect numbering. Tekla Structures gives the same number to identical reinforcing bars that belong to different classes.

The modeling direction of tapered reinforcing bar groups affects numbering. This means that identical bar groups with different modeling directions receive different numbers.

The bar length rounding, step tapering, and some other rebar set settings affect bar geometry and thus the numbering of rebar set bars. The rounding settings that are defined in the `rebar_config.inp` file for drawings and reports do not affect numbering.

See also

[Create reinforcement \(page 427\)](#)

Define what affects numbering

To define which properties affect numbering in your model, modify the settings in the **Numbering Setup** dialog box.

You can have Tekla Structures compare the following properties:

- Holes (if created with the **Bolt** command)
- Part name
- Beam orientation
- Column orientation
- Assembly name
- Assembly phase (XS_ENABLE_PHASE_OPTION_IN_NUMBERING set to TRUE)
- Reinforcement
- Embedded objects (affect only cast units)
- Surface treatment (affects only assemblies)
- Welds (affect only assemblies)

If these properties differ, Tekla Structures treats objects as different, and therefore numbers them differently.

For example, if two otherwise identical parts have different names and you select the **Part name** check box, Tekla Structures gives the parts different numbers.

By default, a part retains its number, as long as only one part has that particular number, regardless of the settings in the **Numbering Setup** dialog box.

See also

[Adjust the numbering settings \(page 636\)](#)

[What is numbering and how to plan it \(page 627\)](#)

[Identical parts \(page 632\)](#)

[Identical reinforcement \(page 632\)](#)

[User-defined attributes in numbering \(page 634\)](#)

[General numbering settings \(page 923\)](#)

User-defined attributes in numbering

You can set in the `objects.inp` file whether a user-defined attribute affects numbering or not. Tekla Structures treats parts and reinforcing bars as different, and therefore numbers them differently, if the values of the user-defined attribute differ.

NOTE Only user-defined attributes of parts and reinforcement can affect numbering. User-defined attributes of other objects, such as phases, projects and drawings do not affect numbering.

If you want Tekla Structures to consider a user-defined attribute when numbering, set the `special_flag` option of the attribute to `yes` in the `Part attributes` section of `objects.inp`. For reinforcement, you need to set `special_flag` to `yes` **also** in the `Reinforcing bar attributes` section. Tekla Structures assigns different numbers to parts or reinforcement that are otherwise identical but have different values for this user-defined attribute.

If you want Tekla Structures to ignore a user-defined attribute when numbering, set the `special_flag` option to `no` in `objects.inp`.

Related information

See also

[Examples of user-defined attributes \(UDAs\) for parts \(page 319\)](#)

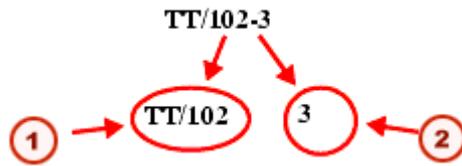
[Select and change the profile or material of a part \(page 316\)](#)

[Define what affects numbering \(page 633\)](#)

Family numbers

With family numbering you can group objects within the same numbering series into different “families”. This can be used, for example, to find similar cast units that can be cast in the same bed.

When you use family numbering, the cast unit position numbers consist of a *family number* and a *qualifier*. For example:



1. Family number
2. Qualifier

Assemblies and cast units that match the comparison criteria you define in the **Numbering Setup** dialog box get the same family number. However, if they have the same family number but different part geometry or materials, they get unique qualifier numbers.

See also

[Numbering series \(page 628\)](#)

[Assign family numbers \(page 635\)](#)

[Change the family number of an object \(page 636\)](#)

[Example: Use family numbers \(page 656\)](#)

Assign family numbers

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Go to the **Family numbering** tab.
3. Define which numbering series to assign family numbers to.
 - a. Click **Add series** to open the **Add series** dialog box.

Tekla Structures displays all the assembly and cast unit numbering series in the model.
 - b. Select a numbering series from the list, and then click **Add**.

The numbering series appears in the family numbering list.
4. Under **Compare**, select the properties that need to be identical for the members of the same family.

Define comparison criteria for each numbering series separately.

Select at least one check box, but not all of them. If you select all the check boxes, the family number will be the same as the normal assembly position, and the qualifier number will be 1 for all. If you do not select any check boxes, only one family number per series is assigned.

5. Click **Apply**.
Tekla Structures stores the settings in the numbering database file (<model_name>.db2) in the current model folder the next time you save the model.
6. If you are assigning family numbers to parts that have already been numbered, clear the existing numbers.
7. Update numbering in the model.
Tekla Structures assigns a family number to all objects in the numbering series.

See also

[Family numbers \(page 634\)](#)

[Clear existing numbers \(page 641\)](#)

Change the family number of an object

You can change the family number and/or family qualifier of an object.

1. Select the objects whose family numbers you want to change.
2. On the **Drawings & reports** tab, click **Change number --> Change family number**.
3. In the **Assign Family Number** dialog box, enter the desired values in the **Family number** and **Family qualifier** boxes.
4. Click **Assign**.

See also

[Family numbers \(page 634\)](#)

5.2 Adjust the numbering settings

You can adjust the numbering settings to better suit your needs. This should be done early in the project, before creating any drawings or reports. Do not change your numbering conventions in the middle of a project.

1. On the **Drawings & reports** tab, click **Numbering settings --> Numbering settings** to open the **Numbering Setup** dialog box.
2. Modify the settings if necessary.
For example, you can define which part properties affect numbering in your model. Using the default settings is effective in most cases.
3. Click **Apply** or **OK**.

NOTE Always check and repair numbering after you have changed the numbering settings.

See also

[Define what affects numbering \(page 633\)](#)

[Numbering settings during a project \(page 661\)](#)

[Repair numbering errors \(page 645\)](#)

5.3 Number parts

Use the **Number modified objects** command to number all parts that have been created or modified since the last numbering. If this is the first time you run numbering for this model, all parts in the model are new and will therefore be numbered.

If the model has changed since the last numbering so that a modified part becomes identical to an unmodified part, Tekla Structures updates the part numbers in the following way when you run numbering:

- If both or none of the parts have drawings, the unmodified part will keep its number.
- If one of the parts has a drawing and the other one does not, the number of the part that has a drawing will stay the same, and the number of the part that does not have a drawing will change.

To number new and modified parts:

- On the **Drawings & reports** tab, click **Perform numbering --> Number modified objects** .

Tekla Structures numbers the parts.

See also

[Number a series of parts \(page 637\)](#)

[Number assemblies and cast units \(page 638\)](#)

[Number reinforcement \(page 639\)](#)

[Number welds \(page 640\)](#)

[Save preliminary numbers \(page 640\)](#)

[Number parts by design group \(page 653\)](#)

Number a series of parts

Use the **Number series of selected objects** command to only number parts that have a certain prefix and start number. This enables you to limit the numbering to certain series of objects only, which can be useful in large models.

Before you start, we recommend that you plan the numbering series carefully, and split the model into smaller numbering series, for example by area or phase.

1. Select the parts that have the desired prefix and start number.
Only parts that have the same prefix and start number as the selected part will be numbered.
2. On the **Drawings & reports** tab, click **Perform numbering --> Number series of selected objects**.
Tekla Structures numbers all parts in the specified numbering series.

See also

[Example: Number selected part types \(page 658\)](#)

[Example: Number parts in selected phases \(page 659\)](#)

Number assemblies and cast units

To number assemblies and cast units, use the same numbering commands as for numbering parts. Before numbering, you can modify the sort order, which defines how assemblies and cast units are given their position numbers. Sorting does not affect part position.

1. If needed, modify the sort order of assemblies and cast units.
 - a. On the **Drawings & reports** tab, click **Numbering settings --> Numbering settings** to open the **Numbering Setup** dialog box.
 - b. Modify the sort order by selecting options from the **Sort by** and **Then by** lists.



Assembly position sort order

Sort by

X Ascending Descending

Then by

Y Ascending Descending

Then by

Z Ascending Descending

The default sort order is XYZ. You have the following options:

- The x, y or z coordinates of the main part of the assembly or cast unit

The sorting is based on the center of gravity (COG) location of the assembly or cast unit. Tekla Structures finds the center of gravity for each assembly and cast unit and compares them in the order you defined.

- The user-defined attribute of an assembly or the main part

If your sorting is based on user-defined attributes, Tekla Structures displays a list that includes all the available user-defined attributes.

- c. Click **Apply** or **OK** to save the changes.
2. If needed, modify the other numbering settings.
3. On the **Drawings & reports** tab, click **Perform numbering --> Number modified objects** to number the model.

NOTE If you add new parts in the model, parts that have already been numbered are **not** renumbered to suit the sorting order. In this case you should check and repair the numbering of those parts.

See also

[Repair numbering errors \(page 645\)](#)

Number reinforcement

To number reinforcement, use the same numbering commands as for numbering parts.

Note that reinforcement may affect the numbering of parts and cast units. To force Tekla Structures to give otherwise identical concrete parts and cast units different numbers if they have different reinforcement, select the **Reinforcing bars** check box in the **Numbering Setup** dialog box.

Part numbering and cast unit numbering do not affect the numbering of reinforcement.

See also

[Number parts \(page 637\)](#)

[Identical reinforcement \(page 632\)](#)

[Define what affects numbering \(page 633\)](#)

[User-defined attributes in numbering \(page 634\)](#)

[Assign running numbers to reinforcement \(page 517\)](#)

Number welds

Use the **Number welds** command to assign numbers to welds. Weld numbers are displayed in drawings and reports.

1. On the **Drawings & reports** tab, click **Perform numbering --> Number welds** to open the **Weld Numbering** dialog box.
2. If needed, modify the [weld numbering settings \(page 925\)](#).
For example, you can define whether to assign numbers for **All welds** or **Selected welds**.
3. If you selected to assign numbers to certain welds only, select the welds.
4. Click **Assign numbers** to start numbering welds.

See also

[Number parts \(page 637\)](#)

Save preliminary numbers

Preliminary mark is a user-defined attribute that defines the part position number. You can save the current part position numbers as preliminary numbers for selected parts. The previous preliminary numbers are overridden.

1. Select the parts.
2. On the **Drawings & reports** tab, click **Numbering settings --> Save preliminary numbers**.

See also

[Number parts \(page 637\)](#)

5.4 Change existing numbers

Use the **Change number** commands to change the existing part, assembly, multi-position, or family numbers into something you have defined yourself. These commands do not change the numbering series of parts. To prevent drawing, modeling, and fabrication errors, Tekla Structures does not allow you to use identical numbers for two different assemblies or parts.

1. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:
 - **Change part number**
 - **Change assembly number**
 - **Change part multinumber**

- **Change assembly multinumber**
- **Change family number**

The corresponding dialog box appears.

2. Select a part in the model.
3. Click **Get** to view the current numbering properties of the part.
4. Enter the numbering properties you want to use for this part.

Note that the position numbers you enter here are not absolute numbers. For example, if the start number of the series is 100, the position numbers refer to numbers in that series. Therefore, position number 1 is actually 100, position number 2 is 101, position number 3 is 102, and so on.

5. If you are changing the assembly number of selected parts, ensure that the **Assign to** option is set to **Selected objects only**.

Otherwise, all parts with the same original number will be renumbered.

6. Click **Assign** to change the number.

If the number you specified is already in use, Tekla Structures displays a warning and keeps the original number.

Tekla Structures also displays a warning if the position number is higher than the highest current number. This is for information only and the number is still changed.

See also

[Number parts \(page 637\)](#)

5.5 Clear existing numbers

Use the **Clear** commands to permanently remove the current position numbers of parts. Next time you run numbering, Tekla Structures assigns new numbers to these parts, irrespective of what their previous numbers were.

1. Select the parts whose numbers you want to clear.
2. On the **Drawings & reports** tab, click **Change number** and select one of the following commands:
 - **Clear part and assembly numbers**
 - **Clear part numbers**
 - **Clear assembly numbers**
 - **Clear reinforcing bar numbers**

Tekla Structures removes the position numbers of the selected parts.

See also

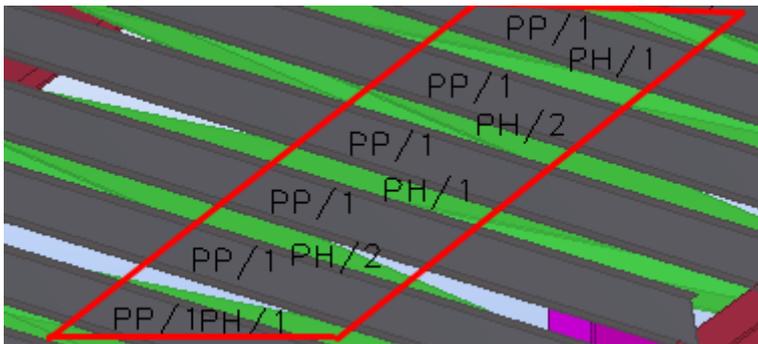
[Number parts \(page 637\)](#)

5.6 Check the numbering

You can check the position numbers in many places:

- You can add the position numbers to the part labels.
 1. In the model, make sure the **Select views** selection switch is active.
 2. Double-click the background to open the **View Properties** dialog box.
 3. Click **Display...** to open the **Display** dialog box.
 4. Go to the **Advanced** tab and add **Part position** to the **Part label**.
 5. Click **Modify**.

The part labels now contain the position numbers.



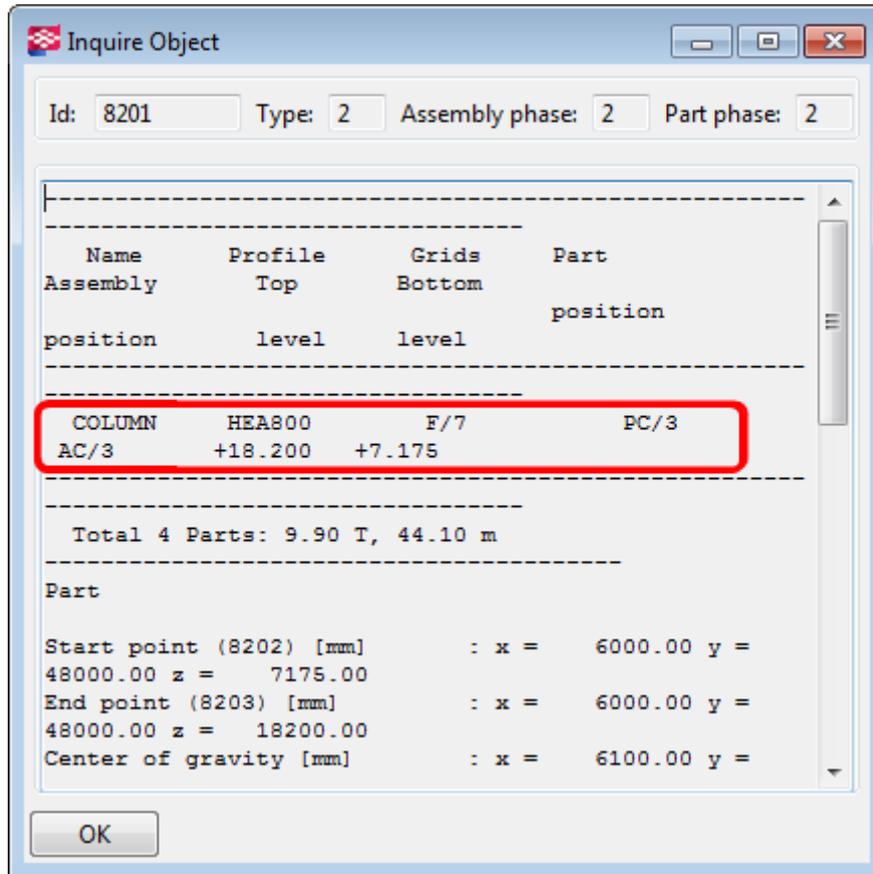
- You can check the part number in Document manager.

[AP.1]	STANDARD
[AR.1]	STANDARD
[AV.1]	STANDARD

- The drawing label can show the position number and the amount of identical parts.

GENERAL NOTES:		ALL HOLES ARE	0.0	mm UNLESS NOTED		
		ALL WELDS ARE	0.0	mm F.W UNLESS NOTED		
MATERIAL LIST FOR ASSEMBLY MK'D			AC/5	3	No. Required	
Mark	Profile	Material	No.	Length	Area	Weight
PC/5	HEA800	S355JR	1	18200	49.1	4086.1
Total					49.1	4086.1

- You can use the **Inquire** commands.



- You can create reports that list your assembly and part positions.

Report

Report

TEKLA STRUCTURES ASSEMBLY PART LIST FOR CONTRACT No.:12345 Page: 1
 TITLE: Paper Industry Building PHASE: Date: 10.02.2012

Assembly	Part	No.	Profile	Grade	Length(mm)	Weight (kg)
4/1		2	D7000			0.0
	Concrete/1	1	D7000	K40-1	800	0.0
A/1		72	HEA300			1183.4
	P/1	1	HEA300	S355JR	13400	1183.4
A/2		2	D6400			4543782.
	P/2	1	D6400	S355JR	18000	4543782.
A/3		3	RHS150*150*5			200.7
	P/3	1	RHS150*150*5	S355JR	8846	200.7
A/4		3	RHS150*150*6			190.9
	P/4	1	RHS150*150*6	S355JR	8415	190.9
A/5		26	IPE600			1610.3
	P/5	1	IPE600	S355JR	13150	1610.3
A/6		2	IPE600			1102.1
	P/6	1	IPE600	S355JR	9000	1102.1
A/7		8	IPE600			692.7
	P/7	1	IPE600	S355JR	5657	692.7
A/8		1	IPE600			508.2
	P/8	1	IPE600	S355JR	4150	508.2
A/9		4	IPE600			734.8
	P/9	1	IPE600	S355JR	6000	734.8
AC/1		1	HEA800			1234.8
	PC/1	1	HEA800	S355JR	5500	1234.8
AC/2		4	HEA800			2924.2
	PC/2	1	HEA800	S355JR	13025	2924.2
AC/3		4	HEA800			2475.2
	PC/3	1	HEA800	S355JR	11025	2475.2

OK

See also

[Repair numbering errors \(page 645\)](#)

5.7 View the numbering history

To view the numbering history:

- On the **File** menu, click **Logs** --> **Numbering history log** .

Tekla Structures displays the numbering log file.

5.8 Repair numbering errors

We recommend that you check and repair numbering in the model every now and then, especially before producing drawings or reports.

NOTE If you work in the multi-user mode, it is very important that you repair numbering regularly.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Ensure that the option **Compare to old** is selected for **New** parts.
3. Ensure that one of the following options is selected for **Modified** parts:
 - **Compare to old**
 - **Keep number if possible**
4. Click **OK** to save the changes.
5. Unless you want to repair the entire model, select the objects whose numbering you want to repair.
6. On the **File** menu, click **Diagnose & repair** and select one of the following commands under **Numbering**:
 - **Diagnose and repair numbering: all**

This command numbers all parts and assemblies, even the unmodified ones.
 - **Diagnose and repair numbering: series of selected objects**

This command numbers all parts and assemblies that have the same prefix and start number as the selected part.

Note that Tekla Structures assigns the position number of the oldest part or assembly to all identical parts, even if a newer part or assembly has a smaller position number.

TIP To manually assign a certain position number on a part or assembly, use the **Change number** command after repairing numbering in the model.

See also

[Change existing numbers \(page 640\)](#)

5.9 Renumber the model

Use the **Renumber all** option when the numbering needs to be started over. This option permanently removes the existing position numbers and resets them with new ones. Any existing drawings will also be removed.

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
2. Select the **Renumber all** check box.
3. Click **Apply** or **OK**.
4. On the **Drawings & reports** tab, click **Perform numbering** --> **Number modified objects** .
5. When you are asked to confirm the renumbering of the model, click **Yes**. Tekla Structures renumbers the entire model.

See also

[Change existing numbers \(page 640\)](#)

[Clear existing numbers \(page 641\)](#)

5.10 Control numbers

Control numbers are additional numbers that can be used to identify parts in a model. Use control numbers if you need to give additional, unique numbers to assemblies or cast units, regardless of their position numbers.

Control numbers can be useful, for example, when delivering a large number of similar wall elements to the site. To successfully pack and unpack the load, you need to plan the order of the wall elements already when the order is shipped. Although all of the wall elements may have the same cast unit position number, you can assign a unique control number to each individual wall element.

See also

[Assign control numbers to parts \(page 646\)](#)

[Control number order \(page 647\)](#)

[Display control numbers in the model \(page 649\)](#)

[Remove control numbers \(page 650\)](#)

[Lock or unlock control numbers \(page 651\)](#)

[Example: Use control numbers to indicate the erection order \(page 651\)](#)

Assign control numbers to parts

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog box.
2. Indicate which parts to assign control numbers to.
 - To number the entire model, do not select any parts.
 - To only number specific parts, select the parts you want to number.
3. If you want to assign control numbers only to parts in a specific numbering series:
 - a. In the **Numbering** list, select **By numbering serie**.
 - b. Enter the **Prefix** and **Start number** in the corresponding boxes.
4. Define the control numbers to be used.
 - a. In the **Start number of control numbers** box, enter the first control number to be used.
 - b. In the **Step value** box, define the control number interval.

For example, to assign the control numbers 2, 5, 8, 11, and so on, enter 2 in the **Start number of control numbers** box and 3 in the **Step value** box.
5. Use the **Renumber** list to specify how to treat parts that already have control numbers.
 - Select **No** to keep the existing control numbers.
 - Select **Yes** to replace the existing control numbers with new ones.
6. Use the **First direction**, **Second direction**, and **Third direction** lists to define the order of control numbers.
7. In the **Write UDA to** list, select where to save the control numbers. The control number will appear on the **Parameters** tab in the user-defined attributes dialog box either for:
 - **Assembly**
 - **Main part**
8. Click **Apply** to save the changes.
9. Click **Create** to number the parts.

See also

[Control number order \(page 647\)](#)

[Control number settings \(page 925\)](#)

Control number order

When you assign control numbers, you must define in what order to assign them. The order is based on the location of each part on the global coordinate system.

The options are:

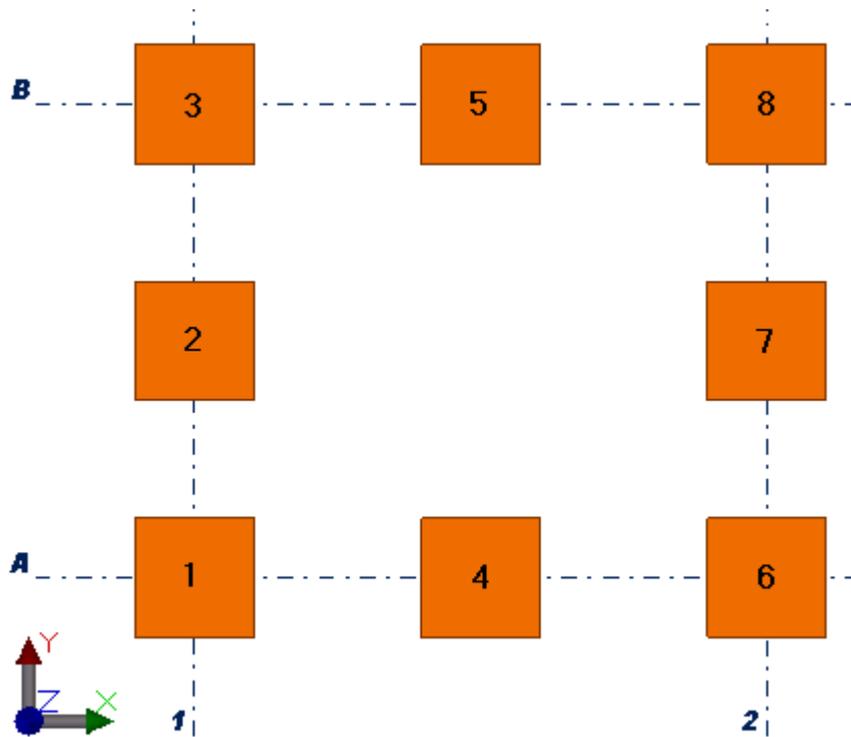
- **None**
- **X**
- **Y**
- **Z**
- **-X**
- **-Y**
- **-Z**

With positive directions (X, Y, and Z), the parts with the lowest coordinate value are numbered first. With negative directions (-X, -Y, and -Z), the parts with the highest coordinate value are numbered first.

For example, if the first direction is X, the second direction is Y, and the third direction is Z, numbering starts from the parts that have the lowest x coordinate value. If multiple parts have the same x coordinate, also their y coordinates are compared. If multiple parts have the same x and y coordinates, also their z coordinates are compared.

Example

In the following example, the first direction is X and the second direction is Y. The numbers 1–8 indicate the control numbers.



See also

[Assign control numbers to parts \(page 646\)](#)

Display control numbers in the model

If the control numbers are not displayed in the model, you can set them visible using the display settings.

1. Double-click the view to open the **View Properties** dialog box.
2. Click **Display...** and go to the **Advanced** tab.
3. Select the **Part label** check box.
4. In the **Properties** list, select **User-defined attributes**, and then click **Add >**.

The **Part label** dialog box is displayed.

5. Enter `ACN` and click **OK**.

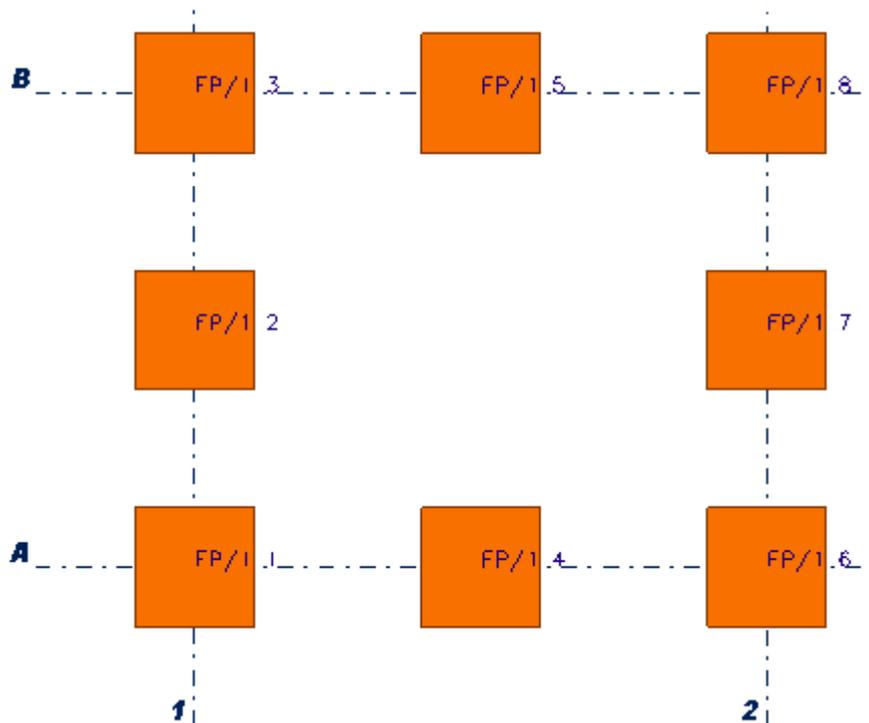
The property is moved to the **Part label** list.

6. Click **Modify**.

The control numbers are displayed in the model, right after the part position numbers.

Example

In the following example, the numbers 1–8 indicate the control numbers.



See also

[Control numbers \(page 646\)](#)

Remove control numbers

If needed, you can remove existing control numbers from all or some of the parts. Do not remove control numbers unless you are absolutely sure that they are no longer needed.

NOTE Removing of control numbers is different than **reassigning** of control numbers. If you only want to reassign new control numbers to parts that already have control numbers, use the **Renumber** option in the dialog box.

1. Double-click the part to open the part properties dialog box.
2. Click **User-defined attributes...**

The current control number of the part is displayed on the **Parameters** tab, next to the **Control Number** box. For example:

Control Number 2

3. Remove the existing control number from the box.
4. Click **Modify** to apply the change.

See also

[Control numbers \(page 646\)](#)

Lock or unlock control numbers

To prevent other users from changing the control numbers of some or all of the parts in the model, use the **Lock/unlock control numbers** command. Later on, if the control numbers need to be changed, you can use the same command for unlocking the numbers.

1. On the **Drawings & reports** tab, click **Numbering settings --> Lock/unlock control numbers** to open the **Lock/Unlock control numbers** dialog box.
2. Define which parts' control numbers to lock or unlock.
 - To lock or unlock the control numbers of all parts, do not select any parts in the model.
 - To lock or unlock the control numbers of specific parts only, select the parts in the model.
3. In the **Status** list, select **Lock** or **Unlock**.
4. Click **Apply** to save the changes.
5. Click **Create** to lock or unlock the numbers.

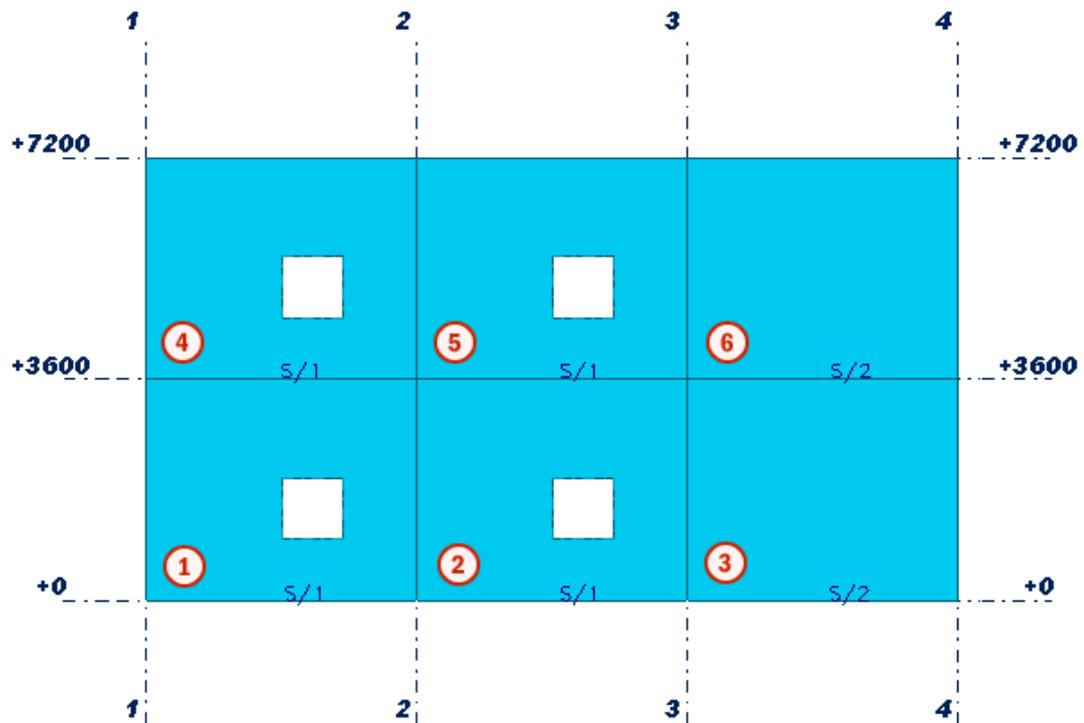
See also

[Control numbers \(page 646\)](#)

Example: Use control numbers to indicate the erection order

This example shows how to assign control numbers to six concrete walls. As four of these walls have the same cast unit position, you cannot make a clear distinction between the cast units based on their position number. That is why each wall will get a unique identifier that indicates their erection order at the site. The erection order also affects the order of shipping. For example, the wall number 1 needs to be shipped on top of the delivery, because it will be erected first on the site; the wall number 2 should be the second on the bundle, because it will be erected next; and so on.

The following image represents the desired end result.



- ① Erected as 1st
- ② Erected as 2nd
- ③ Erected as 3rd
- ④ Erected as 4th
- ⑤ Erected as 5th
- ⑥ Erected as 6th

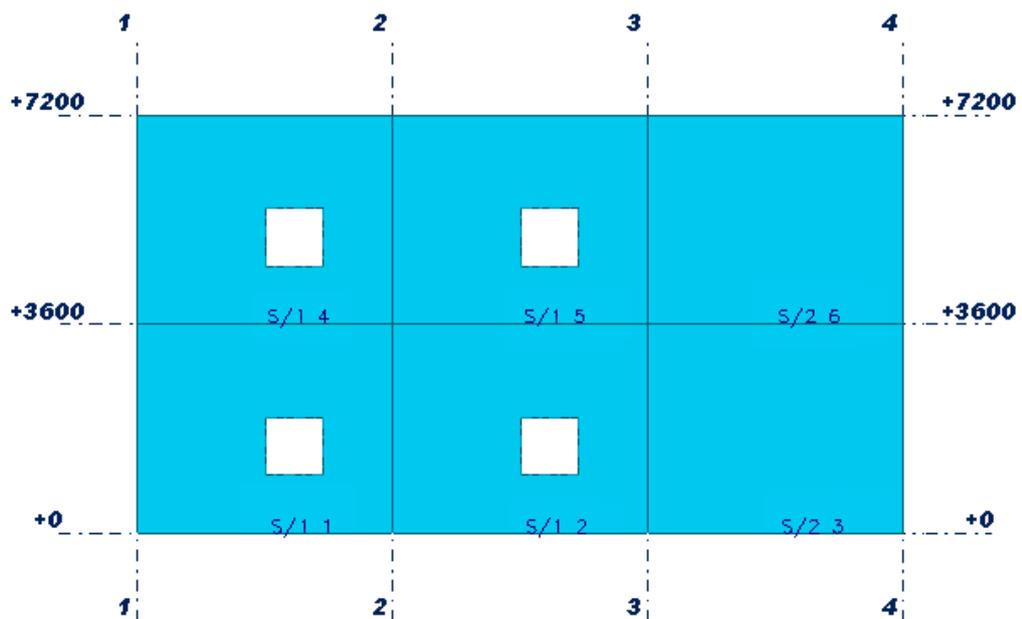
To assign control numbers to the concrete walls:

1. On the **Drawings & reports** tab, click **Numbering settings** --> **Assign control numbers** to open the **Create control numbers** dialog box.
2. Select the six concrete walls.
3. Define that you want to assign control numbers only to parts in the numbering series S with the start number 1.
 - a. In the **Numbering** list, select **By numbering serie**.
 - b. In the **Prefix** box, enter s.
 - c. In the **Start number** box, enter 1.

4. Define that you want to use the numbers 1–6 as the control numbers for these concrete walls.
 - a. In the **Start number of control numbers** box, enter 1.
 - b. In the **Step value** box, enter 1.
5. Define that you want to first number the walls with identical z coordinates in the order they appear in the positive x axis.
 - a. In the **First direction** list, select **Z**.
 - b. In the **Second direction** list, select **X**.
6. Click **Apply** to save the changes.
7. Click **Create** to number the concrete walls.

Each concrete wall gets a unique control number, as shown in the following image.

TIP If you cannot see the control numbers in the model, modify the display settings. For more information on the required settings, see [Display control numbers in the model \(page 649\)](#).



5.11 Number parts by design group

You can number parts by design groups so that you can differentiate the parts from each other in drawings and reports. The design group numbers can be used in engineering documents or as preliminary numbers.

Design group numbers consist of a prefix, a separator, and a number or a letter.

Use the **Design group numbering** application to assign prefixes and numbers or letters to parts on the basis of design groups. The **Design group numbering** application groups parts that match a selection filter into a design group, numbers them, and optionally compares the part lengths. The application also compares the user-defined attributes of parts that are set to affect numbering.

NOTE The **Design group numbering** application only numbers parts that have a profile extruded to create the length of the part, such as beams, columns, panels, and footings. Contour plates, slabs, or items are not numbered.

Before you start:

- Create the needed selection filters that define the design groups.
- If you want to use certain letters in design group numbering, list the allowed letters using the advanced option `XS_VALID_CHARS_FOR_ASSEMBLY_POSITION_NUMBERS`. By default, letters A–Z are allowed.
- In multi-user or Tekla Model Sharing models, make sure that only one of the users runs the **Design group numbering** application.

To number parts by their design group:

1. In the model, Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the arrow next to **Applications** to open the applications list.
3. Double-click **Design group numbering** to start the application.
4. In the **Design group numbering** dialog box:
 - a. Click **Add group** to create design group numbering settings for parts matching a selection filter.
 - Select the filter in the **Group filter** column.
The selection filters are read from specific folders in the standard folder search order.
 - Enter the design group prefix and start number or start letter you want to use for the parts in this group.
 - In the **Compare length** column, define whether the part lengths are compared or not.
 - b. Repeat step 4a for all part groups that you want to number by design group.

- c. If needed, change the order of the groups by using the **Move up** and **Move down** buttons.
If a part belongs to several groups, the last group filter in the list overrides the previous ones.
 - d. If you want to compare part lengths, define the length tolerance.
For example, if you enter 0, parts must be exactly the same length to receive the same design group number (or letter). If you enter 2, part lengths can differ 2 mm from each other.
The default tolerance is 0.05 mm.
 - e. Enter a number separator that is used to separate the design group prefix and number or letter in drawing marks and in reports. For example, enter - .
We recommend that you do not change the separator during the project.
 - f. To reuse old, unneeded numbers or letters, select the **Reuse old numbers** check box.
 - g. In **Number using letters**, select whether to use letters or not.
 - h. In **Renumber all**, select whether to renumber all parts or not.
 - i. To number the parts by design group, click **Perform numbering**.
A design group number is saved as the user-defined attribute `DESIGN_GROUP_MARK` of each part.
By default, the user-defined attribute `DESIGN_GROUP_MARK` is available in the `objects.inp` file in the Engineering configuration in the default and US environments.
 - j. To create a report to show the numbering results, select whether to create a report of all or the selected parts, and then click **Create report**.
Tekla Structures shows the report in the **List** dialog box and also saves the report. The report is saved as `dgnReport.txt` in the folder defined by the advanced option `XS_REPORT_OUTPUT_DIRECTORY`. In the Default environment, the report is saved in the `\Reports` folder under the current model folder.
When you select a row in the **List** dialog box, Tekla Structures highlights and selects the corresponding part in the model.
If the numbering of a part is not up to date, meaning that the part has been modified after numbering, a question mark (?) is added after the design group number.
5. To show the design group numbers in drawing marks or in reports, use the user-defined attribute `DESIGN_GROUP_MARK`.

See also

[Create new filters \(page 158\)](#)

5.12 Numbering examples

This section gives some examples of numbering the model.

Click the links below to find out more:

[Example: Number identical beams \(page 656\)](#)

[Example: Use family numbers \(page 656\)](#)

[Example: Number selected part types \(page 658\)](#)

[Example: Number parts in selected phases \(page 659\)](#)

Example: Number identical beams

This example explains how different numbering settings result in different part numbers when you modify a part.

To number identical beams:

1. Create three identical beams with the numbering series prefix P and start number 1.
2. Number the model. All the beams have the part position number P1.
3. Modify one of the beams.
4. Number the model. You should now have two beams P1 and one P2.
5. Change beam P2 to be identical to the others.
6. Number the model.

Depending on the numbering settings in the **Numbering Setup** dialog box, Tekla Structures assigns one of the following part position numbers to the modified part:

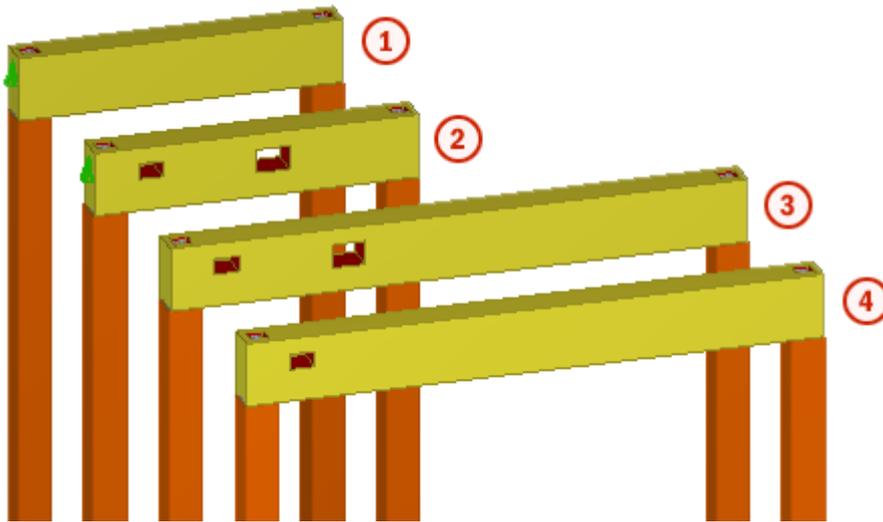
- **Compare to old:** P1
- **Keep number if possible:** P2
- **Take new number:** P3

See also

[Number parts \(page 637\)](#)

Example: Use family numbers

In this example, the following four beams have the numbering series prefix B and the start number 1. The parts have the same main profile, and each pair has the same length, but the holes are different.



- ① Assembly position: B/1
- ② Assembly position: B/2
- ③ Assembly position: B/3
- ④ Assembly position: B/4

We use the following family numbering settings:

- **Use family numbering for series:** add series **B/1**
- **Compare:** select options **Main part profile** and **Overall length**

With the given family numbering criteria, Tekla Structures divides the beams into two families. All beams have the same profile, but each pair has a different length. Within both families the beams get different qualifiers because they have different holes.

- The first beam gets the assembly position number B/1-1
- The second beam gets the assembly position number B/1-2
- The third beam gets the assembly position number B/2-1
- The fourth beam gets the assembly position number B/2-2

See also

[Family numbers \(page 634\)](#)

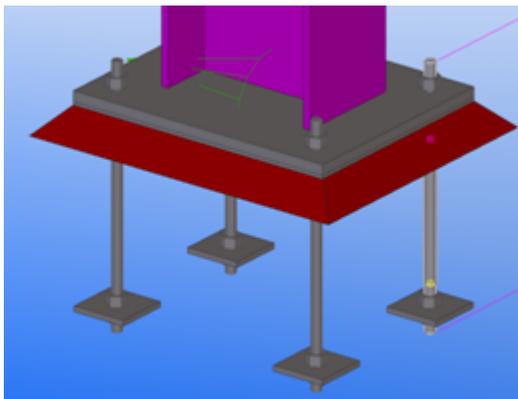
Example: Number selected part types

This example shows how different numbering settings can be used for different part types. We will use one set of numbering settings for steel anchor rods, and another one for steel columns. Note that the **Number series of selected objects** command numbers all parts that have the same assembly prefix.

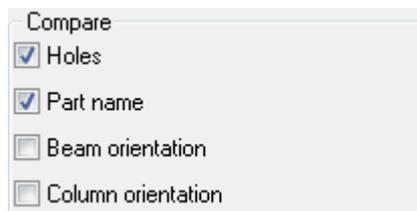
To number anchor rods and columns:

1. Create steel columns.
2. Create anchor rods with the numbering series prefix AR and start number 1.

Ensure that this numbering series is different to any other parts or assemblies in the model.



3. **Apply**
4. Ensure that the **Column orientation** check box is not selected, and then click **Apply**.



5. Select one of the anchor rods in the model.
6. On the **Drawings & reports** tab, click **Perform numbering --> Number series of selected objects**.

All parts with the AR prefix and start number at 1 are numbered.

7. Wait until the anchor rod numbering is complete.
8. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
9. Select the **Column orientation** check box, and then click **Apply**.
10. Select one of the steel columns in the model.
11. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.
All columns belonging to the same numbering series as the selected column are numbered.

See also

[Number a series of parts \(page 637\)](#)

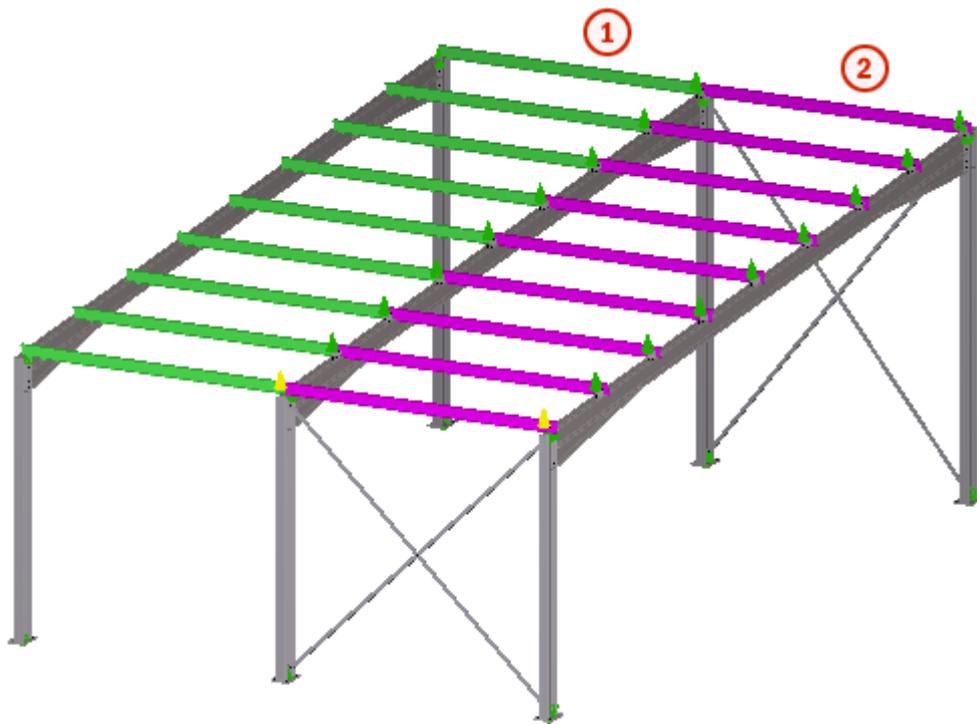
Example: Number parts in selected phases

This example shows how to number a model that consists of multiple phases, each phase having a different detailing and submittal schedule. This enables you to release drawings for a particular phase at any time.

Before you start, divide the model into phases.

To number parts in selected phases:

1. Apply a specific numbering series prefix and start number for parts in each phase.
For example:
 - The beams in phase 1 get the numbering series prefix B and start number 1000.
 - The beams in phase 2 get the numbering series prefix B and start number 2000.



(1) Phase 1: green

(2) Phase 2: magenta

2. Ensure that the numbering series are not overlapping.

For example, to avoid numbering overlaps with the beams in phase 2, phase 1 should not contain more than 1000 position numbers.

3. Select the parts you want to number.

TIP Use selection filters to easily select parts belonging to a certain phase or parts with a specific start number series. You can also use selection filters to ignore specific phases that are already completed or phases that are not ready for numbering.

4. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to open the **Numbering Setup** dialog box.
5. Modify the numbering settings, and then click **Apply**.
6. Select one of the parts you want to number.
7. On the **Drawings & reports** tab, click **Perform numbering** --> **Number series of selected objects**.

All parts belonging to the same numbering series as the selected part are numbered.

See also

[Number a series of parts \(page 637\)](#)

[Numbering settings during a project \(page 661\)](#)

5.13 Tips for numbering

- It is a good idea to have some kind of a routine in numbering. For example, number the model when you start your working day or when you finish for the day.
- To save time, include the numbering series in the default part properties for each type of part before you start modeling.
- Numbering is not just another way of classifying parts. To classify, use **Organizer**, user-defined attributes, or colors.
- If you have overlapping position numbers, Tekla Structures will warn you about it.

You can have a closer look at the overlapping position numbers in the numbering history log. To display the log, click **File menu** --> **Logs** --> **Numbering history log**.

See also

[Numbering settings during a project \(page 661\)](#)

[Numbering examples \(page 656\)](#)

[Create a standard-part model \(page 662\)](#)

Numbering settings during a project

You can use different numbering settings at different times in a project.

For example:

- Before releasing a phase of the project for fabrication, you might use the **Re-use old numbers** option for numbering the entire model.
- If a phase has already been released for fabrication in a project, you might use the **Take new number** option for new and modified parts.
- If you are numbering other phases of the project at earlier stages of detailing, you might use the **Compare to old** option and try to combine as many position numbers as possible.

See also

[Example: Number parts in selected phases \(page 659\)](#)

Create a standard-part model

A standard-part model contains only standard parts with specific part prefixes. You can use these prefixes when numbering parts in another model. The prefixes that you define will be used as actual part position numbers in the other model.

The standard-part model is only used for comparing parts when you number parts in a project model. It cannot be used for creating parts in the project model.

NOTE This functionality only applies to steel parts. Assemblies are not affected.

1. Create a new model and give it a descriptive name.
For example, `StandardParts`.
2. Create the objects you want to use as standard parts.
3. Explode all components.
You can explode the components if you plan to delete the unnecessary parts such as duplicate angles and main parts.
4. Delete all unnecessary items.
5. Give the objects part prefixes that are not used elsewhere (for example, `STD1`, `STD2`, and so on).
Ensure that the standard-part model does not contain duplicate part prefixes. You do not need to define the assembly prefix or the part or assembly start numbers.
6. Save the standard-part model.
To use the standard-part model with Tekla Model Sharing, save the standard-part model in a separate folder under the current model folder.
To use the standard-part model with a multi-user model, save the standard-part model so that all users have access to it.
7. Open a project model that you wish to number.
8. On the **File** menu, click **Settings** --> **Advanced options** --> **Numbering**.
9. Check that the advanced option `XS_STD_PART_MODEL` points to the correct standard-part model.
For example:
`XS_STD_PART_MODEL=C:\TeklaStructuresModels\StandardParts\`
10. On the **Drawings & reports** tab, click **Numbering settings** --> **Numbering settings** to display the **Numbering Setup** dialog box.

11. If you have selected the **Part name** check box, ensure that the project model has the same part names as the standard-part model.
12. Select the **Check for standard parts** check box.
13. Click **Apply** to save the changes.
14. On the **Drawings & reports** tab, click **Perform numbering --> Number modified objects** to number the project model.

As the parts are being numbered, Tekla Structures compares all parts in the project model to the standard-part model. Any part prefixes found in the standard-part model are applied to all identical parts found in the project model. The numbering series of the oldest numbered standard part in the project model is applied to all identical parts found in the project model.

See also

[Number parts \(page 637\)](#)

6 Applications

All available applications, macros, and drawing plugins are located in the section **Applications** of the **Applications & components** catalog. You can also record macros of your own and show them on the list.

Macros

[Macros \(page 666\)](#) are saved as .cs files in the \drawings or \modeling folder under the folders that have been defined with the advanced option . By default, this advanced option is set to ..\ProgramData\Trimble\Tekla Structures\<version>\environments\common\macros.

In addition to this global folder, you can define a local folder and store local macros there, for example, environment macros or firm macros. Specify the local macro folder for the advanced option XS_MACRO_DIRECTORY in addition to the global folder. Define the global folder first and then the local folder. When you create a macro you need to set it as global or local, and the macro will be placed in the global or local folder according to your selection. The macros in the global folder are read first.

Example of the definition for XS_MACRO_DIRECTORY containing a global folder and a local folder:

```
%XSDATADIR%environments\common\macros;%XSDATADIR%environments\uk\General\nuser-macros
```

Macros in the modeling mode

Macro	Description
AutoConnectSelectedParts (page 693)	Use to automatically create connections without opening the AutoConnection dialog box.
AutomaticSplicingTool (page 515)	Use to split long reinforcing bars and bar groups that exceed the stock length, and create splices in split locations.
ContinuousBeamReinforcement	Use to reinforce a continuous beam. The macro creates main top and

Macro	Description
	bottom bars, stirrups, fittings, and additional top and bottom bars using system components.
Convert_DSTV2DXF	Use to create NC files in DXF format by converting DSTV files to DXF files.
CreateSurfaceView (page 31)	Use to create an automatically aligned surface view.
CreateSurfaceView_wEdge (page 31)	Use to create a surface view and align the work plane along the edge you select.
DesignGroupNumbering (page 653)	Use to number parts by design groups so that you can differentiate the parts from each other in drawings and reports.
DirectoryBrowser	Use to to find and modify the location of the various Tekla Structures files and folders, and customize user settings.
RebarClassifier (page 517)	Use to classify the reinforcing bars and reinforcement meshes by their order of depth in concrete slabs and panels.
RebarSeqNumbering (page 517)	Use to assign cast unit specific running numbers (1, 2, 3...) to the reinforcement in the model.
RebarSplitAndCoupler	Use to split a reinforcing bar group and add couplers in relation to the direction of the picked points.
UpdateRebarAttributes	Use to manage the user-defined attributes (UDAs) of the couplers and the end anchor parts created by Rebar Coupler and Anchor Tools .

Macros in the drawing mode

Macro	Description
Add surfacing symbols	Use to add surface treatment symbols in cast unit drawings.
Copy with offsets (Drawing tools)	Use to copy lines, circles, polylines, polygons and rectangles with offsets.
Create fillets (Drawing tools)	Use to connect two intersecting lines by extending the two selected lines to their intersection point.

Macro	Description
Create chamfers (Drawing tools)	Use to create chamfers between two lines using the distance that you specify.
Moment connection symbols (Drawing tools)	Use to create moment connection symbols to show the beams that are connected to columns with rigid connections.
Exaggerate selected dimensions	Use to exaggerate narrow dimensions to make them easier to read.
Rebar layering marker	Use to mark reinforcing bar layers with different marking styles and line types in a drawing.
Rebar mesh view creator	Use to create drawing views each containing one reinforcement mesh.
Remove change clouds	Use to remove dimension change symbols, mark change symbols and associative note change symbols in one go from an open drawing.

Extensions (.tsep)

You can download Tekla Structures extensions that have the `.tsep` file extension from Tekla Warehouse and [import \(page 670\)](#) these extensions to the **Applications & components** catalog. When you restart Tekla Structures, the imported extensions are installed and added to the **Ungrouped items** group in the catalog. You can move them to a suitable group.

Publish groups in the Applications & components catalog

You can collect content to a group that you create in the **Applications & components** catalog. You can then [publish the group \(page 671\)](#) as a catalog definition file to make it available for other Tekla Structures users.

6.1 Working with applications

You can run, add, edit, rename, save as and delete applications, macros and plugins in the **Applications** section of **Applications and Components** catalog. You can also record and edit macros.

To	Do this
Record a macro	1. Click the Applications & components button  in the side pane to open the

To	Do this
	<p>Applications & components catalog.</p> <ol style="list-style-type: none"> Click the Access advanced features button  and then click Record macro > Global or Local depending on whether you want to save the macro in the global or local macros folder. <p>The Local command is only available if you have defined a location for the local macros using the advanced option <code>XS_MACRO_DIRECTORY</code>.</p> <ol style="list-style-type: none"> Enter a name for the macro in the Macro name box. Click OK and perform the actions you want to record. Click Stop recoding to stop recording. <p>The recorded macro is saved under global or local macros in the <code>macros\drawings</code> or <code>macros\modeling</code> folder depending on the mode (drawing or modeling) you were using while you were recording the macro.</p>
Create a macro file and add contents later	<ol style="list-style-type: none"> Click the Applications & components button  in the side pane to open the Applications & components catalog. Click the Access advanced features button  and then click New macro > Global or Local depending on whether you want to save the macro in the global or local macros folder. <p>The Local command is only available if you have defined a location for the local macros</p>

To	Do this
	<p>using the advanced option <code>XS_MACRO_DIRECTORY</code>.</p> <ol style="list-style-type: none"> 3. Enter a name for the macro in the Macro name box. 4. Click OK. <p>This creates an empty macro file that is displayed in the Applications list.</p> <ol style="list-style-type: none"> 5. Right-click the empty macro file and select Edit. 6. Add the macro content, for example, by copying commands from other macro files and save the file.
View or edit a macro	<ol style="list-style-type: none"> 1. Click the Applications & components button  in the side pane to open the Applications & components catalog. 2. Click the arrow next to Applications to open the applications list. 3. Right-click the macro you want to edit and click Edit. <p>The macro can be opened in any text editor.</p> <ol style="list-style-type: none"> 4. If needed, edit the macro and save the macro file.
Run an application	<ol style="list-style-type: none"> 1. Click the Applications & components button  in the side pane to open the Applications & components catalog. 2. Click the arrow next to Applications to open the applications list. 3. Double-click the application you want to run.

To	Do this
Save an application with another name	<ol style="list-style-type: none"> 1. Click the Applications & components button  in the side pane to open the Applications & components catalog. 2. Click the arrow next to Applications to open the applications list. 3. Right-click the application that you want to save with another name and click Save as. 4. Enter a new name for the application and click OK. The application will be added in the list.
Rename an application	<ol style="list-style-type: none"> 1. Click the Applications & components button  in the side pane to open the Applications & components catalog. 2. Click the arrow next to Applications to open the applications list. 3. Right-click the application you want to rename and click Rename. 4. Enter a new name for the application and click OK. The name of the application changes.
Delete an application	<ol style="list-style-type: none"> 1. Click the Applications & components button  in the side pane to open the Applications & components catalog. 2. Click the arrow next to Applications to open the applications list.

To	Do this
	3. Right-click the application you want to delete and click Delete . The application is removed from the list.

See also

[Applications \(page 664\)](#)

6.2 Import a .tsep extension to the Applications & components catalog

You can import Tekla Structures .tsep extensions (Tekla Structures extension package) to the **Applications & components** catalog. First download the extension from Tekla Warehouse, and then import it to the catalog.

NOTE Some Tekla Structures extensions have an .msi installation file. You have to install these extensions separately. Download the .msi installation file from Tekla Warehouse and double-click the file to run the installation.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click  > **Manage extensions** > **Extension manager**.
Alternatively, you can open **Extension manager** from **File menu** --> **Extend** --> **Extension manager**.
3. Click the Tekla Warehouse link and log in to Tekla Warehouse with your Trimble Identity.
4. Search for the .tsep extension and click [Download](#).
5. Click the downloaded extension in your browser.
Tekla Structures opens a dialog box that lists the installed Tekla Structures versions that are compatible with the extension.
6. Select the Tekla Structures versions to which you want to import the extension.
7. Click **Import**.

The extension is shown in **Extension manager** in all the Tekla Structures versions that you selected.

Alternatively, if you want to import the extension to the current Tekla Structures version only, you can do the import in **Extension manager** after downloading the extension. In **Extension manager**, click **Import** and then double-click the `.tsep` file.

You can still remove the extension from **Extension manager** before installing it. Select the extension and click **Cancel**.

8. If needed, repeat steps 4 - 7 to import more Tekla Structures extensions.
9. Restart Tekla Structures to install the imported extension.
10. Open the **Applications & components** catalog.

The extension is shown in the **Ungrouped items** group in the catalog. You can move the extension to a more suitable group, or create a new group.

You can uninstall installed extensions in **Extension manager**. Select one or several extensions (use **Ctrl** or **Shift**) and click **Remove**. The selected extensions are removed when you restart Tekla Structures. Installing and uninstalling extensions creates a log file to `\Tekla Structures\.`

System administrators can copy multiple `.tsep` extension files to a Tekla Structures user's computer to the `\Tekla Structures\ folder. The extensions are installed the next time the user restarts Tekla Structures.`

Copy .tsep extensions to a new Tekla Structures version

When starting to use a new Tekla Structures version, you can use the Migration Wizard tool to copy the installed `.tsep` extensions to the new version. You can open Migration Wizard either from the **Applications & components** catalog,

click  > **Manage extensions** > **Migrate extensions**, or from **File menu** --> **Extend** --> **Migrate extensions**. Once copied, the extensions are listed in **Extension manager** in the new Tekla Structures version. Restart Tekla Structures to install the copied extensions.

See also

[How to use the Applications & components catalog \(page 681\)](#)

6.3 Publish a group in the Applications & components catalog

You can collect content, such as macros, extensions, and system and custom components to a group that you create in the **Applications & components** catalog. You can then publish the group as a catalog definition file to make it

available for other Tekla Structures users. For the published content to work correctly in another Tekla Structures installation, the content must also exist in that installation.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Create a new group:
 - a. Right-click in the catalog and select **New group**.
 - b. Enter a name for the group.
 - c. Select the group and click the small arrow on the right to add a description to the group.
 - d. Add content to the group.

Some content in the **Applications & components** catalog may be hidden. To publish the hidden content, select the **Show hidden items** check box at the bottom of the catalog.

Note that the model-specific items you add to the group are visible in the modeling mode and the drawing-specific items in the drawing mode.
 - e. Add the needed information to the items in the group: description, tags, and additional thumbnail images.

Use a thumbnail image from the `\Tekla Structures\<>version>\Bitmaps` folder to ensure that the image is available for other Tekla Structures users.
3. Right-click the group and select **Publish group** to create a catalog definition file.

The file contains the following information:

 - Name and description of the published group
 - Names and descriptions of the subgroups
 - References to the items that you have added to the group

The file does not contain the actual items. When other users use the group, they must ensure that the referenced items exist in their Tekla Structures installation and model.

 - Descriptions, tags and thumbnail references of the items in the group

The file does not contain the actual thumbnail image files.
4. Add a unique prefix to the filename in the **Publish group** dialog box.

The filename format must be `<prefix>_ComponentCatalog.ac.xml`.
5. Click **Save**.

The file is by default saved to the model folder.

6. Make the group available for other Tekla Structures users by placing the `<prefix>_ComponentCatalog.ac.xml` catalog definition file to an appropriate folder:

- Project, firm or system folder defined in `XS_PROJECT`, `XS_FIRM`, or `XS_SYSTEM`.
- `\attributes` folder under the current model folder
- Extensions folder in `\Tekla Structures\<version>\environments\common\extensions`, or to any of the folders defined in `XS_EXTENSION_DIRECTORY`.

The **Applications & components** catalog also searches the subfolders of these folders. We recommend that you use the extension folders if you have created your own extensions and have included them in the group.

7. Check that the catalog definition file works correctly:
- a. Delete the published group from your **Applications & components** catalog.
 - b. Click  > **Catalog management** > **Reload catalog** to load and view the published group.

When you have checked the group, other users can start using it:

- If the group content is already included in other users' Tekla Structures installation, they can use the group immediately after reloading the catalog by clicking  > **Catalog management** > **Reload catalog**.
- If the group content, for example extensions, is not included in other users' Tekla Structures installation, they have to download the missing extensions from Tekla Warehouse first, and then re-open the model where they are going to use the group.

7 Components

Components are tools that you can use to connect parts in the model. Components automate tasks and group objects so that Tekla Structures treats them as a single unit. You can save the properties of a component and use them in other projects.

Components adapt to changes in the model, which means that Tekla Structures automatically modifies a component if you modify the parts it connects. When you copy or move objects, Tekla Structures automatically copies or moves all the associated components with the objects.

All components are stored in the Applications & components catalog. Click the

Applications & components button  in the side pane to open the **Applications & components** catalog.

System components

Tekla Structures contains a wide range of predefined system components by default. There are three types of system components:

- **Connection** components connect two or more parts, and create all the required objects such as cuts, fittings, parts, bolts, and welds.

For example, end plates, clip angles, and bolted gussets are connections.

In the **Applications & components** catalog, the connection symbol is ▲.

- **Detail** components add a detail or a reinforcement to the main part. A detail is only connected to one part.

For example, stiffeners, base plates, and lifting hooks are steel details, and beam reinforcement and pad footing reinforcement are concrete details.

In the **Applications & components** catalog, the detail symbol is ▲.

- **Detailing** components automatically create and assemble the parts to build a structure, but do not connect the structure to existing parts.

For example, stairs, frames, and towers are detailing components.

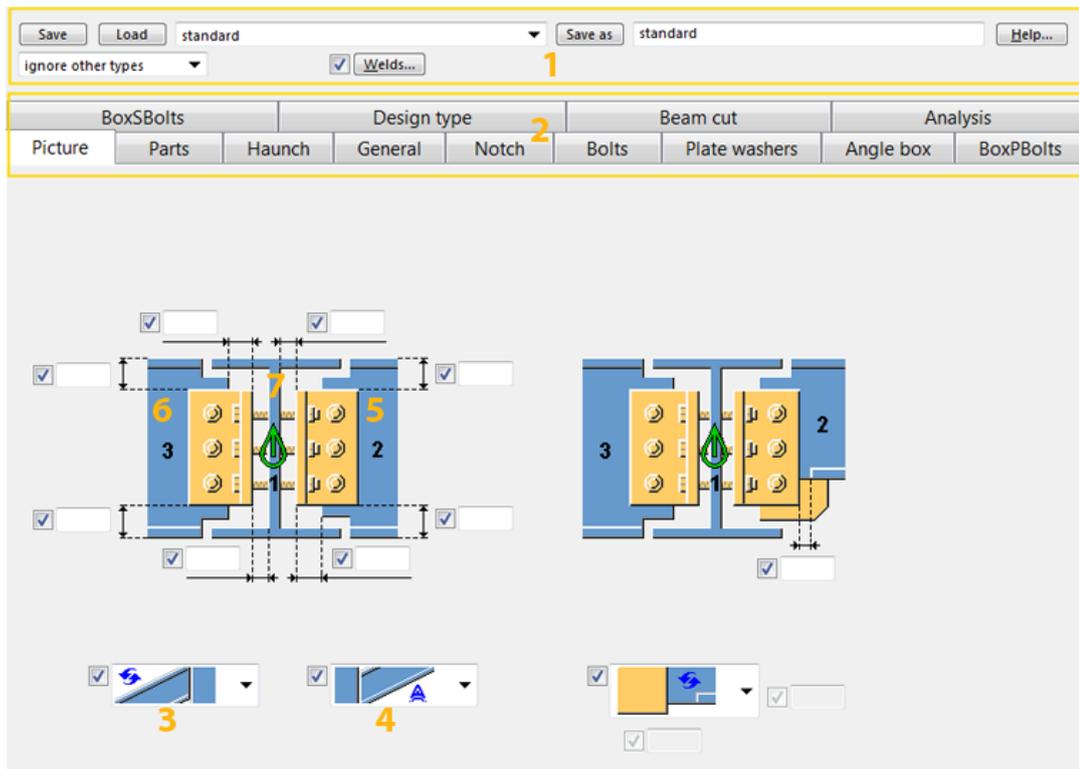
In the **Applications & components** catalog, the detailing symbol is .

You can also create your own components, custom components, if you do not find a system component that suits your needs. You can use custom components in the same way as system components.

7.1 Component properties

Each component has a dialog box where you can define the properties of the component. You can open the dialog box by double-clicking the component in the **Applications & components** catalog.

The image below shows a typical example of a steel connection, the **Two sided clip angle (143)** connection. Concrete and reinforcement component dialog boxes may have different options.



Description	
1	In the upper part of the dialog box, you can save and load predefined settings. Some components have buttons for accessing bolts, welds and DSTV properties. When modifying connections and details, you can select whether Tekla Structures ignores other types of connections and details, or modifies all selected connections and details irrespective of their type. With the

	Description
	<p>modify option, the type of the selected components is changed to match the type of the component you are currently modifying.</p> <p>For more information, see Save and load object properties (page 111).</p>
2	<p>On the tabs, you can define the properties of the parts and bolts that the component creates. You can enter values manually, use system default values, AutoDefaults values, automatic values, or for some steel connections, the values in the <code>joints.def</code> file.</p> <p>Manually entered values, AutoDefaults, automatic values and the properties defined in the <code>joints.def</code> file all override the system defaults. System default values are used if you do not manually enter a value or select any other type of property value. You cannot change the system default values.</p> <p>For more information on <code>joints.def</code>, see Define connection properties in the joints.def file (page 708).</p>
3	<p>If you select an AutoDefaults  option, Tekla Structures uses the property defined in the AutoDefaults rules.</p> <p>The image in the AutoDefaults option is an example and does not necessarily match the outcome in the model.</p> <p>For more information on AutoDefaults, see AutoDefaults (page 694).</p>
4	<p>If you select an automatic  option, Tekla Structures automatically determines which option to use for a property.</p> <p>For example, when you use the automatic option for the stiffener in End plate (144), the connection automatically adds the stiffener to a beam-to-column connection, but does not add it to a beam-to-beam connection.</p> <p>For more information on AutoConnection, see AutoConnection (page 689).</p>
5	<p>The parts that are yellow in the component dialog box are created by the component.</p>
6	<p>The parts that are blue in the component dialog box should already exist in the model before you create the component.</p>
7	<p>Up direction indicates how the connection is rotated around the secondary part, relative to the current work plane. The  symbol on the Picture tab of the component dialog box indicates the correct up direction.</p> <p>If there are no secondary parts, Tekla Structures rotates the connection around the main part. The options are: $+x$, $-x$, $+y$, $-y$, $+z$, $-z$.</p> <p>You can change the default up direction on the General tab of the component dialog box. Try changing the positive directions first.</p>

7.2 Add a component to a model

When you add a component to a model, you either attach the component to existing parts in the model, or pick positions to indicate the location or length of the component.

Connections and details have a main part that you select first. Connections also have one or more secondary parts that you select after you have selected the main part. Detailing components do not always have a main part and secondary parts. Instead, they automatically create and assemble the parts to build a structure when you pick a position in the model.

If you use a component you are unfamiliar with, use the default properties of the component. Then check what needs to be modified, and modify only few properties at a time to see how the modifications affect the component. This is quicker than trying to set all the component properties before seeing what the component actually creates.

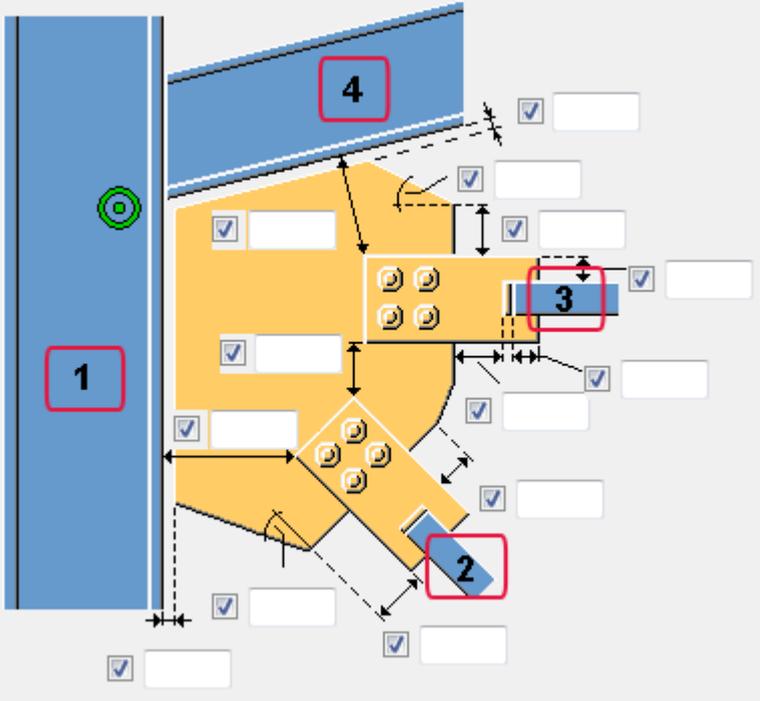
Tekla Structures opens a command prompt when you add a component. Do not close the prompt window, because it displays information on adding the component. This information can be useful in problem situations.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

You can also press **Ctrl + F**.

2. Select a component and do one of the following:

To	Do this
Add a connection ▲	<ol style="list-style-type: none">1. Select the main part.2. Select the secondary part or parts.<ul style="list-style-type: none">• If there is one secondary part, the connection is automatically created when you select the secondary part.• If there are several secondary parts, click the middle mouse button to finish selecting the parts and to create the connection. <p>In the example image below, the numbers from 1 to 4 show the selection order of parts. The blue parts should already exist in the model before you create the component.</p>

To	Do this
	
Add a detail 	1. Select the main part. 2. Pick a position in the main part to determine the location of the detail.
Add a detailing component 	Pick one to three positions to determine the location of the objects that the detailing component creates.

When you have added components to the model, you can use the property pane to list the components:

- If you select one component in the model, the property pane shows the name and number of that component. You can open the component properties dialog box by clicking the **Component properties** button in the property pane.
- If you select several different components in the model, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.
- If you select components and other model objects, click the **Object type list** button  in the property pane to open a list of the selected object types, and select **Component** to list the components.

Component status

When you have added the component, Tekla Structures shows the component status using the symbols shown in the table below. Double-click the symbol to open the component properties.

Color	Status
	<p>The green symbol shows that the component was created successfully.</p> <p>The detailing component symbol in the model is .</p>
	<p>The yellow symbol shows that the component was created, but has problems.</p> <p>This often occurs when bolts or holes have an edge distance that is less than the default value.</p>
	<p>The red symbol shows that the component was not created.</p> <p>Common reasons are that the properties are incorrect or that the up direction is not appropriate.</p>

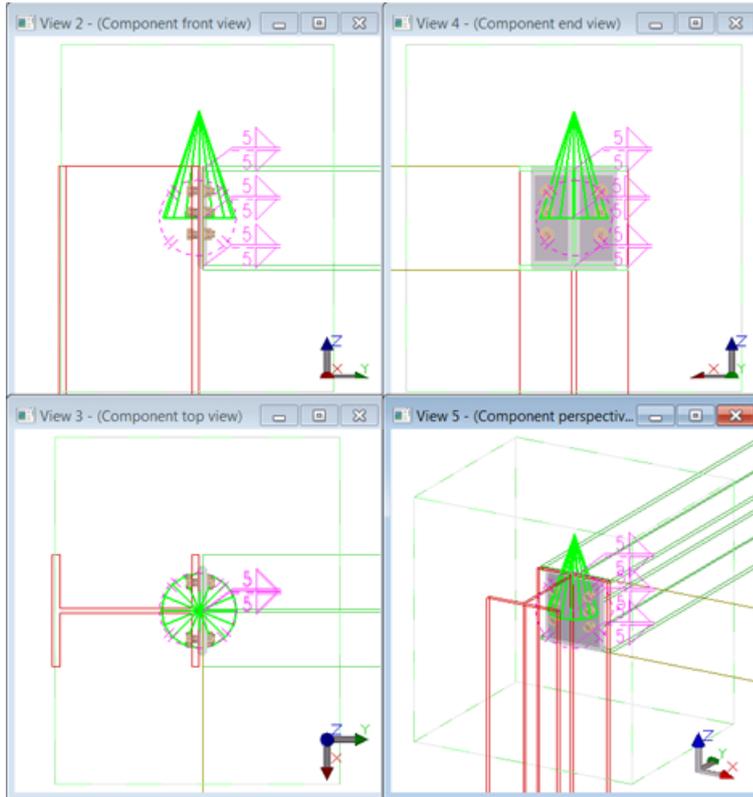
7.3 View a component in a model

You can create several views of a component to view it from different viewpoints.

1. Click the component symbol in the model to select the component.
2. Right-click and select **Create View --> Default views of component** .

Tekla Structures creates four views: front, end, top, and perspective.

The example image shows the default views of the **End plate (144)** connection.



NOTE You can check the dimensions, such as bolt locations and edge distances, using the **Measure** tool in the **Component front view**.

7.4 Component tips

Default properties

If you use a component you are unfamiliar with, use the default properties of the component. Then check what needs to be modified, and modify only few properties at a time to see how the modifications affect the component. This is quicker than trying to set all the component properties before seeing what the component actually creates.

Valid profiles

Some components work with certain profiles only. If a component is not created successfully, try entering a valid profile.

Select components switch

You can switch on the **Select components** switch  to be able to select any object that belongs to the component.

Component is not added to the model

If the component is not added to the model, check the status bar. For example, you may need to click the middle mouse button to stop selecting parts before Tekla Structures creates the component.

Using thickness to create needed parts

If a component does not by default create the parts that you need, look for options to create them. If there are no options, try entering a thickness value for the parts.

If a component creates parts that you do not need, look for options to remove them. If there are no options, enter a zero (0) as the thickness of the parts.

Many secondary parts are found

If you are using a connection that only allows one secondary part, you may see the message `Many parts found` on the status bar. This means that Tekla Structures cannot determine which parts to connect. You may have several parts in the same location, or the view may be set too deep.

7.5 How to use the Applications & components catalog

Components are stored in the **Applications & components** catalog and organized into two different types of groups: default groups are automatically available and predefined groups depend on your environment.

Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog. You can also press **Ctrl + F**.

To [use a component \(page 674\)](#), select it in the catalog and follow the instructions on the status bar to add the component to the model. Double-click the component in the catalog to open the component properties dialog box.

Groups in the catalog

Default groups and predefined groups are shown against different background color in the catalog.

Default groups are automatically available:

- **Recent** contains the 12 components and applications that you have most recently used in the model.

- **Ungrouped items** contains the components and applications that are not in any predefined group.

Ungrouped items can be, for example, imported components that have not been moved to any other group yet.

- **Applications** contains [applications \(page 664\)](#), macros and drawing plugins.

If you create your own macros, you can add them to this group.

- **Connections** contains connections and seams.
- **Detailing** contains detailing components.
- **Details** contains details.
- **Parts** contains custom parts.
- **Legacy catalog** shows the folder structure of the **Component Catalog** used in previous Tekla Structures versions if the catalog definition files are found in the standard folder search paths.

Depending on your environment, the catalog may also contain **predefined groups** for specific usage, such as **Steel** --> **Beam to beam connections**. You can create your own groups according to your needs, for example, for your own favorite connections. This way you can find these connections quickly and easily. You can also hide the groups that you are not using so that only the groups you use are visible in the catalog.

Model-specific components are shown in the modeling mode and drawing-specific components are shown in the drawing mode.

Search for a component in the catalog

To search for a component in the catalog, enter the search term in the search box. The search is case-insensitive.

Note that the search does not find catalog content that has been hidden. Select the **Show hidden items** check box to show the hidden content.

The search uses the following rules:

- Non-numeric search terms find partial matches, for example, `bolt` shows both `bolt` and `bolted` in the search result.

If you have more than one word in the search term, for example `bolt plate`, the words are automatically combined so that the search result shows the components that contain both `bolt` and `plate` in their name, description, or tags.

- Numeric (integer) search terms find the exact match, for example, 121 shows component number **121** in the search result.
You can use the *, ? and [] wildcards to search for partial numeric matches. For example, 10* finds components number **10, 110, 104, 1040**, and so on.
- You can restrict the search to specific tags, groups and types of components by using the keywords `tag`, `group`, or `type`. For example, `10 tag:advanced` finds the number **10** components that have advanced in the tag, and `type:custom` finds all custom components.

Change the view in the catalog

- Click  to show the thumbnail view.
- Click  to show the list view.
- Click  to show the compact view.

The compact view shows the thumbnail images of the group that you select from the list above the search box. You can use the compact view to have more space on the screen.

- Click  to show the normal view.

Show selected components in the catalog

Click **Show selected** to show a **Selected components** group that contains the components that are selected in the model or in the drawing.

Click **Show selected** again to hide the **Selected components** group.

The **Show selected** button is not available when you are using the search in the catalog.

TIP You can use the property pane to list the components that you have selected in the model. If you select one component, the property pane shows the name and number of the component. If you select several different components, the property pane shows lists that have the text **Varies**. Open the lists to show the names and numbers of the selected components.

If you select components and other model objects, click the **Object type list**

button  in the property pane to open a list of the selected object types, and select **Component** to list the components.

View and modify component information in the catalog

Each component has an information box that shows the type of the component and the groups the component belongs to. You can add a description for the component and tags that can be used in the search.

1. Select a component in the catalog and click the small arrow on the right to open the component information box.
2. Type a description in the **Description** box.
3. Click  to add a tag and enter a tag in the box.
4. If needed, click  again to add more tags. You can also remove tags.
5. Click outside the information box to close it.

The descriptions and tags that you add are by default saved in the `ComponentCatalog.xml` file in the model folder.

Add a thumbnail image for a component in the catalog

Components have a default thumbnail image that shows a typical situation where the component can be used. You can add several thumbnails for a component and select which thumbnail is shown in the thumbnail view in the **Applications & components** catalog.

1. Select a component in the catalog.
2. Right-click and select **Thumbnails**.
3. Click **Add thumbnail**.
4. Select an image and click **Open**. You can use any standard image format, for example, `.png`, `.jpeg`, `.gif`, `.tiff`, and `.bmp`.
5. Select the check boxes of the thumbnails that you want to show in the component information box. You can also remove thumbnails, except for the default thumbnail.
6. Click **Close**.

The thumbnail information that you add is by default saved in the `ComponentCatalog.xml` file in the model folder.

Publish a component in the catalog

You may need to use the same component with different settings in different situations. To easily use the component, you can define the settings for each situation and publish the component in the catalog.

For example, you may need **End plate (144)** in three different situations. Add **End plate (144)** once to each situation in the model. Define the settings needed and then publish each **End plate (144)** in the catalog. You will then have **End plate (144)** saved in the catalog as three separate components, each with different settings. You can use these components from the catalog in the same way as other components.

1. Add a component once to all needed situations in the model.
2. Define the desired settings for each situation.
3. Select one of the components you added in the model, right-click and select **Publish in Catalog...**
4. Enter a descriptive name for the component and click **OK**.
5. Repeat steps 3-4 for each component you added.

The components are first placed in the **Ungrouped items** group in the catalog. They have the name you entered and the thumbnail image of the original component.

You can move the components to a more appropriate group in the catalog and change the thumbnail image. For example, you can create a basic component view in each situation, and use an image of the view as the thumbnail.

Create and modify groups in the catalog

You can create groups and subgroups, and move groups to different locations in the predefined groups section in the catalog. You can add and remove components from the groups, rename the groups, and add descriptions for the groups.

To	Do this
Create a group	Right-click in the catalog and select New group.... Drag the group to the desired location.
Create a subgroup	Right-click a group in the catalog and select New group....
Name a group	Right-click a group, select Rename... and type the name.
Add components to the group	<ul style="list-style-type: none">• Select components in the catalog and drag them to another group.• Select components in the catalog, right-click and select Add to group. Then select the group to which the components are added.

To	Do this
	<ul style="list-style-type: none"> Right-click a group, select Add all to group and select the group to which all the components in the group are added. <p>Note that the components are copied, not moved, to the other groups.</p>
Remove a group	Right-click a group and select Remove from group .

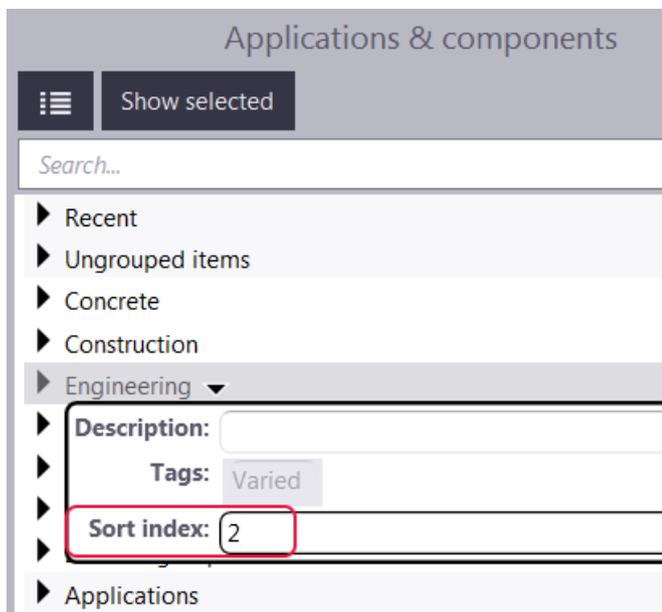
The groups that you create are by default saved in the `ComponentCatalog.xml` file in the model folder.

NOTE You cannot add or remove groups in the default groups, and you cannot modify the content of the default groups. However, you can hide the default groups and the individual items in the groups.

Change the order of groups in the catalog

You can change the order of the predefined groups in the **Applications & components** catalog. Predefined groups depend on your environment, for example, **Steel** and **Concrete** can be such groups. Note that you cannot change the order of the default groups, for example, **Applications**, **Connections**, and **Detailing**.

You can control the order with a sort index. The **Sort index** option is available in the group information of each predefined group in the **Applications & components** catalog. Sort indexes are saved in the catalog definition files.



You can change the sort index by entering either a negative or a positive integer number, or 0, in the **Sort index** option box. A negative sort index

moves a group towards the top and a positive sort index moves a group towards the bottom in the predefined groups section. Enter 0 or clear the value to revert to the default order. By default, the groups are in alphabetical order.

The sort index changes you make are model specific and they are saved in the `ComponentCatalog.xml` file in the `\model` folder. Administrators can define the order of groups for an environment or a project using the catalog definition files in the environment, firm and project folders. Do not edit these files if you are not an administrator.

Note that even if administrators have defined the order, you can still make model-specific changes to the order of the groups by entering a different sort index value for a group. If you need to revert to the default order, enter 0 as the sort index.

To change the order:

1. Select a predefined group.
2. Click the small arrow on the right to open the group information box.
3. Enter a number in the **Sort index** box.
The group is immediately moved.
4. Save the model to keep the order.

Hide groups and components in the catalog

1. Select a group or a component in the catalog.
2. Right-click and select **Hide / Unhide** to hide the group or the component.
3. To view the hidden group or component again, select the **Show hidden items** check box at the bottom of the catalog. The hidden group or component is shown as dimmed.
4. To show the hidden group or component normally, right-click it and select **Hide / Unhide**.

Show the catalog message log

If there are errors or warnings, for example, in the catalog definition files, the **Message Log** button is shown in the lower right corner in the catalog. The button is not shown if there are no errors or warnings.

To view the error log, click the **Message Log** button.

Errors and warnings are also written to the `ComponentCatalog_<user>.log` file in the `\logs` folder under the model folder.

Catalog definitions

The commands in **Access advanced features**  > **Catalog management** are used for modifying catalog definitions. Generally, there is no need to modify catalog definitions. Do not modify the definition files if you are not an administrator. For more information on administrator tasks, see *Customize the Applications & components catalog*.

7.6 Converting a conceptual or a detailed component

Depending on the Tekla Structures configuration you are using, you can create either detailed or conceptual components.

- Detailed components include all the information needed for fabrication, such as assemblies, cast units, and reinforcing bars.

Detailed components have a round symbol in the model:  or .

- Conceptual components look similar to detailed components but do not include the option to change part numbering or assembly numbering settings. Conceptual components are meant to be used as reference information for further fabrication detailing.

Conceptual components have a rectangular symbol in the model:  or .

You can create conceptual components in the **Engineering, Rebar Detailing**, and **Construction Modeling** configurations.

You can edit conceptual components and convert them to detailed components in the **Full, Primary, Steel Detailing**, or **Precast Concrete Detailing** configurations.

Modifying part properties, such as the size of the component main part, does not automatically convert a detailed component to a conceptual component, or vice versa. For example, if you use the **Engineering** configuration and modify the model, detailed components are not converted to conceptual components. However, when you modify a detailed component in the **Rebar Detailing** configuration, the component changes to a conceptual component.

You can convert components in the **Applications & components** catalog. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

Do one of the following:

To	Do this	Configuration
Convert a conceptual component to a detailed component	<ol style="list-style-type: none"> 1. Click  > Convert to detailing component. 2. Select the component symbol. 	Full, Primary, Steel Detailing, Precast Concrete Detailing
Convert a detailed component to a conceptual component	<ol style="list-style-type: none"> 1. Click  > Convert to conceptual component. 2. Select the component symbol. 	Engineering, Construction Modeling, Rebar Detailing

7.7 Automate connection creation

This section describes the tools that you can use to automate connection creation in the model.

Click the links below to find out more:

[AutoConnection \(page 689\)](#)

[AutoDefaults \(page 694\)](#)

[AutoConnection and AutoDefaults rules \(page 699\)](#)

AutoConnection

Use the AutoConnection tool to automatically select and apply connections with predefined properties to selected parts in a model. With AutoConnection, Tekla Structures automatically creates similar connections for similar framing conditions.

You can use AutoConnection to quickly add connections individually, in phases, or project-wide. This is useful when you are working on a large project using many connections, modifying a model, or importing modified profiles.

NOTE Before using AutoConnection in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use the test model to check the rules and properties of various connection types. The model also acts as a quick reference for connection information.

See also

[Define AutoConnection settings and rules \(page 690\)](#)

[Create a connection using AutoConnection \(page 693\)](#)

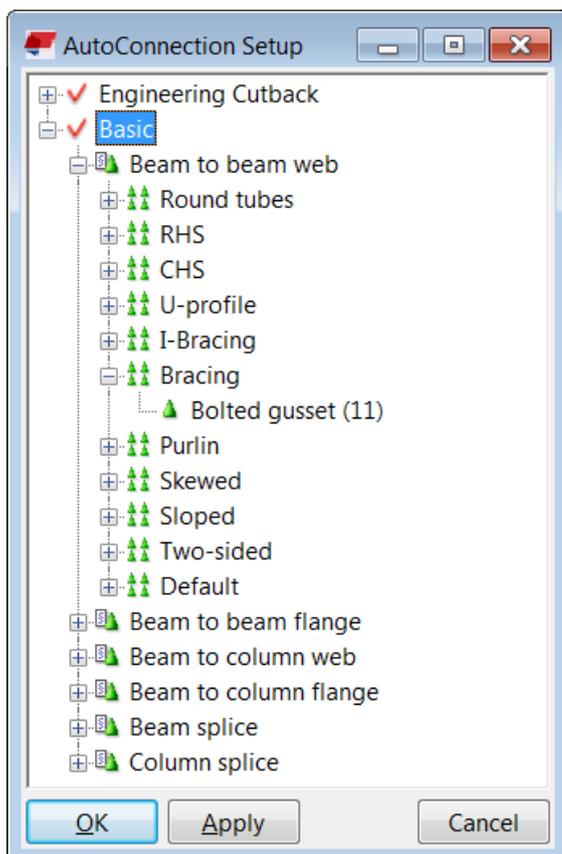
[AutoConnection and AutoDefaults rules \(page 699\)](#)

Define AutoConnection settings and rules

With AutoConnection you can define groups of rules which Tekla Structures automatically applies when creating connections in a model. By using a rule group to select connections and connection properties you do not have to select each connection and define its properties separately. For example, you can create separate rules for different standards, projects, manufacturers, and even individual models.

AutoConnection settings

To open the **AutoConnection Setup** dialog box, on the **File** menu, click **Catalogs --> AutoConnection settings**.



Icon	Setup level	Description
✓	Rule group	You can use rule groups to organize connections and connection properties according to different standards,

Icon	Setup level	Description
		projects, manufacturers, and models. You can create, modify and delete rule groups.
	Framing condition	Framing conditions are predefined connection types that you cannot change. Tekla Structures creates the framing conditions automatically: <ul style="list-style-type: none"> • Beam to beam web • Beam to beam flange • Beam to column web • Beam to column flange • Beam splice • Column splice
	Rule set	You can use rule sets to define which connection to use in a certain situation. You can create additional rule sets.
	Connection	The connection to apply if the rule set criteria are met. To apply a particular connection, the conditions in the model have to match all the rules in the branch that contains the connection.

Rules.zxt file

When you use AutoConnection, Tekla Structures saves the AutoConnection information in a zipped `rules.zxt` file in the `\attributes` folder under the current model folder.

You can copy the `rules.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoConnection setup you need to recopy this file to the firm and project folders. To use the modified setup in other models, restart Tekla Structures.

Create a rule group for AutoConnection

You can define rule groups for AutoConnection to organize connections and connection properties according to different standards, projects, manufacturers, and models.

1. On the **File** menu, click **Catalogs --> AutoConnection settings**.
2. Right-click an existing rule group and select **New Rule Group**.

3. Click the **New** group and enter a name.

Give the rule group a name that reflects the group of [connections that you want to create \(page 693\)](#). For example, use the fabricator's name, the project name, or any name that clearly identifies the connection rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing framing conditions in the group.

Create a rule set for AutoConnection

You can create AutoConnection rule sets under framing conditions to specify which connection properties to use when specific conditions in the model are met.

You only need to create AutoConnection rule sets if you plan to [use different connections \(page 693\)](#) to connect similar framing conditions. For example, in the model, some beam-to-beam connections require clip angles, others need shear tabs. You need to define rule sets to determine where each connection type should be used.

1. On the **File** menu, click **Catalogs --> AutoConnection settings**.
2. Click the plus icon in front of the rule group  to open the tree structure.
3. Right-click the relevant framing condition  and select **Create Additional Rule Sets**.
4. Right-click the new rule set and select **Edit Rule Set...**
5. Enter a name for the rule set.
6. Select a rule from the **Available rules** list.
7. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
8. Enter the values used in the rule: either an exact value, or minimum and maximum values.
9. Click **OK**.

NOTE The order of the rules in the tree structure is important. Tekla Structures uses the first rule that matches the conditions in the model so you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Change a connection in an AutoConnection rule set

You can change the connection in a rule set by selecting a connection in the **Applications & components** catalog.

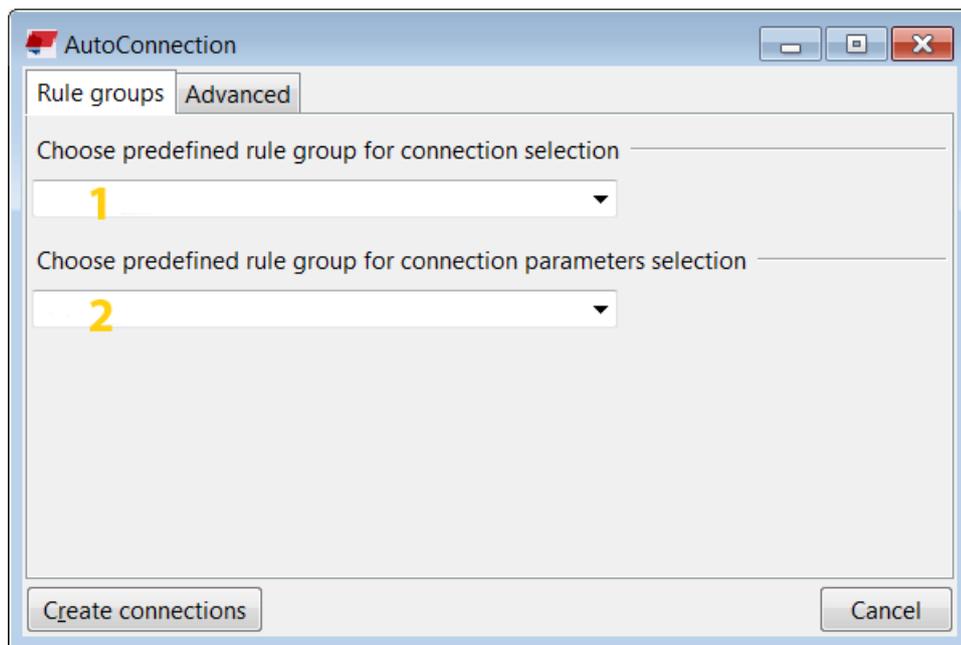
1. On the **File** menu, click **Catalogs --> AutoConnection settings**.

2. Click the plus icon in front of the relevant framing condition  and rule set  to find the connection that you want to change.
3. Right-click the connection and select **Select Connection Type....**
4. Double-click a connection in the **Select component** dialog box.
5. Click **OK** in the **AutoConnection Setup** dialog box.

Create a connection using AutoConnection

Use AutoConnection to have Tekla Structures automatically create connections using the properties of predefined rules. When you use AutoConnection, Tekla Structures ignores the properties in the connection dialog boxes. Tekla Structures does not modify the existing connections.

1. In the model, select the parts to connect.
2. On the **Edit** tab, click **Components --> Create AutoConnections.**
3. Select the rule groups from the lists on the **Rule groups** tab.



1	Rule group for AutoConnection
2	Rule group for AutoDefaults

4. If needed, go to the **Advanced** tab to change the rules used in the framing conditions:
 - a. Select the connection in the **Connection selection** option:
 - **AutoConnection** applies the connection defined in the rule group that you have selected in the first list on the **Rule groups** tab.
 - **None** does not create a connection.

- Click **Select...** to select a connection from the **Applications & components** catalog. Tekla Structures creates the connection using the default properties.
 - b. Select the connection properties in the **Parameters selection** option:
 - **Autodefaults** applies the properties of the rule group that you have selected in the first list on the **Rule groups** tab.
 - **No autodefaults** applies the default connection properties.
5. Click **Create connections**.

TIP You can also use the **Auto connect selected parts** macro to automatically create connections using the current properties without opening the **AutoConnection** dialog box.

Macros are located in the **Applications** group in the **Applications & components** catalog.

See also

[Define AutoConnection settings and rules \(page 690\)](#)

AutoDefaults

Use AutoDefaults to set up properties for existing connections. AutoDefaults allows you to modify the default connection properties and save them for use in specific circumstances. When you use AutoDefaults, Tekla Structures automatically creates the connections with the predefined AutoDefaults properties. You can also use AutoDefaults for a single connection.

For example, you can use AutoDefaults to automatically adjust the thickness of each base plate you create, according to the main part profile. If the main part profile changes, Tekla Structures automatically adjusts the thickness of the base plate.

NOTE Before using AutoDefaults in a working model, we recommend that you create a test model, and create all the connection conditions in it that you need for a particular project. You can then use this test model to check the rules and properties of various connection types. It also acts as a quick reference for connection information.

See also

[Define AutoDefaults settings and rules \(page 695\)](#)

[Modify a connection using AutoDefaults \(page 699\)](#)

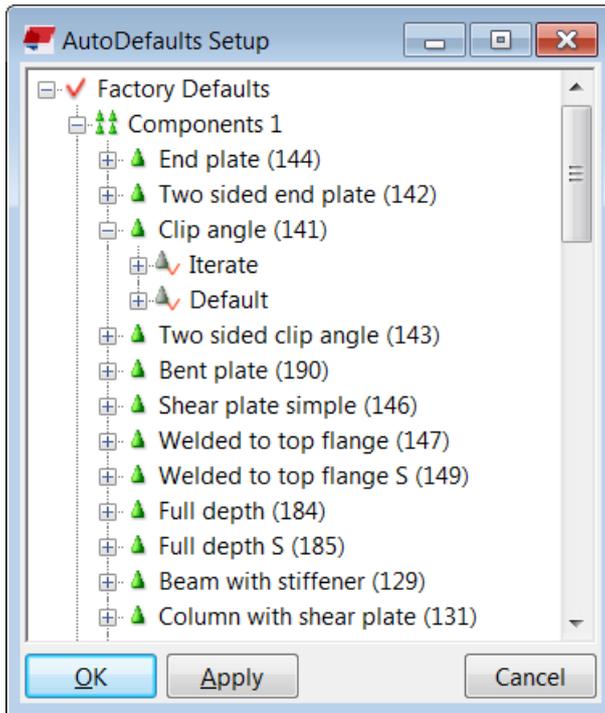
[AutoConnection and AutoDefaults rules \(page 699\)](#)

Define AutoDefaults settings and rules

Use AutoDefaults to set up properties for existing connections. With AutoDefaults you can create rules that define the situations where the predefined properties are used.

AutoDefaults settings

To open the **AutoDefaults setup** dialog box, click **File --> Catalogs --> AutoDefaults settings**.



Icon	Setup level	Description
✓	Rule group	You can use rule groups to organize settings according to different standards, projects, manufacturers, and models. You can create, modify and delete rule groups.
🌲 🌲	Components	The component tree structure shows the connections that are available on component toolbars in Tekla Structures.
🌲✓	Rule set	Rule sets control which properties to use in certain situations. You can create additional rule sets. Tekla Structures processes AutoDefaults rule sets in the order in which they are in the tree, so you can control the selection of properties.

Icon	Setup level	Description
	Properties file	<p>The properties files are under the rule sets. By default, each connection has a standard properties file that defines the standard properties, for example, <code>standard.j144</code> or <code>standard.j1042</code>.</p> <p>You can create additional properties files for the properties that you want to use again and give the files distinctive names.</p>

Defaults.zxt file

When you use AutoDefaults, Tekla Structures saves the AutoDefaults rules in a zipped `defaults.zxt` text file in the `\attributes` folder under the current model folder.

You can copy the `defaults.zxt` file to the project or firm folder to make it available in other models. Each time you modify the AutoDefaults setup, you need to recopy this file to the firm or project folder. To use the modified setup in other models, restart Tekla Structures.

NOTE We do not recommend that you edit the `defaults.zxt` file using a text editor, but if you do, ensure that you are using the right syntax. The easiest way to unzip the `.zxt` file is to change the file extension `.zxt` to `txt.gz` and unzip the file. Change the extension back to `.zxt` when you have finished. You do not need to zip the file after editing it, Tekla Structures can also read the unzipped file.

Create a rule group for AutoDefaults

You can define rule groups for AutoDefaults to group the rules according to different standards, projects, or manufacturers, for example.

1. On the **File** menu, click **Catalogs --> AutoDefaults settings**.
2. Right-click an existing rule group and select **New Rule Group**.
3. Click the **New** group to rename it.

Give the rule group a name that reflects the contents of the group. For example, use the fabricator's name, the project name, or any name that clearly identifies the rules that you want to use for a specific model.

When you create a new rule group, Tekla Structures automatically adds the existing components to the group.

Create a rule set for AutoDefaults

You can create rule sets to define which connection properties are used when specific conditions in the model are met.

1. On the **File** menu, click **Catalogs --> AutoDefaults settings**.

2. Click the plus icon in front of the rule group ✓ to open the tree structure.
3. Click the plus icon in front of the relevant group of components 🌲 and connection 🌲.
4. Right-click an existing rule set and select **New Rule Set**.
5. Right-click the new rule set and select **Edit Rule Set...**
6. Enter a name for the rule set.
7. Select a rule from the **Available rules** list.
8. Click the right arrow button to move the selected rule into the list of **Rules in rule set**.
9. Enter the values used in the rule: either an exact value, or minimum and maximum values.
10. Select from the **Parameter files selection in rule set** list how the properties are selected in the rule set.

Option	Description
Iterate until connection symbol is green	Tekla Structures uses the properties files it finds in the first matching sub-rule set and does not check other rule sets.
Iterate until connection symbol is green	Tekla Structures checks sub-rule sets until it finds matching properties.
Iterate until connection symbol is yellow	Tekla Structures checks sub-rule sets until it finds matching properties.
Use combination of all parameters	<p>Tekla Structures checks all rule sets and uses the properties files in all matching rule sets. The order of the properties files is important.</p> <p>When Tekla Structures combines the properties files, the most recent files (the lowest in the tree) override previous ones. If you do not enter any values for the properties, Tekla Structures does not override the previous property values.</p>

11. Click **OK**.

NOTE The [order of the rules \(page 702\)](#) in the tree structure is important. Tekla Structures uses the first rule that matches the conditions within the model so

you should place the most limiting rule highest in the tree, and the most generic rule lowest.

You can change the priority of a rule set by right-clicking the rule set and selecting **Move up** or **Move down**.

Modify connection properties for AutoDefaults

Each connection has a default standard property file that defines the properties for the connection. You can modify the properties that the standard file uses. Save the connection properties that you want to use and set the standard file to [use these properties \(page 699\)](#) in the AutoDefaults settings.

1. On the **File** menu, click **Catalogs --> AutoDefaults settings**.
2. Click the plus icon in front of the rule group  to open the tree structure.
3. Click the plus icon in front of the relevant group of components  and connection .
4. Right-click the `standard.j` connection file that you want to modify, for example, `standard.j144` and select **Edit Connection Parameters...**
5. In the connection dialog box, set the properties that you want to save. Such properties could be, for example, bolt properties, profiles, and materials.
6. Enter a descriptive name for the properties in the box next to the **Save as** button.
7. Copy this name in the **Connection code** option on the **General** tab. Using the same name allows you to check which properties Tekla Structures used in specific situations. Tekla Structures does not automatically show the AutoDefaults values in the connection dialog box.
8. Click **Save as**.
Tekla Structures saves the properties file in the `\attributes` folder under the current model folder. The filename consists of the name you entered in **Save as** and the file extension `.jxxx`, where `xxx` is the connection number, for example, `sec_0-190.j144`.
9. Click **Cancel** to close the connection dialog box and return to the **AutoDefaults setup** dialog box.

If you click **OK** to close the connection dialog box, you need to load the default properties the next time you use the connection. Using the default properties ensures that AutoDefaults can modify the properties.

10. Right-click the `standard.j` file again and select **Select Connection Parameters...**

The **Attribute File List** dialog box that opens contains the properties that have been set and saved in the connection dialog box.

11. Select a file in the **Attribute File List** dialog box.
12. Click **OK**.

Modify a connection using AutoDefaults

When you use a connection that you are unfamiliar with, first use the default properties. Then use AutoDefaults to modify the properties.

1. Double-click a connection symbol in the model to open the connection dialog box.
2. On the **General** tab, select a rule group from the **AutoDefaults rule group** list.
3. On all tabs, select the AutoDefaults options marked with the arrow symbol  for the properties in which you want to use AutoDefaults.
4. Click **Apply**.

If you manually modify the properties after using AutoDefaults, Tekla Structures uses the manually modified properties.

For example, you have manually set the base plate thickness of a connection to 20 mm. AutoDefaults is active and sets the plate thickness according to the main part profile. If you modify the main part profile, Tekla Structures does not update the base plate thickness. It remains at 20 mm.

NOTE You can view which AutoDefault rules and properties are used:

- To view AutoDefaults rules, select the connection symbol in the model, right-click and select **Inquire** .
Tekla Structures shows the rule group, rule sets and properties files used.
- To view the AutoDefaults properties, double-click the connection symbol in the model, select `<AutoDefaults>` in the list box next to the **Load** button and click **Load**.

See also

[Define AutoDefaults settings and rules \(page 695\)](#)

AutoConnection and AutoDefaults rules

You can create your own AutoConnection and AutoDefaults rules for project and company defaults. By defining rules you can accurately select connections and connection properties when using AutoConnection and AutoDefaults.

General rules

- **Profile name** is the name in the profile catalog.
- **Profile type**

Profile type	Number
I	1
L	2
Z	3
U	4
Plate	5
Round bar	6
Pipe	7
Square pipe	8
C	9
T	10
ZZ	15
CC	16
CW	17
Polygon plate	51

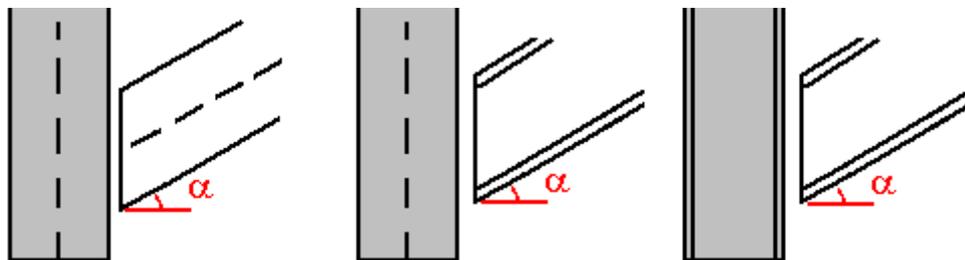
- Number of secondary parts
- Number of main parts
- Material name

Orientation rules

Depending on the relative angle of a beam, the connections can be classified as sloped, skewed, or cant. The angle value can be between - 90 and 90 degrees.

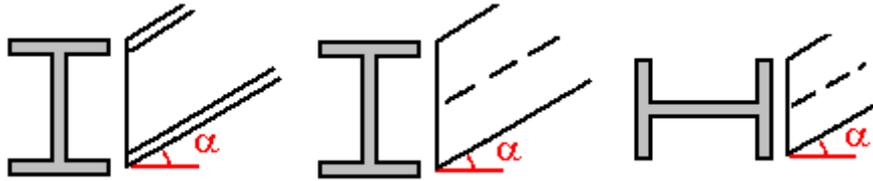
- **Sloped** angle (relative to main part cross section)

The longitudinal axis of the secondary part follows the slope of the longitudinal axis of the main part.



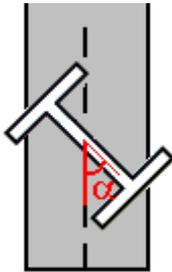
- **Skewed** angle (relative to main part longitudinal axis)

The longitudinal axis of the secondary part is skewed according to the main part cross section. The angle is the smaller of the angles between the longitudinal axis of the secondary part and the main part Z or Y axis.



- **Cant angle**

For rotated secondary parts



Dimension rules

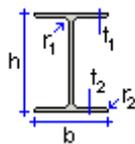
- **Profile depth**

- **Web depth**

For profiles with an upper and lower flange, the web depth is: $h - t_1 - t_2 - 2 * r_1$

Or, if t_2 is zero: $h - 2 * t - 2 * r_1$

For profiles with one flange, the web depth is $h - t - r_1 - r_2$.



- **Web thickness**

- **Flange thickness**

Forces and strengths

- Shear force
- Axial force
- Bending moment

See also

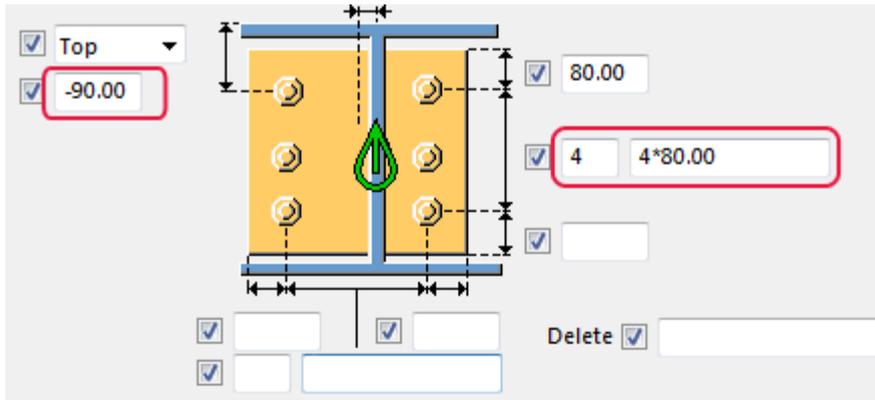
[Combining and iterating properties for AutoDefaults \(page 702\)](#)

[AutoDefaults example: Using iteration with connection check \(page 704\)](#)

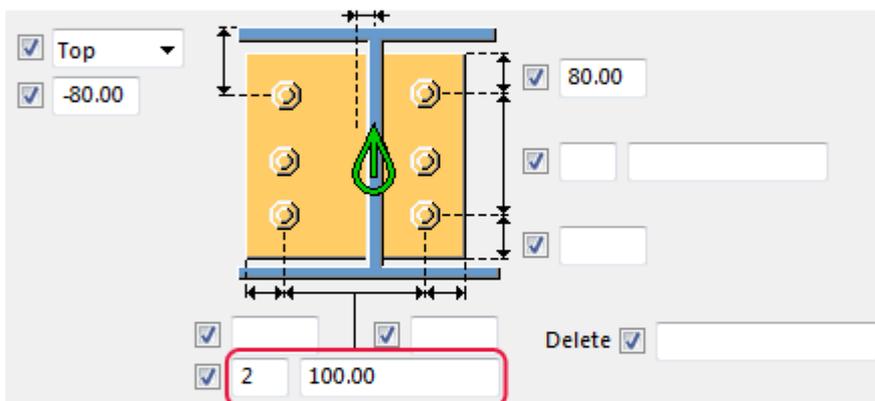
[Using reaction forces and UDLs in AutoDefaults and AutoConnection \(page 706\)](#)

Combining and iterating properties for AutoDefaults**Combining properties**

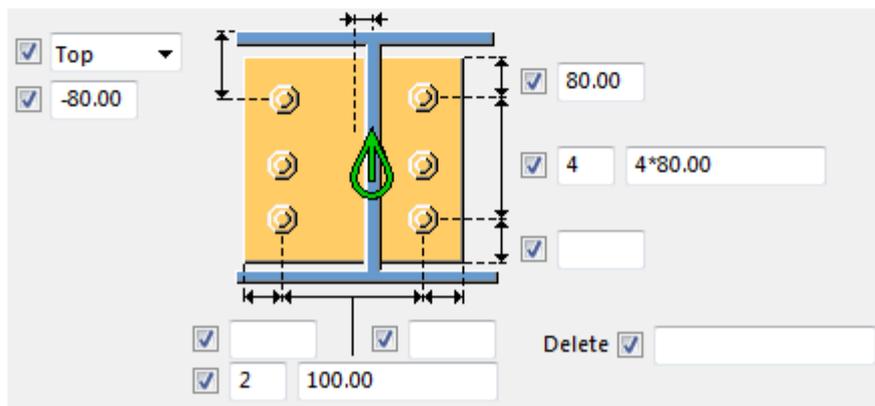
You can save properties files that cover different groups of properties and use these files to define many rules. For example, you can have one file for bolt properties and another for profile properties. AutoDefaults combines the separate files into one file. This means that you can define fewer files because you can use one file for several rules. If the files contain different values for the same property, Tekla Structures uses the last property it finds, see the example image below.



+



=



Iterating properties

Tekla Structures tests the properties until the connection symbol is yellow or green. Iteration changes the properties automatically if the connection is not created successfully, even if the rules match. If connection check is active, the iteration results in properties that have passed the check.

Limitations

- Tekla Structures cannot iterate property files directly. Use a single iteration rule set with sub-rule sets.
- You cannot have many parallel iteration rule sets. Use a single iteration rule set and place it just before the default rule set.
- Place the combination rule sets above the iteration rule set in the AutoDefaults tree structure.
- Combination rule sets can only be one level deep.
- Tekla Structures disregards empty rule sets, so include at least one rule in each rule set.

See also

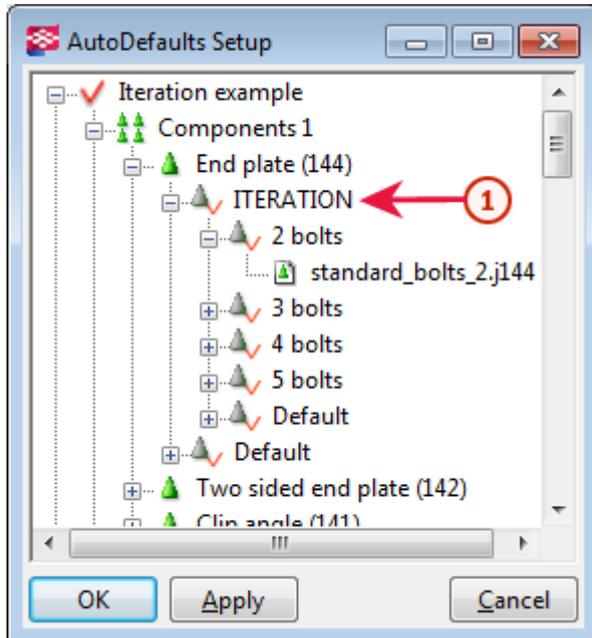
[Define AutoDefaults settings and rules \(page 695\)](#)

AutoDefaults example: Using iteration with connection check

You can use the connection check result when using AutoDefaults with iteration. If an iteration rule matches, but the connection does not pass the connection check and the connection symbol remains red, AutoDefaults continues testing other rules and properties until the connection symbol is green.

In this example, you will create iteration rules to set the number of bolts according to the result of the connection check. After this, you will use the

rules group and connection check together for a connection. The example image below shows the rules in the **AutoDefaults setup** dialog box.



To create iteration rules for use with connection check:

1. On the **File** menu, click **Catalogs --> AutoDefaults settings**.
2. Right-click the tree and select **New Rule Group**.
3. Click the new rule group and rename it to *Iteration example*.
4. Browse the *Iteration example* tree to find **End plate (144)**, right-click it, and select **Create Additional Rule Sets**.
5. Right-click the **New** rule set and select **Edit Rule Set**.
6. Change the rule set name to *ITERATION*.
7. Set the **Parameter file(s) selection in rule set** option to **Iterate until connection symbol is green**.
8. Click **OK**.
9. Right-click the *ITERATION* rule set and select **Create Additional Rule Sets**.
10. Right-click the **New** rule set and select **Edit Rule Set**.
11. Change the rule set name to *2 bolts*.
12. Select the rule **Secondary 1 depth** and set the minimum and maximum depth values for two bolts.
13. Set the **Parameter file(s) selection in rule set** option to **Use combination of first parameters**.
14. Click **OK**.

15. Right-click the connection properties file `standard.j144` under `2 bolts` and select **Select Connection Parameters**.
16. Select a properties file for two bolts in the **Attribute File List** and click **OK**.

TIP If there is no suitable properties file, you can create a new file. Right-click the `standard.j144` file and select **Edit Connection Parameters**. Save the needed properties and click **Cancel** to close the dialog box. The saved properties are now available in the **Attribute File List**.

17. Click **Apply** to have the changes available in the connection dialog box.
18. Repeat steps 9 to 16 for other rule sets.
19. Open the **End plate (144)** dialog box.
20. Select `<Defaults>` from the list next to the **Load** button and click **Load**.
21. On the **General** tab, set the **AutoDefaults rule group** option to the `Iteration example` you created.
22. On the **Design type** tab, set the **Check connection** option to **Yes**.
23. Enter the load from secondary members in the **Shear, Tension, and Moment** options.
24. Click **OK**.

See also

[Define AutoDefaults settings and rules \(page 695\)](#)

[Combining and iterating properties for AutoDefaults \(page 702\)](#)

Using reaction forces and UDLs in AutoDefaults and AutoConnection

You can set reaction forces for AutoConnection and AutoDefaults in the user-defined attributes of a part, and for AutoDefaults also on the **Design** tab in the connection dialog box.

Reaction forces

When you use reaction forces in a rule and AutoDefaults is activated, Tekla Structures first searches for reaction forces in the corresponding connection's properties. If the properties do not contain reaction forces, Tekla Structures searches the user-defined attributes of the secondary part of the connection. If Tekla Structures does not find forces there, you cannot use reaction force rules.

Shear force calculation

If you have not given any reaction force values, shear force is calculated using the UDL (uniformly distributed load) shear force routine. The UDL calculation

is mainly intended for use with imperial units. It uses the yield stress value, profile dimensions, and UDL percentage to calculate the maximum shear force allowed.

- Yield stress is defined in the material catalog.
- Profile dimensions come from the profile catalog.
- UDL percentage is taken either from the connection dialog box or from an advanced option.

Tekla Structures compares the result with the shear force rule in AutoDefaults.

To use UDLs for AutoConnection and AutoDefaults:

To	Do this
Use UDL for AutoConnection	<ol style="list-style-type: none"> 1. On the Design tab in the connection dialog box, set the UDL option to Yes. 2. Enter the UDL percentage in the UDL% box. If you do not enter any value, Tekla Structures uses a default percentage set with the <code>XS_AUTODEFAULT_UDL_PERCENT</code> advanced option.
Use UDL for AutoDefaults	<ol style="list-style-type: none"> 1. On the Design tab in the connection dialog box, set the Use UDL option to Yes. 2. Enter the UDL percentage in the UDL % box. If you do not enter any value, Tekla Structures uses a default percentage set using the <code>XS_AUTODEFAULT_UDL_PERCENT</code> advanced option.

See also

[Design and Design type tabs \(page 731\)](#)

7.8 Advanced component settings

This section describes how to set default properties for different connection types, how to use Excel spreadsheets in connection design, and advanced component properties, such as analysis and design properties.

Click the links below to find out more:

[Define connection properties in the joints.def file \(page 708\)](#)

[Excel spreadsheets in connection design \(page 720\)](#)

[General tab \(page 730\)](#)

[Design and Design type tabs \(page 731\)](#)

Define connection properties in the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types. You can use the `joints.def` file to set the default properties for different connection types. `Joints.def` is a text file that you can open and edit in any standard text editor.

Tekla Structures uses the values defined in the `joints.def` file for the properties that do not have values in the connection dialog boxes. If you manually enter values in the connection dialog boxes, the manually entered values are used instead of the values in the `joints.def` file. AutoDefaults also override the values defined in the `joints.def` file.

Tekla Structures stores the `joints.def` file in the system folder. Tekla Structures searches for the `joints.def` file in the standard search order: model, project, firm, and system folder.

How to use the joints.def file

The `joints.def` file contains general connection settings and connection-specific settings for different connection types in separate sections. You can modify the `joints.def` file using any standard text editor.

When you modify the file:

- Enter absolute values or names.
- Do not use feet and inch symbols.
- Ensure that the profiles exist in the profile catalog.
- Ensure that the bolts exist in the bolt catalog.
- You can set the measurement units at the beginning of the file.
- You can define in the `JOINTDEFAULT` line whether Tekla Structures uses the default values in the `joints.def` file or the system default values, for example, as follows:

```
// is default file available (1) or not (0)  
JOINTDEFAULT 1
```

- Value 1 means that the default values defined in the `joints.def` file are used.
- Value 0 means that the system default values are used.
- The `//` characters at the beginning of a line mean that the line is a comment line. Tekla Structures does not use the information on these lines.

- You can force Tekla Structures to use the system default for a particular property by entering the value `-2147483648` for the property.

Connection-specific properties

The properties for clip angles, shear tabs, end plates, gusset connections and diagonal connections are in separate sections. Each section begins with a header row that contains the column labels, for example as follows:

```
joints.def
// name          part      lproflength  diameter  number_of_bolts
BOLTHEIGHT      GUSSET    100          20.0      2
```

Do not add columns to the file. If Tekla Structures cannot find a property in the connection-specific section, it searches for the default property in the general defaults section.

Connections that use the joints.def file

The following connections use the `joints.def` file:

- **Welded gusset (10)**
- **Bolted gusset (11)**
- **Bracing cross (19)**
- **Tube gusset (20)**
- **Tube crossing (22)**
- **Two sided angle cleat (25)**
- **Corner tube gusset (56)**
- **Corner bolted gusset (57)**
- **Wraparound gusset (58)**
- **Hollow brace wraparound gusset (59)**
- **Wraparound gusset cross (60)**
- **Wrapped cross (61)**
- **Gusseted cross (62)**
- **Corner wrapped gusset (63)**
- **Beam with stiffener (129)**
- **Column with shear plate (131)**
- **Bolted moment connection (134)**
- **Clip angle (141)**
- **Two sided end plate (142)**
- **Two sided clip angle (143)**
- **End plate (144)**
- **Shear plate simple (146)**

- **Welded to top flange (147)**
- **Welded to top flange S (149)**
- **Moment connection (181)**
- **Column with stiffeners W (182)**
- **Full depth (184)**
- **Full depth S (185)**
- **Column with stiffeners (186)**
- **Column with stiffeners S (187)**
- **Column with stiffeners (188)**
- **Shear plate tube column (189)**
- **Bent plate (190)**

Example: How Tekla Structures uses the joints.def file

This example explains how Tekla Structures calculates the bolt diameter and other properties of the **Bolted gusset (11)** connection using the `joints.def` file.

The height of the diagonal profile is 10". Tekla Structures calculates the bolt size and the number of bolts according to the profile height. It searches the `BOLTHEIGHT` rows for a profile height of 10".

As the profile height is greater than 8.0 but under 12.0, Tekla Structures uses the row with profile height 8.0. This sets the bolt diameter to 0.75.

```
// DIAGONAL JOINTS
// diagonal default bolt diameters depending on prof height, higher prior than
//
// name      part      profileheight  diameter  number_of_bolts
BOLTHEIGHT  DIAGONAL  3.0           0.75     1
BOLTHEIGHT  DIAGONAL  8.0           0.75     2
BOLTHEIGHT  DIAGONAL  12.0          0.75     3
BOLTHEIGHT  DIAGONAL  16.0          0.75     4
BOLTHEIGHT  DIAGONAL  18.0          0.75     5
```

Tekla Structures uses the bolt diameter to assign the bolt and part properties. It searches the `DIAGBOLTPART` rows for bolt diameter 0.75.

```
// name      bolt diameter  angle profile  conn.plate thickness | horizontal bolts | vertical bolts | edge.
DIAGBOLTPART 0.5         L4X3X1/2      0.375      2         1.5         1.0         -2147483648 -2147483648 1.0
DIAGBOLTPART 0.75        L4X4X1/2      0.375      2         2.5         1.5         -2147483648 -2147483648 1.5
DIAGBOLTPART 1.0         L5X5X1/2      0.375      2         3.0         2.0         -2147483648 -2147483648 2.0
```

The following property values are used:

Bolt diameter	0.75
Number of bolts horizontally	2
Edge distance horizontally	1.5

Edge distance vertically	1.5
Distance between bolts horizontally	2.5
Distance between bolts vertically	System default is used.

Tekla Structures does not use the connection plate thickness or angle profile properties in this connection.

General defaults in the joints.def file

Tekla Structures uses the general defaults in the `joints.def` file if it cannot find a connection property in the connection-specific section.

For example, for clip angles, Tekla Structures determines the bolt diameter and the number of bolts according to the secondary beam height. If the secondary beam is higher than the highest value in the clip angle section in the `joints.def` file, Tekla Structures uses the default bolt diameter in the general defaults.

The properties in the general defaults section in the `joints.def` file are:

Property	Description
<code>boltdia</code>	Bolt diameter
<code>pitch</code>	Distance from the center of one bolt to the center of the next bolt
<code>clipweld</code>	Weld size
<code>angle-cc-inc</code>	Tekla Structures adds bolt to bolt distance and web thickness, then rounds up the result using this value. Complies with the US AISC standard.
<code>lprofgapinc</code>	Tekla Structures rounds up the angle profile gap using this value. Complies with the US AISC standard.
<code>lsize</code>	Size of the angle profile
<code>copedepth</code>	Notch size
<code>copelength</code>	Notch size
<code>boltedge</code>	Edge distance
<code>webplatelen</code>	Haunch plate height (h)
<code>webplatewid</code>	Haunch plate width (b)
<code>beamedge</code>	Setback distance between the end of the beam and the main part
<code>knifeclr</code>	No longer used
<code>clipedge</code>	Edge distance for bolts (clip angles only)

Property	Description
gap	No longer used
shearplatethk	Shear tab thickness
endplatethk	End plate thickness
shearweld	Size of weld
cliplsize	Size of angle profile (clip angles only)
flangecutclear	Flange cut clearance
slotsize	Size of slotted hole
clipslots	Part with slotted holes: <ul style="list-style-type: none"> • 1 = beam • 2 = angle profiles • 3 = both This property is the Slots in option on the Bolts tab.
clip_attac	Clip angle attached to the main part and secondary parts: <ul style="list-style-type: none"> • 1 = both parts bolted • 2 = main part bolted/secondary part welded • 3 = main part not welded • 4 = main part welded/secondary part bolted • 5 = both parts welded • 6 = main part not bolted • 7 = secondary part not welded • 8 = secondary part not bolted • 9 = both parts bolted/welded This property is the bolt attachment option on the Bolts tab where the location of bolts is defined.
copedepth_inc	Tekla Structures rounds up notch depth using this value.
copelength_inc	Tekla Structures rounds up notch length using this value.

Bolt diameter and number of bolts in the joints.def file

In the `joints.def` file, the `BOLTHEIGHT` rows in each connection-specific section show the default bolt diameter and default number of bolt rows for the connection type.

Tekla Structures determines the bolt diameter and the number of bolts according to the connection type based on the following properties:

For	According to
Clip angles	Secondary beam height
Shear tabs	Secondary beam height
End plates	Secondary beam height
Gusset connections	Angle profile length
Diagonal connections	Profile height

Clip angle, shear tab, and end plate connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows vertically according to the height of the secondary beam. You can enter the following properties:

Property	Description
name	BOLTHEIGHT
part	ANGLECLIP
sec.beam.height	Maximum height of the secondary beam for a certain number of bolts
diameter	Bolt diameter. The diameter must exist in the bolt catalog.
number_of_bolts	Number of bolts vertically

Gusset connections

Tekla Structures calculates the default bolt diameter and the number of bolt rows horizontally according to the length of the angle profile. You can enter the following properties:

Property	Description
name	BOLTHEIGHT
part	GUSSET
lproflength or angleproflength	Length of the angle profile
diameter	Bolt diameter. The diameter must exist in the bolt catalog.
number_of_bolts	Number of bolts horizontally

Diagonal connections

Tekla Structures calculates the default bolt diameter and number of bolt rows horizontally according to the profile height. You can enter the following properties:

Property	Description
name	BOLTHEIGHT
part	DIAGONAL
conn.pl.height Or profileheight	Profile height
diameter	Bolt diameter. The diameter must exist in the bolt catalog.
number_of_bolts	Number of bolts horizontally

Bolt and part properties in the joints.def file

Once Tekla Structures has used the `joints.def` file to calculate the bolt diameter, it uses the result to assign other properties to bolts and parts, according to the connection type.

In clip angle connections, for example, the default properties for bolts and parts are in the rows that begin with `ANGLECLBOLTPART` in the `CLIP ANGLE` section of the `joints.def` file.

The table below lists the properties that you can assign for bolts and parts in each connection type.

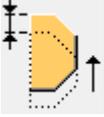
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
name	Identifies the connection type. For example, <code>GUSSETBOLTPART</code> for gusset connections.	*	*	*	*	*
bolt diameter	The bolt diameter must exist in the bolt catalog.	*	*	*	*	*
shear plate thickness	Thickness of the shear tab		*			
end plate thickness	Thickness of the end plate			*		

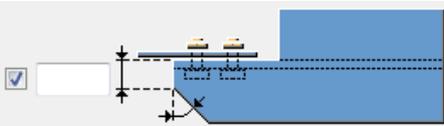
Property	Description	Clip angle	Shear tab	End plate	Gusset	Diagonal
gusset thickness	Thickness of the gusset plate				*	
conn. plate thickness	Thickness of the connection plate					*
angle profile or L profile	The name of the used angle profile must exist in the profile catalog. Enter the exact profile, for example: L100*100*10.	*			*	*
number	Number of bolts in each row vertically and horizontally.	*	*	*	*	*
pitch	Distance between the bolts from the center of each bolt for vertical and horizontal bolts	*	*	*	*	*
edge distance	Distance from the center of a bolt to the edge of the part for vertical and horizontal bolts	*	*		*	*
vert. bolt first hole	Position of the first vertical row of bolts	*	*		*	

Gusset connection properties in the joints.def file

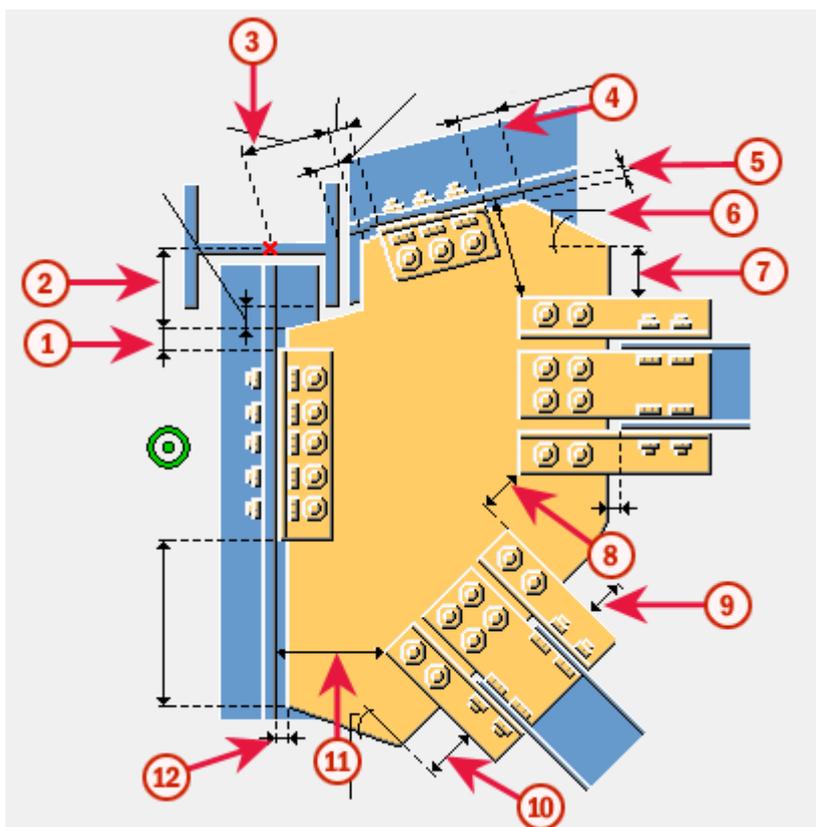
Enter the additional default properties for gusset connections in the GUSSETDEFDIM row . All gusset connections do not use all the properties.

Property	Description	Affects plate shape
name	GUSSETDEFDIM	
boltdia_def	Bolt diameter for all bolt groups Tekla Structures uses this value if the Bolt size box is empty in the connection dialog box.	
tol_prim	Tolerance between the gusset and main part web	

Property	Description	Affects plate shape
tol_sec	Tolerance between the gusset and secondary part web	
dist_diag_prim	Clearance between the first secondary part selected and the main part	
dist_diag_sec	Perpendicular distance from the last secondary part selected to the nearest secondary part	
angle_first_corner	Corner angle dimension	Yes
angle_sec_corner		
dist_between_diag	Clearance between braces	
first_bolt_from_line	Bolt edge distance for the bolt groups on the Gusset tab	
corner_dx	Corner dimension	
corner_dy	Corner dimension	
movey	 option on the Gusset tab	
movez	 option on the Gusset tab	
dist1	Edge length of the gusset plate perpendicular to the lowest brace	Yes
dist2	Edge length of the gusset plate perpendicular to the braces	Yes
dist3	Edge length of the gusset plate perpendicular to the uppermost brace	Yes
tol_lprof	Edge tolerance from gusset plate to connection plate	
tol_stiffener	Stiffener tolerance	
chamfer_dx	Stiffener chamfer dimension on the Gusset tab	
chamfer_dy	Stiffener chamfer dimension on the Gusset tab	

Property	Description	Affects plate shape
chamfer_corner_dx		
chamfer_corner_dy		
side_length	Side length	
diafit_length	Fit length in the Bracing cross (19) connection. Tekla Structures uses this value if the option on the Parameters tab is empty. 	

The example image below shows the properties of the **Wraparound gusset (58)** connection on the **Picture** tab.

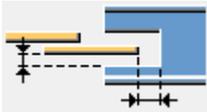
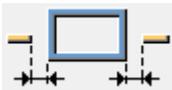


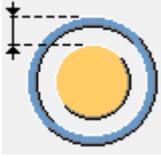
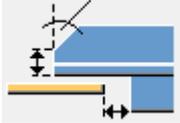
1. tol_lprof
2. corner_dy
3. corner_dx

4. dist_diag_sec
5. tol_sec
6. angle_sec_corner
7. dist3
8. dist_between_diag
9. dist2
10. dist1
11. dist_diag_prim
12. tol_prim

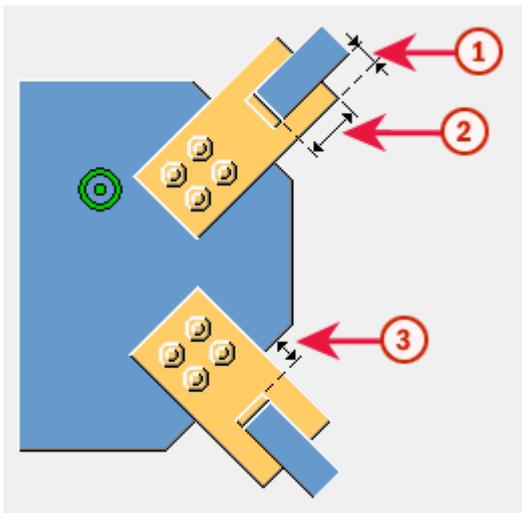
Diagonal connection properties in the joints.def file

Enter the additional default properties for bolts and parts in the `DIAGDEFDIM` row. All diagonal connections do not use all the properties.

Property	Description
name	DIAGDEFDIM
boltdia_def	Bolt diameter for all bolt groups Tekla Structures uses this value if the Bolt size box is empty in the connection dialog box.
dist_gus_diag	Gap between the gusset plate and the brace If the tube profiles are closed with end plates, <code>dist_gus_diag</code> is the gap between the gusset plate and the end plate. See the Tube crossing (22) image below.
dist_in	Cut depth in the brace. Enter a negative value to prevent the connection plate from being inside the tube brace. See the Tube crossing (22) image below.
dist_dv	Brace edge distance to the edge of the connection plate. This dimension changes the width of the connection plate. See the Tube crossing (22) image below.
sec_cut_tol	On the Brace conn tab:
slot_length_tol	
tube_cut_tol	On the Brace conn tab: 

Property	Description
conn_cut_dx	On the Brace conn tab: 
conn_cut_dy	
round_plate_tool	On the Brace conn tab: 
flanges_cut_angle	On the Brace conn tab: 
dist_flanges_cut	
dist_skew_cut	
end_plate_thk	

The example image below shows the properties of the **Tube crossing (22)** connection on the **Picture** tab:



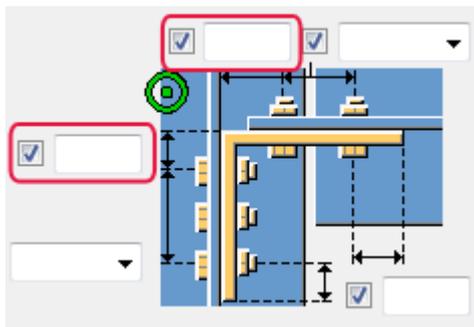
1. dist_dv
2. dist_in
3. dist_gus_diag

Profile dependent bolt dimensions in the joints.def file

For some connections, such as **Clip angle (141)** and **Two sided clip angle (143)**, Tekla Structures calculates the bolt size according to the profile size.

For these connections, Tekla Structures takes the bolt size from the PROFILEBOLTDIM rows of the PROFILE TYPE-DEPENDENT BOLT

DIMENSIONS section in the `joints.def` file if you leave the corresponding options empty on the **Bolts** tab.



Property	Description
width	Profile width
one bolt firsthole	For single bolts, distance from the edge of the profile angle to the first hole
two bolts firsthole	For two bolts, distance from the edge of the profile angle to the first hole
pitch	Distance between bolts from the center of each bolt, for vertical and horizontal bolts

For example, to find the bolt dimensions to be used with an L6X6X1/2 profile in a clip angle connection:

1. Tekla Structures first searches the `PROFILEBOLTDIM` rows for L6X6X1/2 in the `PROFILE TYPE-DEPENDENT BOLT DIMENSIONS` section.
2. If there is no match, Tekla Structures then searches the `ANGLECLBOLTPART` rows in the `CLIP ANGLE` section.

Excel spreadsheets in connection design

You can use Excel spreadsheets in connection design for all steel connections that have the **Design** or **Design type** tab in the connection dialog box.

You can link connections to Excel spreadsheets by selecting **Excel** in the **External design** option on the **Design** or **Design type** tab. The connection information is transferred to the connection type-specific spreadsheet where the needed calculations are made. The calculated properties are saved to an output file and the modified component property values are transferred back to the connection. The connection is modified according to the changes.

You can create an Excel spreadsheet for a connection type using the `component_template.xls` file available in the `..\Tekla Structures\<version>\Environments\common\exceldesign` folder, or use a predefined file.

Files used in Excel spreadsheet connection design

The following files are used in the connection design with Excel spreadsheets:

File	Description
Visual Basic script file	<p>The <code>Excel.vb</code> file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations. The file is located in the <code>..\Tekla Structures\<version>\Environments\common\exceldesign</code> folder.</p> <p>Excel searches for the relevant spreadsheet file in the following order:</p> <ol style="list-style-type: none"> 1. From the <code>\exceldesign</code> folder in the current model folder: file named as <code>component_ + number or name + .xls</code>, for example, <code>..\test_model\exceldesign\component_144.xls</code>. 2. From the location defined with the <code>XS_EXTERNAL_EXCEL_DESIGN_PATH</code> advanced option as follows: <pre>XS_EXTERNAL_EXCEL_DESIGN_PATH (= %XS_DIR%\environments\common \exceldesign\) + "component_" + number + ".xls"</pre>
Component type-specific Excel spreadsheet	<p>The component type-specific spreadsheet contains predefined calculations. When you run the connection design, the connection properties and information of the main and secondary parts are transferred to the Input and Component sheets of the Excel spreadsheet.</p>
Connection specific result file	<p>The result file contains the modified connection properties.</p> <ul style="list-style-type: none"> • The result file is created automatically from the Calculation sheet. • The file is stored in the <code>\exceldesign</code> folder in the model folder and named with the GUID, Globally Unique Identifier. • The file is updated each time you modify the connection. • The calculation results can be stored as an Excel spreadsheet, or in HTML or PDF format, depending on how the calculation spreadsheet is configured.

File	Description
Template spreadsheet	The ..\Tekla Structures\<>version>\Environments\common\exceldesign folder contains a component_template.xls spreadsheet you should use to create your own spreadsheet applications to use with Tekla Structures components.

Example of an Excel spreadsheet in connection design

The images in this example show the Excel spreadsheet that is used for the **End plate (144)** connection.

The sample spreadsheet has the following sheets:

The **Calculation** sheet contains a report of the calculations.

	A	B	C	D	E	F	G	H	I	J	K
1											
2											
3											
4											
5											
6											
7											
8											
9											
10											
11											
12											
13											
14											
15											
16											
17											
18											
19											
20											
21											
22											
23											
24											
25											
26											
27											
28											
29											
30											
31											
32											
33											
34											
35											
36											
37											
38											
39											
40											
41											
42											
43											
44											
45											
46											
47											
48											
49											
50											
51											
52											



Endplate Unity Check

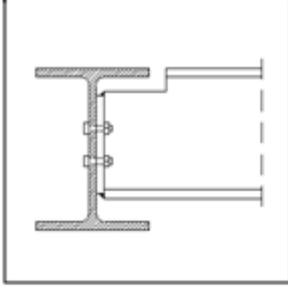
Contract Tekla Sample

Ref: User

Date: 17.03.05 4:52:44 PM

Calculated according to
BS

Connection Referen 130



Framing Condition Beam - Beam

Section			
Primary Section	IPE300	S235JR	
Secondary Section	IPE300	S235JR	
Endplate Size	10 mm	S275JR	
Plate (length*width)	200 * 180 mm		

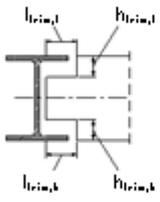
Bolt		
Bolt	20	
Bolt Grade	7990	
Shear area	Thread	
Screw Thread	Rollad	
Endplatelength in calculation		

Parameters of Connection

$e_{1,u}$ = 40 mm	a_{weld} = 6 mm	$e_{k,all}$ = 66 mm
$e_{1,b}$ = 40 mm	s_1 = 60 mm	$n_{s,prism}$ = 3
e_2 = 40 mm	s_2 = 67,09999%	$n_{s,prism}$ = 2

Notch

$l_{riv,u}$ = 82 mm	$h_{riv,u}$ = 26 mm
$l_{riv,b}$ = 82 mm	$h_{riv,b}$ = 26 mm



Shear of the endplate
$F_{t,u,d}$ = 261 kN [BS 5950-1:2000 6.2.3/6.2.4]
Bearing of the endplate
$F_{t,u,d}$ = 606 kN [BS 5950-1:2000 6.3.3.3]
Shear of beam near the weld
$F_{t,u,d}$ = 200 kN [BS 5950-1:2000 6.8.7.3]
Shear (& Tension) on the bolts
$F_{t,u,d}$ = 230 kN $F_{t,u,d}$ = 23 kN [BS 5950-1:2000 6.3.2/6.3.4.3]

The **Inputs** sheet contains the properties of the connection from the connection dialog box.

	A	B	C	D	E	F
1				Attribute	Value	Type
2		Plate				
3			Material	mat		string
4			Thickness	tpl1	10	double
5			Depth	hpl1	-2147483648	double
6			Width	bpl1	180	double
7						
8		Bolt				
9			Diameter	diameter		double
10			Grade	screwdin		string
11				lbd	-2147483648	string
12				lwd	-2147483648	string
13				lba	-2147483648	double
14				nb	-2147483648	int
15				nw	-2147483648	int
16				rb1	-2147483648	double
17				rb2	-2147483648	double
18				rw1	-2147483648	double
19				rw2	-2147483648	double
20						
21		Weld				
22				w3_size	-2147483648	double
23						
24		Notch				
25				t_cut_length	-2147483648	double
26				t_cope_length	-2147483648	double
27				b_cut_length	-2147483648	double
28				b_cope_depth	-2147483648	double
29						
30		Loading				
31				designcode	0	int
32				END		

The **Outputs** sheet contains the design results. These values are transferred to the connection and the connection in the model is modified accordingly.

The **Component** sheet contains calculations, information on the connection geometry, and on the main part and the secondary parts. The component attributes in the spreadsheet are the same as in the corresponding `.inp` file. See more about `.inp` files in Input files.

	A	B	C	D
1	Connection	Attribute	Value	
2	Connection id in model	id	130	
3	Connection class	group	99	
4		flags	50	
5	Number of the connection	jointnumber	144	
6	Local x-coordinate of Connection up direction	up.x	0	
7	Local y-coordinate of Connection up direction	up.y	0	
8	Local z-coordinate of Connection up direction	up.z	1000	
9	Model Directory	ModelDirectory	C:\TeklaStructuresModels\	
10		END		
11			Primary	Secondaries
12		attribute	value	value 1
13	Primary and secondary ids	id	108	70
14	PartCoordinateSystem	x.x	-9,11626E-13	6000
15	y-coordinate of part origin (first end) point	x.y	8000	-9,13758E-13
16	z-coordinate of part origin (first end) point	x.z	-150	-150
17	x-coordinate of second end point of part	y.x	12000	6000
18	y-coordinate of second end point of part	y.y	8000	8000
19	z-coordinate of second end point of part	y.z	-150	-150
20	x-coordinate of parts up direction point	z.x	-9,11626E-13	6000
21	y-coordinate of parts up direction point	z.y	8000	-9,13758E-13
22	z-coordinate of parts up direction point	z.z	850	850
23	PartExtrema			
24	Minimum x value of primary or secondary part	min.x	-9,11626E-13	5925
25	Minimum y value of primary or secondary part	min.y	7925	-9,13758E-13
26	Minimum z value of primary or secondary part	min.z	-300	-300
27	Maximum x value of extrema	max.x	12000	6075
28	Maximum y value of extrema	max.y	8075	8000
29	Maximum z value of extrema	max.z	0	0
30	FramingCondition			
31	Member type (Column, Beam)	Type	1	1
32	Profile name	Name	COLUMN	BEAM
33	Profile type	ProfileType	1	1
34	Skew angle between primary/sec	SkewAngle		0
35	Slope angle between primary/sec	SlopeAngle		0
36	Cantilever angle between primary/sec	AngleCant		90
37		Offset		0
38	Shear force at connection end of the beam	ShearForce		-2147483648
39	Axial force at connection end of the beam	AxialForce		-2147483648
40	Moment at connection end of the beam	BendingMoment		-2147483648
41	Use uniformly distributed load	UseUDL		0
42	How many percents from maximum uniformly	UDLPercent		0

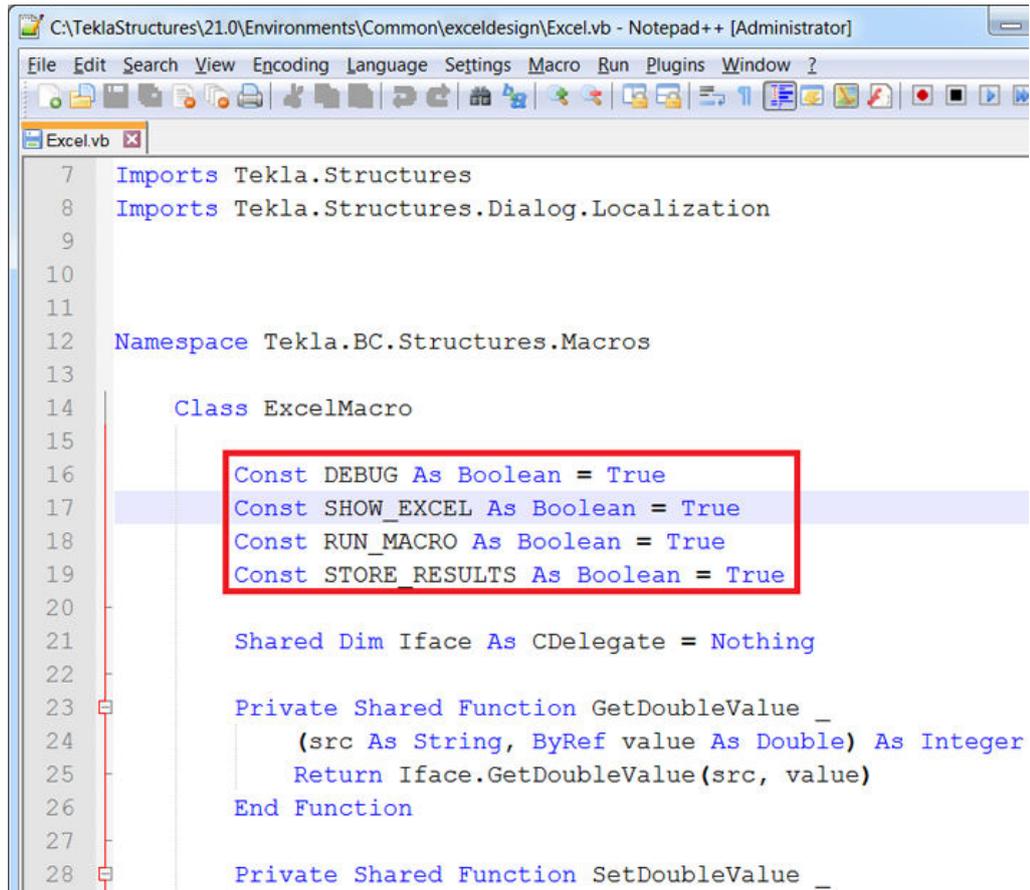
The following sheets are also included:

- **Data** shows catalog information.
- **Norm** shows the code selection and references to codes.
- **Language** shows translations.

Example of visualizing the Excel connection design process

You can define in the `Excel.vb` file how the Excel connection design process is visualized. The `Excel.vb` file links Tekla Structures with the external software and defines the Excel spreadsheet file names and the locations.

1. Open the `Excel.vb` file located in `..\Tekla Structures\<>version>\Environments\common\exceldesign`.
2. Configure the `Excel.vb` file as follows:



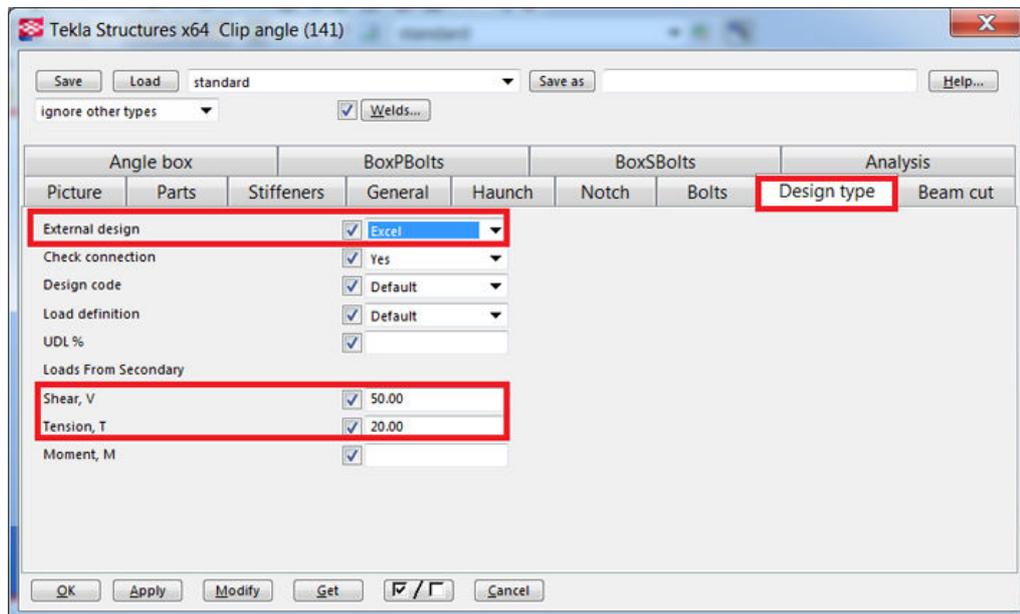
```
7 Imports Tekla.Structures
8 Imports Tekla.Structures.Dialog.Localization
9
10
11
12 Namespace Tekla.BC.Structures.Macros
13
14     Class ExcelMacro
15
16         Const DEBUG As Boolean = True
17         Const SHOW_EXCEL As Boolean = True
18         Const RUN_MACRO As Boolean = True
19         Const STORE_RESULTS As Boolean = True
20
21         Shared Dim Iface As CDelegate = Nothing
22
23         Private Shared Function GetDoubleValue _
24             (src As String, ByRef value As Double) As Integer
25             Return Iface.GetDoubleValue(src, value)
26         End Function
27
28         Private Shared Function SetDoubleValue _
```

- Visualizing control - `Const DEBUG As Boolean = True`
- Visualizing using Excel - `Const SHOW_EXCEL As Boolean = True`
- Storing the output - `Const STORE_RESULTS As Boolean = True`

3. Save the file.

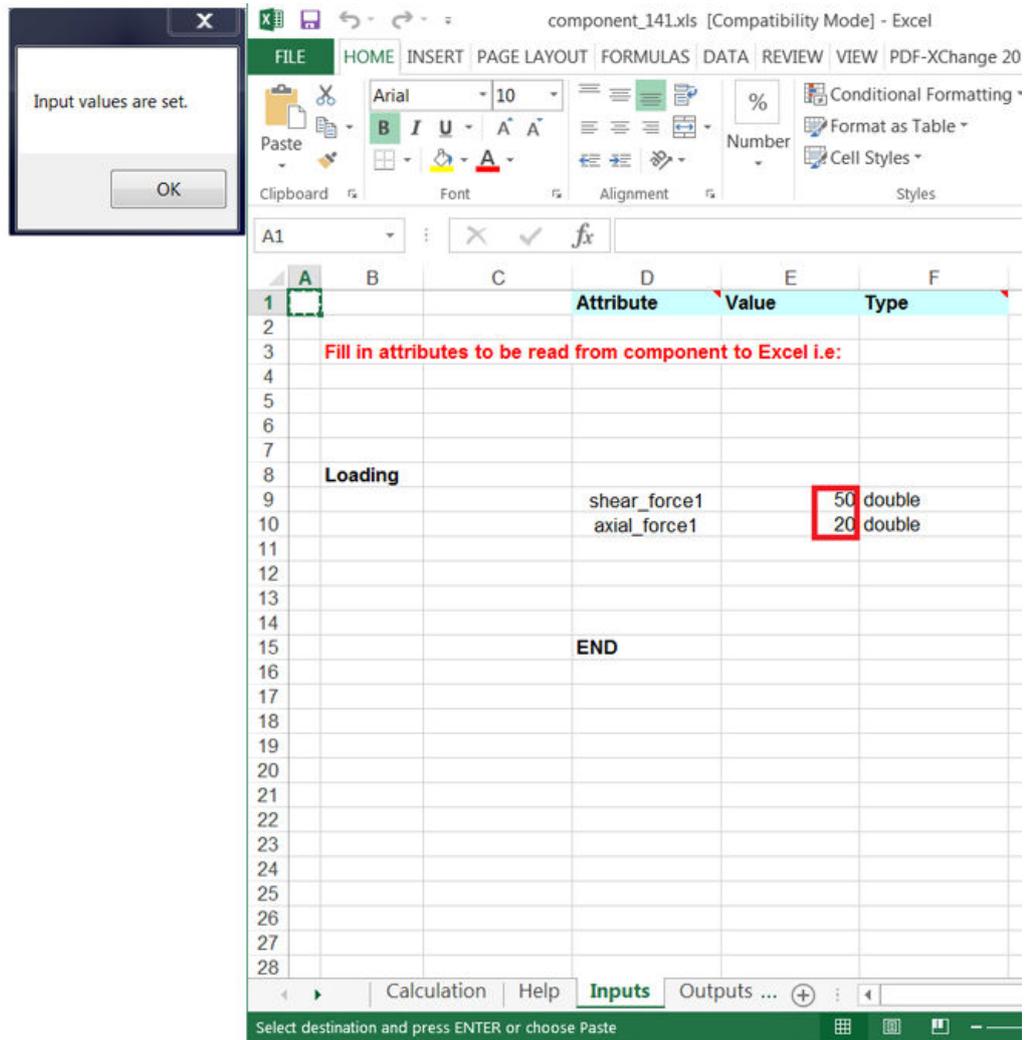
4. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
5. Search for **Clip angle (141)** and double-click it to open the properties dialog box.
6. On the **Design type** tab:

- a. Select **Excel** in the **External design** option.
- b. Enter the load values.



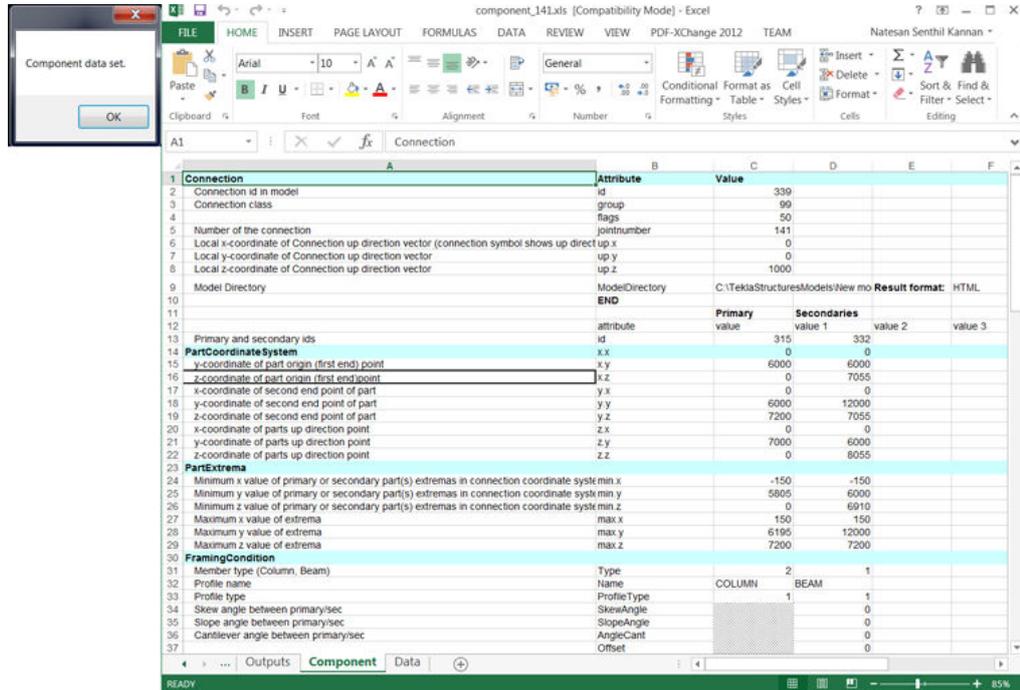
7. Click **Modify**.

The Excel design file opens and shows the **Inputs** sheet.



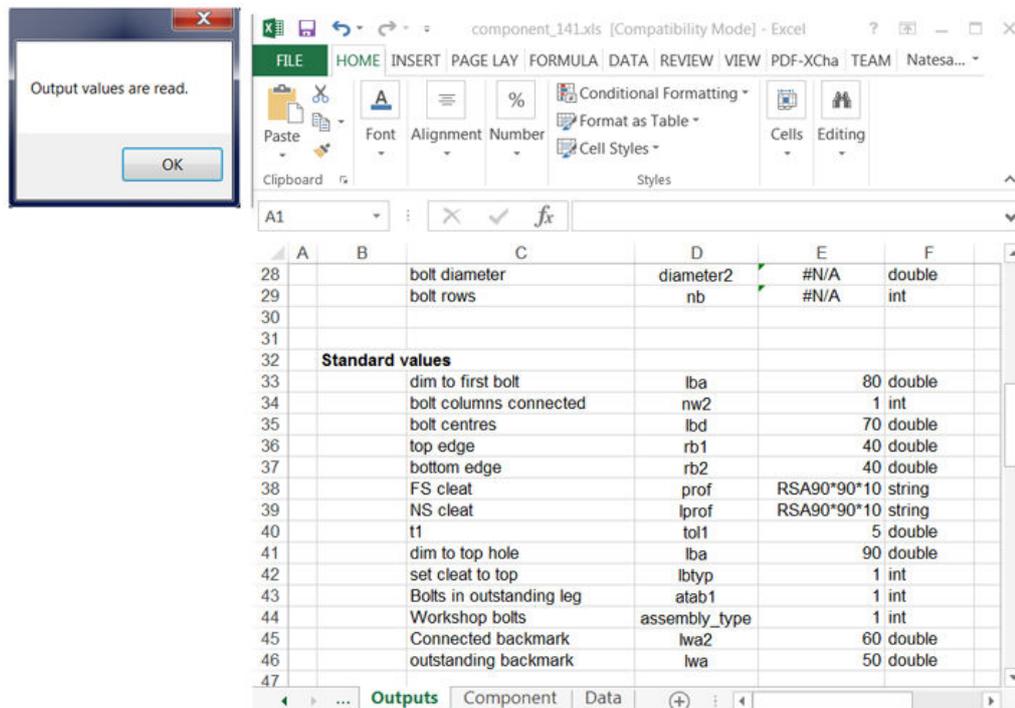
8. Click **OK** to continue.

The Excel design process now calculates the data that is shown on the **Component** sheet, and then opens the **Component** sheet.



9. Click **OK** to continue.

The Excel design process now calculates the result output values and then opens the **Outputs** sheet. The calculated results values are transferred to the connection.



10. Save the file in the model folder.
11. When you click **OK**, the design process is completed and the Excel design file is closed.

Showing connection status in Excel connection design

When you use Excel spreadsheets in connection design, you can have Tekla Structures use different colors in component symbols to show the status of a component in the model.

You can do this by including an error attribute on the **Outputs** sheet of the component's Excel spreadsheet. The type of the attribute is `int`.

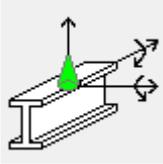
The possible values are:

Value	Color	Status
1	Green	Bolt edge distances are sufficient. The connection passes the connection design check using the UK and US design codes embedded in the system.
2	Yellow	Bolt edge distances are insufficient according to the value defined in the Components settings in File --> Settings --> Options .
3	Red	Tekla Structures cannot calculate the component properties. The possible reasons are: <ul style="list-style-type: none"> • The connection direction is not correct. • The work plane is not correct. • The selected connection is not appropriate for the situation. • The connection design check was carried out using the embedded UK and US design codes and the connection cannot support the loading you have defined.

NOTE The component symbol color can only be controlled for system components, not for custom components.

General tab

The **General** tab is available in steel connections and steel details.

Option	Description
<p>Up direction</p> 	<p>Rotates the connection around the secondary part or the detail around the main part.</p> <p>You can define the rotation angle around the x- and y-axis of the secondary part. The upper box is for the y-axis and the lower box for the x-axis.</p>
<p>Position in relation to primary part</p>	<p>Available only for details. The check boxes next to the images indicate the position of the definition point of the detail, relative to the main part.</p> <p>Horizontal offset and Vertical offset define the horizontal and vertical alignment of the detail, relative to the main part.</p>
<p>Locked</p>	<p>Prevents modifications.</p> <p>You can use the <code>privileges.inp</code> file to control the access to the Locked attribute.</p>
<p>Class</p>	<p>A number given to all parts the connection creates. You can use class to define the color (page 594) of the parts in the model.</p>
<p>Connection code</p>	<p>Identifies the connection. Tekla Structures can display this connection code in connection marks in drawings.</p>
<p>AutoDefaults rule group</p>	<p>Automatically sets connection properties according to the selected rule group. Rule group None switches AutoDefaults off.</p>
<p>AutoConnection rule group</p>	<p>Automatically switches the connection to another according to the selected rule group.</p>

See also

[AutoDefaults \(page 694\)](#)

[AutoConnection \(page 689\)](#)

Design and Design type tabs

Some component dialog boxes include a **Design** tab, others include a **Design type** tab. You can use the options on these tabs to check if the component will bear the uniform distributed load (UDL). Some **Design** tabs include only the design check. Tekla Structures saves the design summary as a `.txt` file in the model folder.

You can use AutoDefaults rule groups and Excel files in the design check:

- AutoDefaults rule groups automatically modify component properties to take the calculated load. To define which AutoDefaults rule group to use, go

to the **General** tab and select the rule in the **AutoDefaults rule group** list box.

For more information, see [Using reaction forces and UDLs in AutoDefaults and AutoConnection \(page 706\)](#).

- The information in an Excel file checks the connection design and automatically updates component properties to bear the UDL. This is useful when you want to check connection design according to other design codes. See [Excel spreadsheets in connection design \(page 720\)](#).

Design tab

This design check is intended to be used with imperial units.

To check the design:

1. Go to the **Design** tab and select **Yes** in the **Use UDL** list.
2. To use information in an Excel spreadsheet in the UDL calculation, select **Excel** in the **External design** list.
3. Enter the information you want to use in the calculation.
4. Select the connection in the model and click **Modify**.
Tekla Structures checks the component. A green component symbol indicates that the connection will bear the UDL, red indicates that it will not.
5. To view the results of the check, right-click the component symbol and select **Inquire** from the pop-up menu.

The **Inquire object** dialog box shows the summary of the design check and related information.

See also [Excel spreadsheets in connection design \(page 720\)](#).

Design type tab

This design check is intended to be used with imperial units.

To check the design:

1. Go to the **Design type** tab and select **Yes** in the **Check connection** list.
Tekla Structures checks the connection each time it is used or changed in the model.
2. Enter the information you want to use in the calculation.
3. Select the connection in the model and click **Modify**.
Tekla Structures checks the component. A green component symbol indicates that the connection will bear the UDL, red indicates that it will not.

4. To view the results of the check, right-click the component symbol and select **Inquire** from the pop-up menu.

The **Inquire object** dialog box shows the summary of the design check: the part checked, the name of the check, the applied and allowed force and how much capacity has been used, the results and possible solutions.

Design tab for check design only

The design is based on the British standard BS5950.

The design has the following limitations:

- Design only works in the UK environment.
- Design is available only if the main part and the secondary parts are perpendicular.
- Design is available only with two bolts horizontally.
- Design is available only when vertical bolts are defined from the top.
- Design is valid for I profiles only.

To check the design:

1. Go to the **Design** tab and select **On** in the **Design** list.
2. Enter the **Tie Force** in kilo Newtons (kN).

Tie force is required if the design check is turned on and the framing type of the connection is beam-to-column. If there is no tie force, enter 0.

3. Enter the **Shear Force** in kN.

If the design check is turned on, enter a positive value. If there is no shear force, enter 0.

4. Select the connection in the model and click **Modify**.

The connection symbol shows the design check status:

- Green means that the design check was successful.
- Yellow means that a warning occurred in the design check.
- Red means that a fatal error occurred in the design check.

5. To view the results of the check, right-click the connection symbol and select **Inquire** from the pop-up menu.

The **Inquire object** dialog box shows the summary of the design check and related information.

NOTE If the message **Numbering not up to date** is shown in the **Inquire object** dialog box, the marks will not be correct. You need to renumber the model to ensure that the marks are up-to-date. After that use the **Inquire** command again to get the correct marks to the design check summary.

Analysis tab

Use the **Analysis** tab in the steel connection or detail dialog box to define how Tekla Structures handles connections and details in the analysis.

The screenshot shows the 'Analysis' tab in a dialog box. It contains the following settings:

- Use analysis restraints:** Yes
- Member selection:** Primary
- Restrain combination:** [Pencil icon]
- Support condition:** Connected
- Restraints table:**

Restraint	Value	Offset
Ux	Free	0.00
Uy	Free	0.00
Uz	Free	0.00
Rx	Pinned	0.00
Ry	Pinned	0.00
Rz	Pinned	0.00
- Longitudinal member offset:** 0.00
- Analysis profile:** [empty]
- Analysis profile length:** 0.00

Option	Description
Use analysis restraints	Set to Yes to use the analysis properties of the connection or detail in the analysis instead of the analysis properties of the parts in the connection. You also need to set Member end release method by connection to Yes in the Analysis Model Properties dialog box when you create the analysis model. For more information, see Analysis model properties.
Member selection	Use to associate the analysis properties with each connection part (Primary , 1. secondary , 2. secondary , and so on).
Restraint combination	For more information, see Defining support conditions.
Support condition	

Option	Description
Longitudinal member offset	For more information, see Analysis part properties.
Analysis profile	Tekla Structures uses this profile in the analysis instead of the one in the physical model to take the stiffness of the connection or detail into account.
Analysis profile length	In the analysis, Tekla Structures overrides the profile of the part in the physical model for this length.

8 Custom components

You can create customized connections, parts, seams, and details for your project. These are called *custom components*. You can use custom components in the same way as any Tekla Structures system component. You can even create intelligent custom components that automatically adjust to changes in the model.

When to use

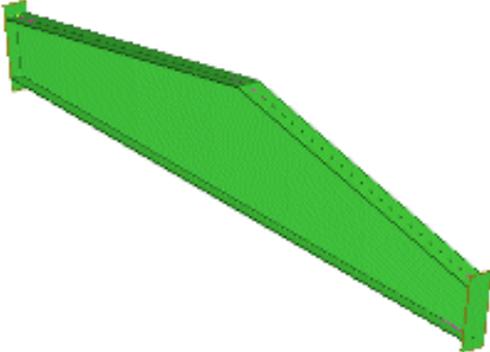
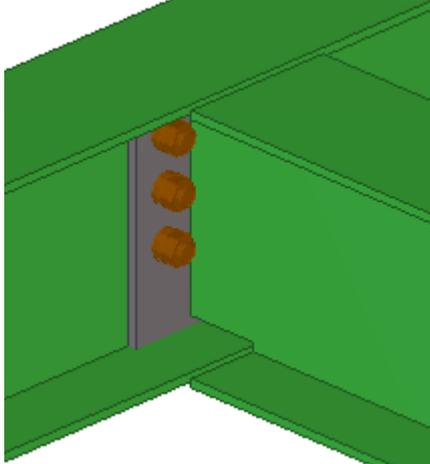
Create a custom component if you cannot find a predefined [system component \(page 674\)](#) that meets all your needs. Especially if you need to create a large number of complex model objects and copy them across several projects.

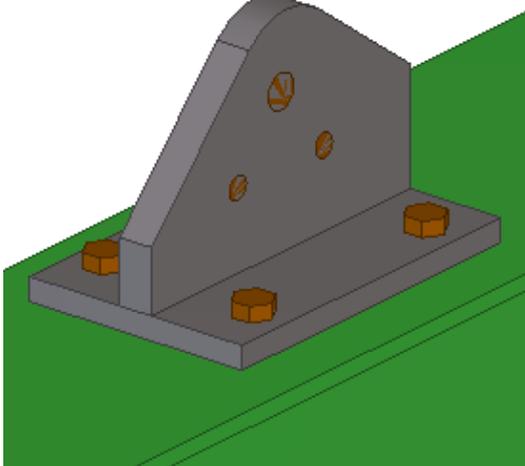
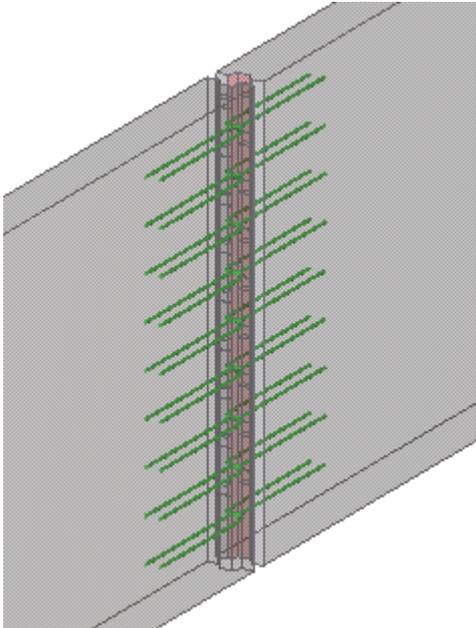
Benefits

Once you store a custom component in the **Applications & components** catalog, you can easily access it from the catalog and use it in another location in the same model. If you must modify the custom component, you only need to make the changes once. When you save the changes, they will be automatically applied to all copies of that custom component in the model. You can also import and export custom components as `.uel` files between models and share the custom components with your colleagues.

Custom component types

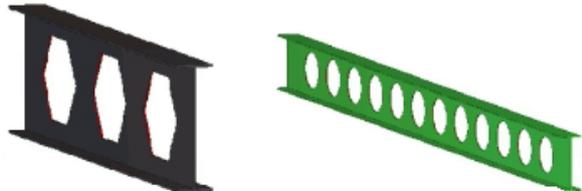
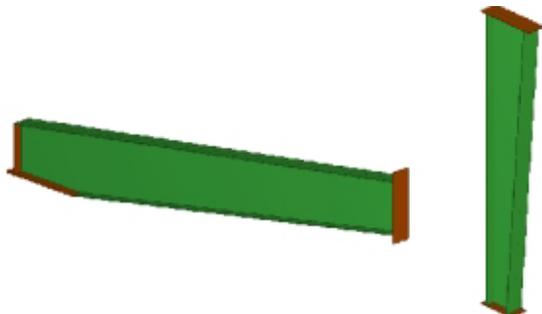
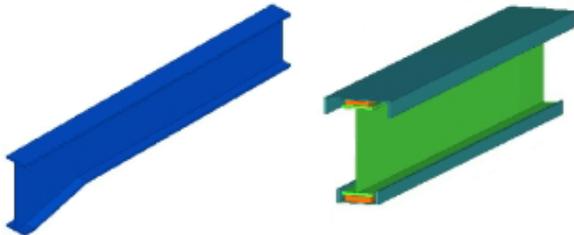
You can create four types of custom components:

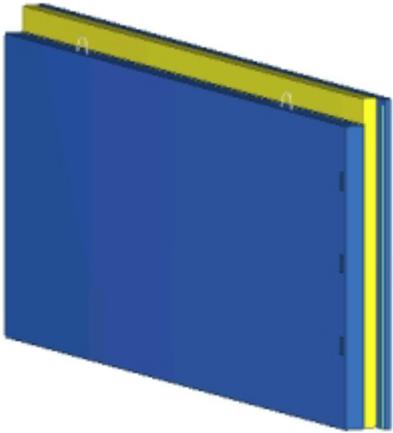
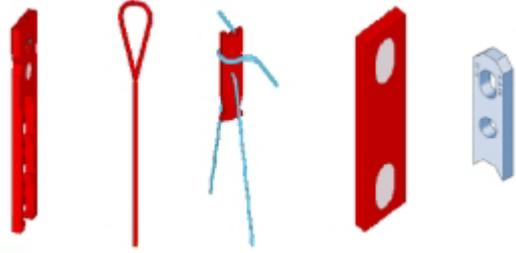
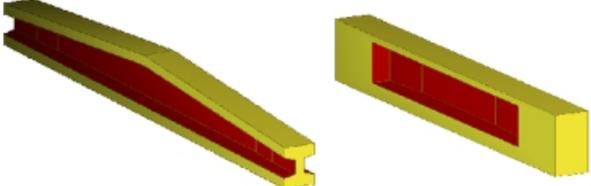
Type	Description	Example
Custom part (page 738)	<p>Creates a group of objects that may contain connections and details.</p> <p>Note: Unlike other custom components, custom parts are not marked with a component symbol  in the model. Custom parts have the same position properties as beams have.</p>	
Custom connection (page 740)	<p>Creates connection objects and connects the secondary parts to the main part. The main part may be continuous at the connection point.</p>	

Type	Description	Example
Custom detail (page 742)	Creates detail objects and connects them to a single part at the location you picked.	 <p>A 3D perspective view of a grey, L-shaped base plate with a curved top edge, resting on a green surface. Several orange, hexagonal fasteners are attached to the base plate, illustrating a custom detail.</p>
Custom seam (page 744)	Creates seam objects and connects the parts along a line that you create by picking with two points. The parts are usually parallel.	 <p>A 3D perspective view of a grey, L-shaped base plate with a curved top edge, resting on a green surface. A vertical seam is visible, showing the connection between two parallel parts. The seam is highlighted with a red vertical line and green diagonal lines, illustrating a custom seam.</p>

8.1 Custom parts

Custom parts may consist of a single part or a group of parts, and they often have a complex composition. The following images show some examples of custom parts:

Steel	Company standard bracing plates	
	Castellated beam and cell beam	
	Built-up beams/columns	
	Built-up beams	
	Standard glazing fixings	

Precast concrete	Sandwich panel	
	Lifters	
	Standard embeds/inserts	
	Standard beams	

See also

[Custom connections \(page 740\)](#)

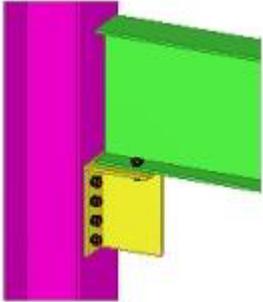
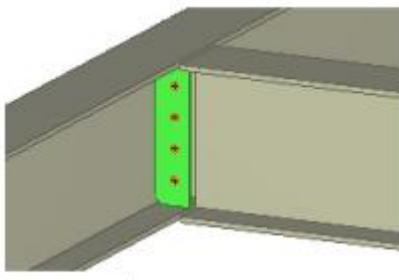
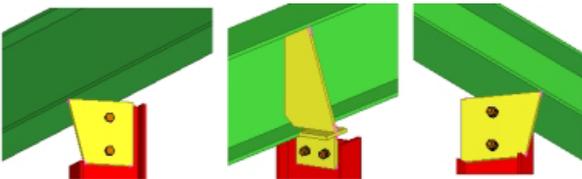
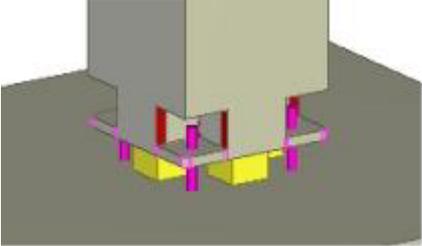
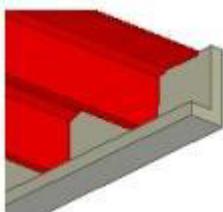
[Custom details \(page 742\)](#)

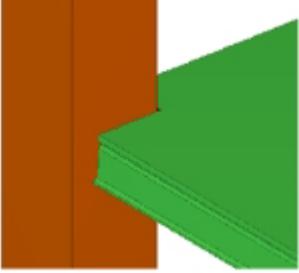
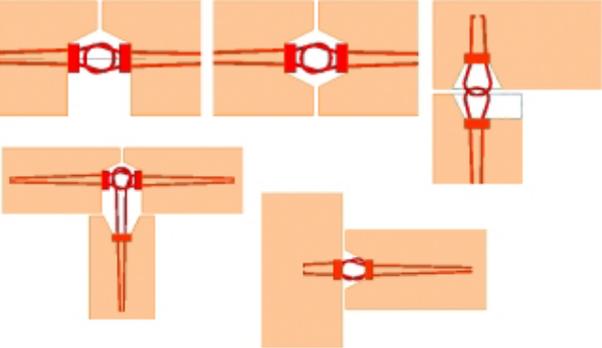
[Custom seams \(page 744\)](#)

8.2 Custom connections

Custom connections can be used to connect a main part to up to 30 secondary parts. The connection is made between the main part and the ends of the

secondary parts. The following images show some examples of custom connections:

Steel	Built-up plate seat	
	Shear plate	
	Typical japanese post connections	
Precast concrete	Base detail	
	Double tee to L profile	

Column cut-out	
Wall panel connections	

See also

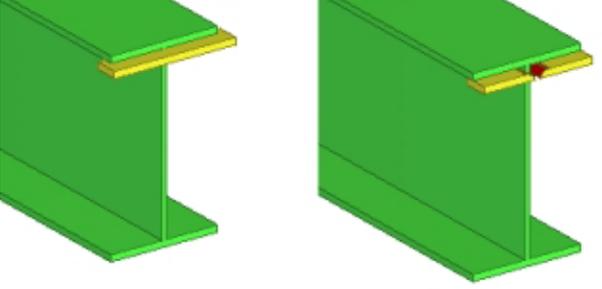
[Custom parts \(page 738\)](#)

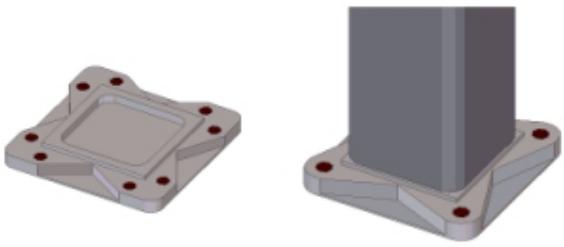
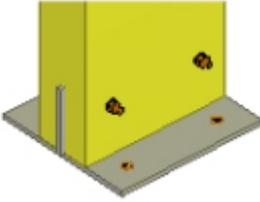
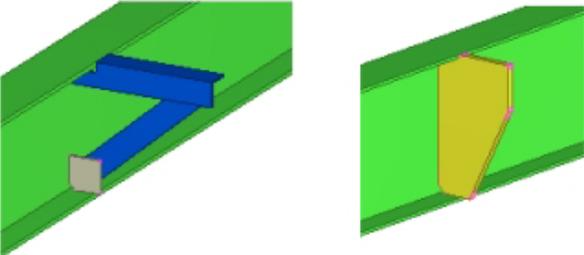
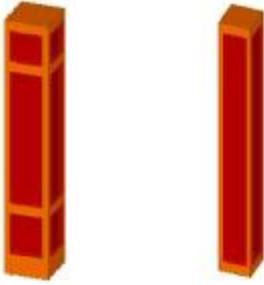
[Custom details \(page 742\)](#)

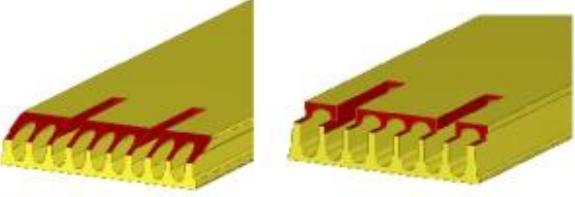
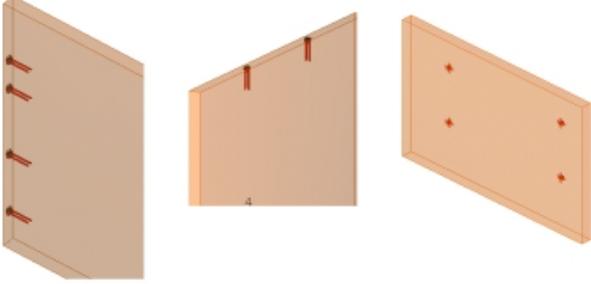
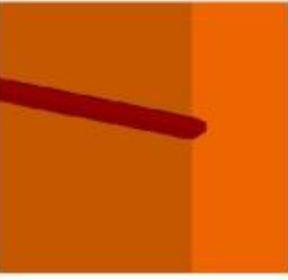
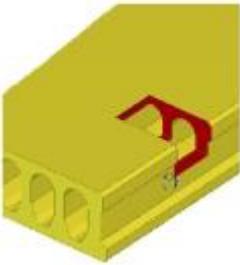
[Custom seams \(page 744\)](#)

8.3 Custom details

Custom details can be used to add more information to a single part, such as extra plates or cut-outs. The following images show some examples of custom details:

Steel	Backing plates	
-------	----------------	--

	Cast base	
	Timber base	
	Out rigger (stiffeners) and out rigger plate (stiffeners)	
Precast concrete	Door and window	
	Column patterns	

Hollow core end details	
Lifting details	
Mock joint/reveal	
Side pocket	

See also

[Custom parts \(page 738\)](#)

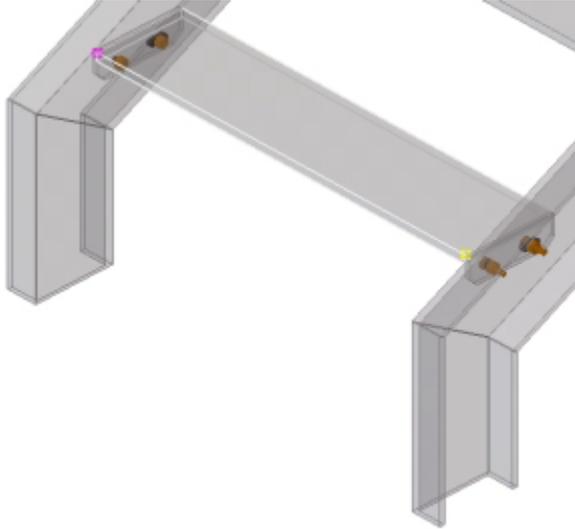
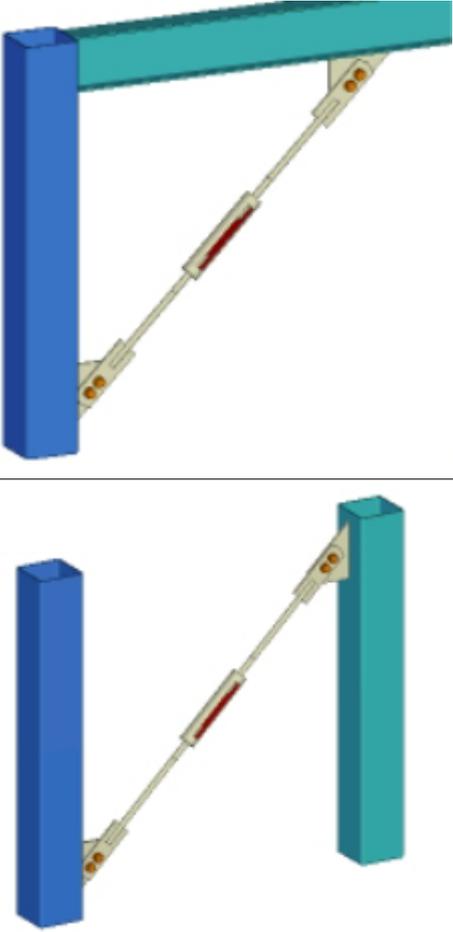
[Custom connections \(page 740\)](#)

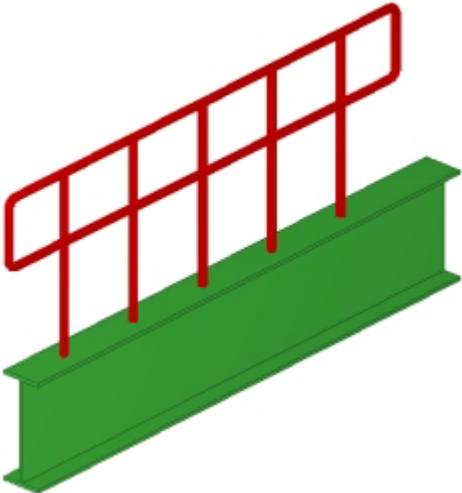
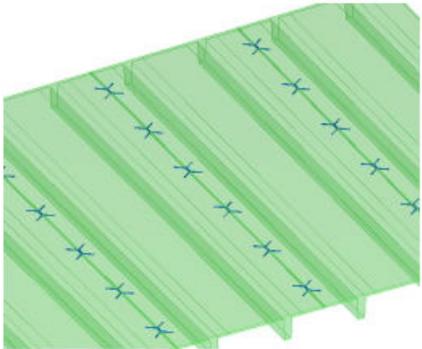
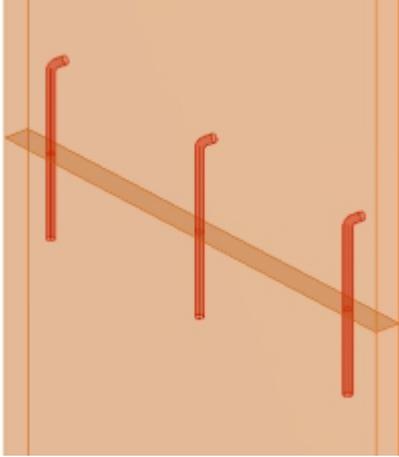
[Custom seams \(page 744\)](#)

8.4 Custom seams

Custom seams can be used to connect a main part to up to 30 secondary parts. They can also be used on one main part only. The seam is made along

the length of the part. The following images show some examples of custom seams:

Steel	Steel stair step	 A 3D CAD model of a steel stair step assembly. It consists of three main parts: a vertical riser on the left, a horizontal tread in the middle, and another vertical riser on the right. The parts are connected at their ends with custom seams, indicated by small colored dots (purple and yellow) at the joint locations. The model is shown in a perspective view.
Turnbuckles		 Two 3D CAD models of turnbuckle assemblies. The top model shows a turnbuckle connecting a horizontal teal beam to a vertical blue post. The bottom model shows a turnbuckle connecting two vertical posts, one blue and one teal. Both models show the turnbuckle in a perspective view, with the central rod and end fittings clearly visible.

	Handrail	
Precast concrete	Double tee connection	
	Panel to panel grout tube connection	

See also

[Custom parts \(page 738\)](#)

[Custom connections \(page 740\)](#)

[Custom details \(page 742\)](#)

8.5 Create a custom component

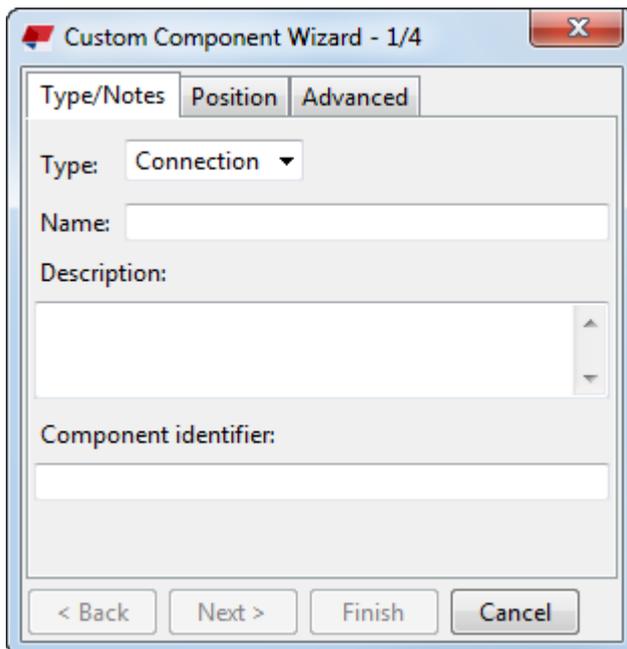
You can create customized components that have all the details you need. Start by creating a simple custom component which you can modify later. In the following example, we will create a simple custom connection.

1. In the model, create a sample component that contains all the necessary component objects, such as parts, cuts, fittings, and bolts.

To do this quickly, [explode and modify \(page 750\)](#) a similar existing component.

2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Click the **Access advanced features** button  and select **Define custom component**.

The **Custom Component Wizard** dialog box opens.



4. In the **Type** list, select the [component type \(page 736\)](#): connection, detail, seam, or part.
5. In the **Name** box, enter a unique name for the component.
6. Modify the other [properties \(page 864\)](#) on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next**.
7. In the model, select the objects you want to include in the custom component.



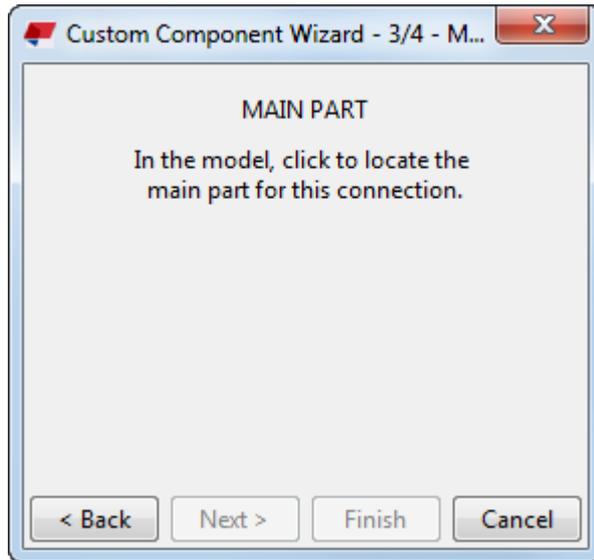
You can use area selection to [select multiple objects simultaneously \(page 116\)](#). The main and secondary parts and the grids are ignored when you select objects for the custom component.

NOTE If you cannot select desired objects in the model, check the [selection switches \(page 122\)](#) and the [selection filter settings \(page 158\)](#).

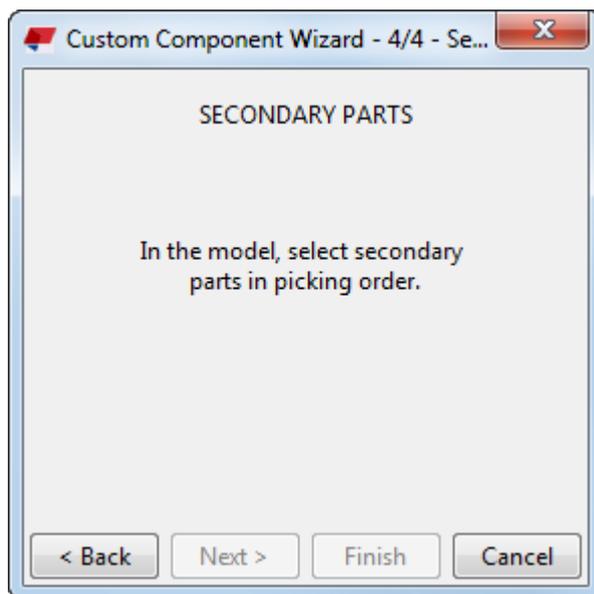
If you want to include [rebar set modifiers \(page 486\)](#) in the custom component:

- Ensure that **Direct modification** is switched off when you select the modifiers.
- Hold down **Shift** and select the modifiers one by one. Area selection does not select modifiers.

-
8. Click **Next**.
 9. Select the main part for the component.



10. Click **Next**.
11. Select the secondary parts for the component.



To select multiple secondary parts, hold down **Shift** when you select them. The maximum number of secondary parts in a custom component is 30.

NOTE Pay attention to the order in which you select secondary parts. Tekla Structures will use the same picking order when you use the custom component in a model.

12. Define any other properties required for this custom component, such as detail or seam position. The properties depend on the component type that you selected in step 4.

13. If you want to adjust any of the settings at this stage, click **Back** to return to the previous page of the **Custom Component Wizard**.

14. When you are happy with the settings, click **Finish** to create the custom component.

The custom component is added in the model and in the **Applications & components** catalog.

15. If you want to change these settings afterwards:

a. On the [custom component editor toolbar \(page 761\)](#), click the

Modify custom component settings button .

b. Change the settings.

c. Click **OK**.

See also

[Create a nested custom component \(page 750\)](#)

[Example: Create a custom end plate component \(page 754\)](#)

[Tips for creating custom components \(page 881\)](#)

Explode a component

Exploding means that you ungroup the objects of an existing component. This can be useful when you want to create custom components more quickly. Once the objects are detached, you can modify them to suit your needs and then create new custom components using these objects.

1. Select the component you want to explode.

2. Right-click and select **Explode Component**.

Tekla Structures separates the component objects. You can modify the objects and use them to [create new custom components \(page 747\)](#).

Create a nested custom component

You can create more complex custom components by joining two or more components together as a nested component. The original components become sub-components in the nested component.

1. In the model, create the components and other model objects that you want to include in the nested component.

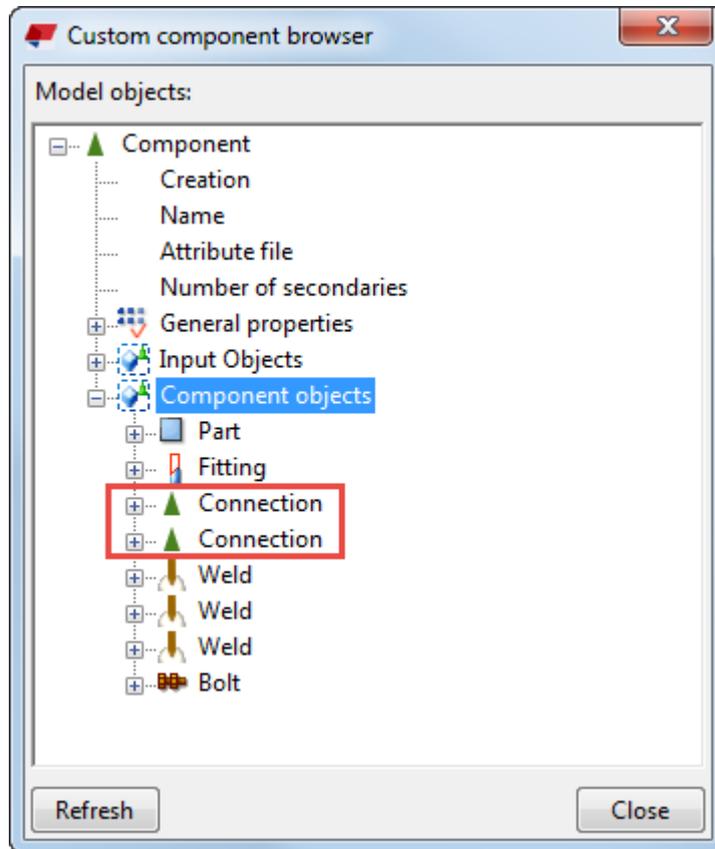
2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.

3. Click the **Access advanced features** button  and select **Define custom component**.

The **Custom Component Wizard** dialog box opens.

4. In the **Type** list, select the type of the nested custom component.
5. In the **Name** box, enter a unique name for the nested component.
6. Modify the other properties on the **Type/Notes** tab, **Position** tab, and **Advanced** tab, and then click **Next**.
7. Select the components and any other objects you want to include in the nested component, and then click **Next**.
8. Follow the instructions in the **Custom Component Wizard** to continue.
You will be asked to select the main and secondary parts for the nested component. Depending on the component type that you selected in step 3, you may also have to define other properties, such as detail or seam position.
9. When you are happy with the settings, click **Finish** to create the nested component.

The component is added in the model and in the **Applications & components** catalog. The sub-components are shown in the (page 761), together with the other component objects:



10. If you want to change the settings afterwards:
 - a. In the custom component editor (page 761), click the **Modify custom component settings** button .
 - b. Change the settings.
 - c. Click **OK**.

WARNING If you use a component of the type plug-in as a sub-component of a nested component, and change the sub-component's properties in the custom component editor, note that those changes may be lost when you save the nested component and use it in a model.

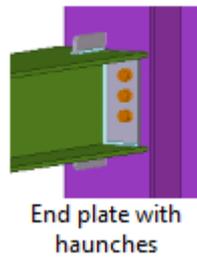
To prevent losing any properties, link a variable to each plug-in property that you want to keep. You can also use component attribute files to do this. For more information, see [Examples of parametric variables and variable formulas \(page 797\)](#).

Create a thumbnail image of a custom component

Create a thumbnail image for each custom component to make it easier to find a suitable component when modeling.

1. In the model, select the custom component.
2. Right-click and select **Edit Custom Component**.
3. Adjust the view and hide unnecessary objects to have a clear view of the custom component.
4. Take a screenshot of the custom component.
 - a. On the **View** tab, click **Screenshot** --> **Screenshot** to open the **Screenshot** dialog box.
 - b. Click **Pick view** and select the view you want to take a screenshot of.
 - c. Click **Options** to open the **Screenshot Options** dialog box.
 - d. Select **Print to file**.
 - e. Select **White background** and click **OK**.
 - f. In the **Screenshot** dialog box, click **Capture**.
 - g. Click **Close** to close the dialog box.
 - h. On the **File** menu, click **Open the model folder**.
 - i. Browse to the `\screenshots` folder under the model folder.
 - j. Open the screenshot file in a graphics editor.
 - k. Crop the image, if needed.
5. Save the thumbnail image in the `\screenshots` folder under the model folder.
6. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
7. Right-click the custom component and select **Thumbnails**.
The **Thumbnails** dialog box opens.
8. Click **Add thumbnail**.
9. Browse to the `\screenshots` folder under the model folder.
10. Select the thumbnail image and click **Open**.
11. In the **Thumbnails** dialog box, select the check box next to the image you want to use and clear the other check boxes.
12. Click **Close**.

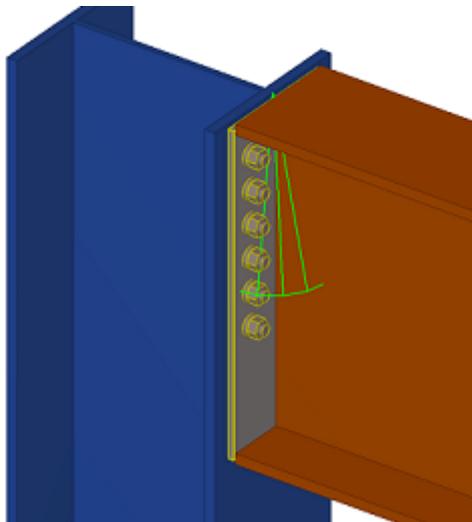
Tekla Structures shows the thumbnail image in the **Applications & components** catalog:



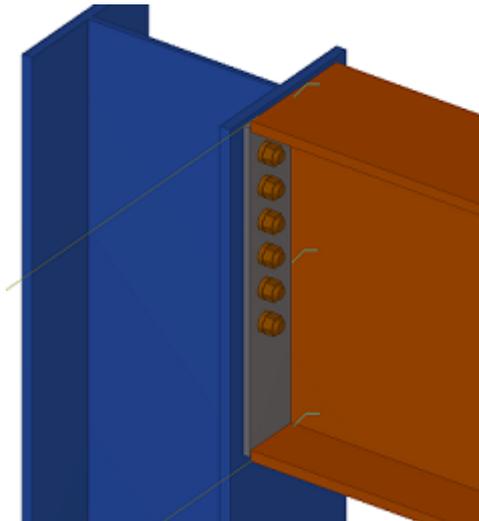
Example: Create a custom end plate component

In this example, you will create a simple custom component based on an existing end plate component.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the **Access advanced features** button  and select **Explode component**.
3. Select the end plate component in the model.



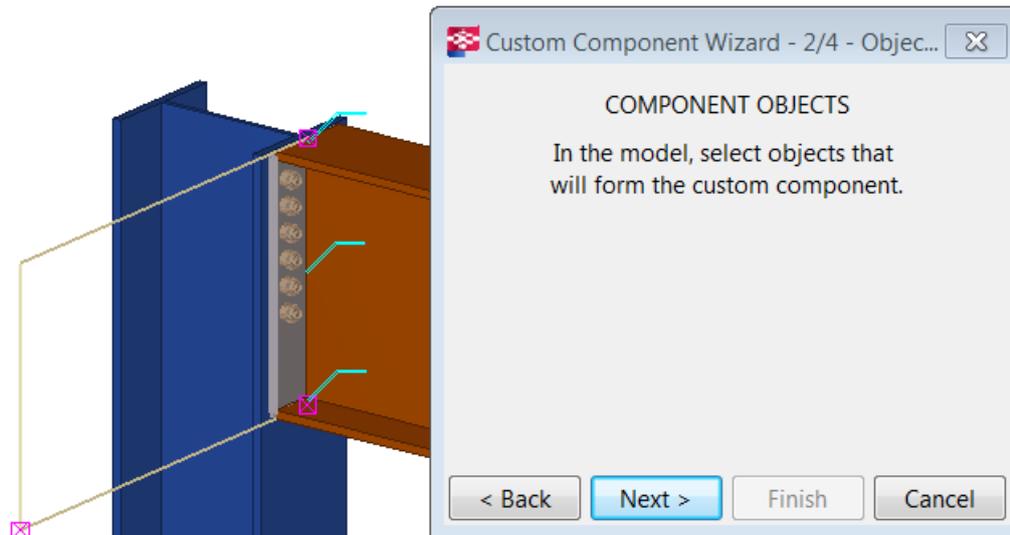
Tekla Structures separates the objects in the component.



4. Click the **Access advanced features** button  and select **Define custom component**.
5. In the **Type** list, select **Connection**.
6. In the **Name** box, enter a unique name for the custom component.

A screenshot of the 'Custom Component Wizard - 1/4' dialog box. The dialog has three tabs: 'Type/Notes', 'Position', and 'Advanced'. The 'Type/Notes' tab is active. It contains a 'Type:' dropdown menu with 'Connection' selected, a 'Name:' text box containing 'End Plate', a 'Description:' text area, and a 'Component identifier:' text box. At the bottom, there are four buttons: '< Back', 'Next >', 'Finish', and 'Cancel'. The 'Type' and 'Name' fields are highlighted with red boxes in the original image.

7. Click **Next**.
8. Select the objects you want to use in the custom component, and then click **Next**.



You can use area selection (left to right) to select the objects. Tekla Structures ignores the main part and secondary parts and the grids when you are selecting objects to include in the custom component.

9. Select the column as the main part, and then click **Next**.

The main part supports the secondary part.

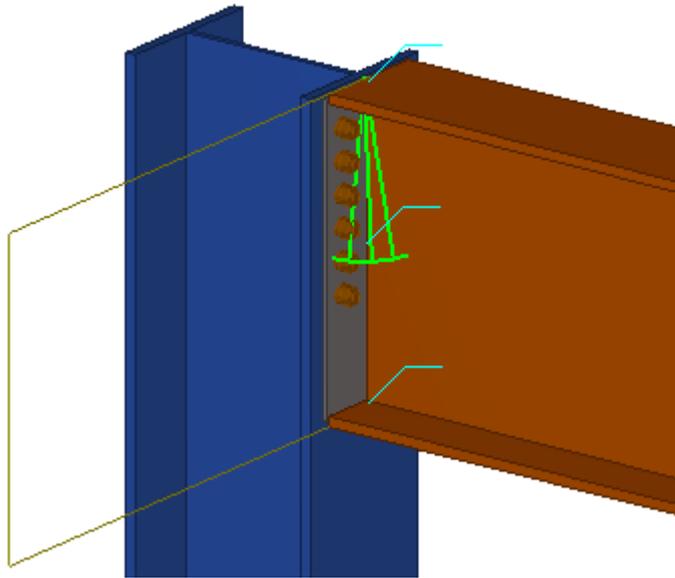
10. Select the beam as the secondary part.

The secondary part is supported by the main part.

NOTE When you select multiple secondary parts, pay attention to the order of selection. The custom component will use the same selection order when you add the component in a model. The maximum number of secondary parts in a custom component is 30.

11. Click **Finish**.

Tekla Structures displays a component symbol for the new component.

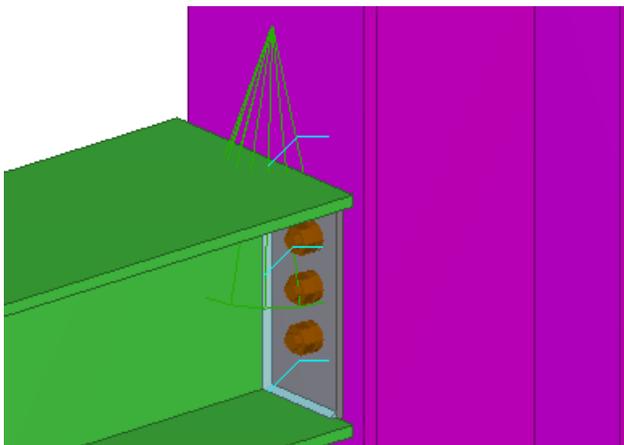


You have now defined a simple custom component, which you can use in locations similar to where it was originally created. This component is not intelligent and Tekla Structures does not adjust dimensions to suit any changes in the model. To make the custom component intelligent, you need to [modify \(page 761\)](#) it in the custom component editor.

Example: Create a nested connection with stiffeners

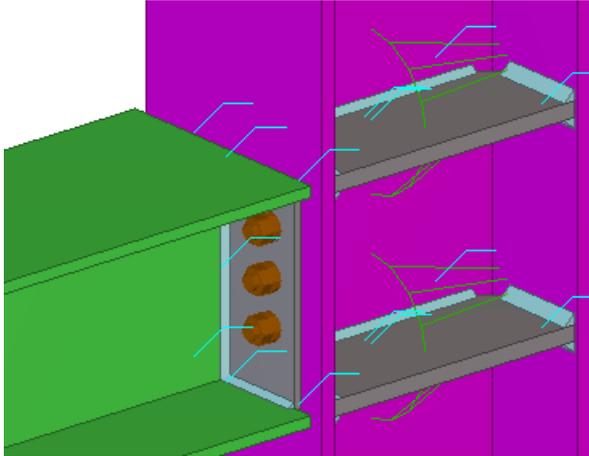
In this example, you will create a nested custom connection that consists of an end plate, a bolt group, welds, and two **Stiffeners (1003)** components. The stiffeners are optional, which means that you can choose whether or not to create them when using the component in a model.

1. Add an **End plate (144)** component.



2. [Explode \(page 750\)](#) the end plate component.

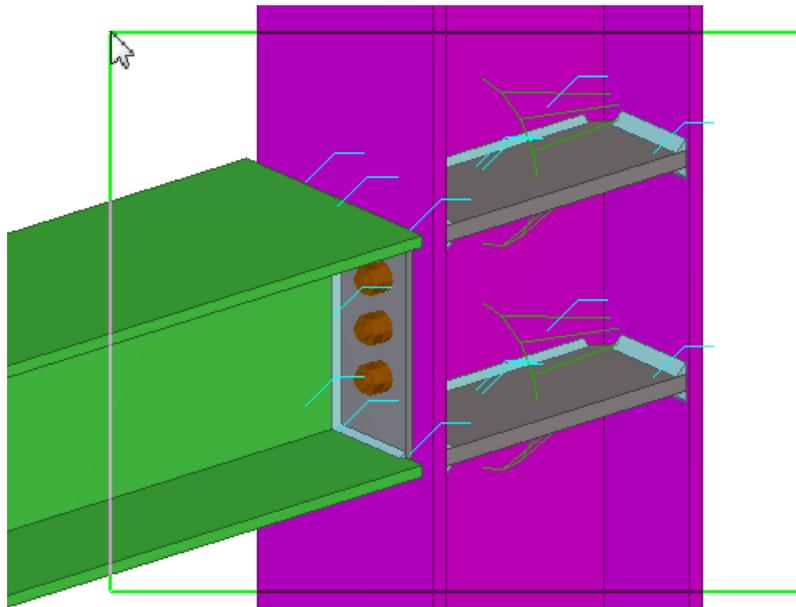
3. Add two **Stiffeners (1003)** components.



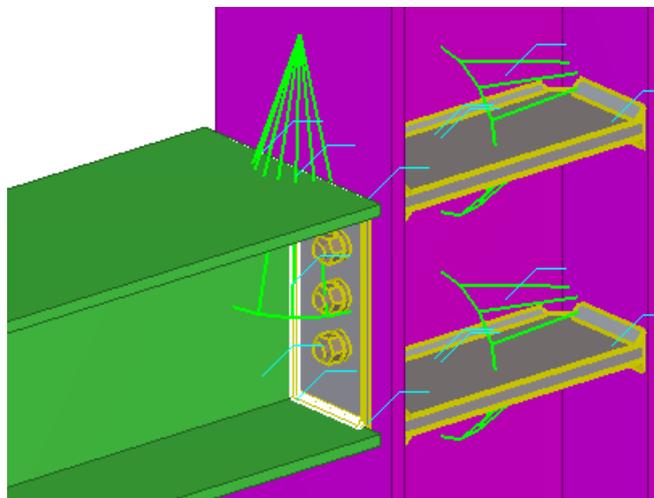
4. Create a nested custom component that contains the stiffeners and the end plate objects.

- a. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
- b. Click the **Access advanced features** button  and select **Define custom component**.
- c. In the **Type** list, select **Connection**.
- d. In the **Name** box, enter `End plate with stiffeners`.
- e. Click **Next**.
- f. Make an area selection (from right to left) to include the following objects in the nested component: the column, the beam, the stiffener

components, and all the end plate objects.



- g. Click **Next**.
- h. Choose the column as the main part of the nested component, and then click **Next**.
- i. Choose the beam as the secondary part of the nested component, and then click **Finish**. Tekla Structures creates the nested component.

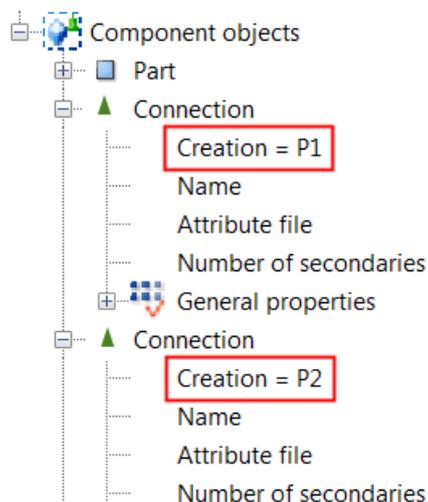


5. Select the nested component you just created.
6. Right-click and select **Edit Custom Component**.
7. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.

8. Create the following parametric variables:
 - a. Click **Add** to create a new parametric variable P1.
 - b. In the **Value type** list, select **Yes/No**.
 - c. In the **Label in dialog box** box, enter `Create Stiffener 1`.
 - d. Click **Add** to create a new parametric variable P2.
 - e. In the **Value type** list, select **Yes/No**.
 - f. In the **Label in dialog box** box, enter `Create Stiffener 2`.

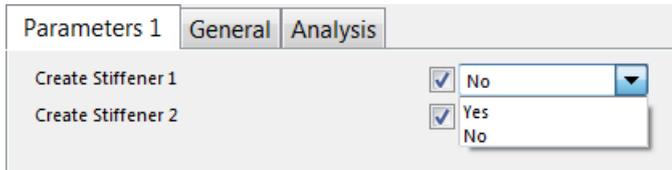
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create Stiffener 1
P2	0	0	Yes/No	Parameter	Show	Create Stiffener 2

9. Link the variables to the **Creation** property of the two stiffeners:
 - a. In the **Custom component browser**, browse for the uppermost **Connection**.
 - b. Right-click **Creation** and select **Add Equation**.
 - c. Enter `P1` after the equal sign, and then press **Enter**.
 - d. Browse for the second **Connection**.
 - e. Right-click **Creation** and select **Add Equation**.
 - f. Enter `P2` after the equal sign, and then press **Enter**.



10. [Save and close \(page 823\)](#) the nested component.

You now have the following options in the nested component's dialog box:



8.6 Modify a custom component

Use the custom component editor to fine-tune existing custom components. When you modify a custom component, Tekla Structures updates all instances of that component throughout the model with the changes you have made.

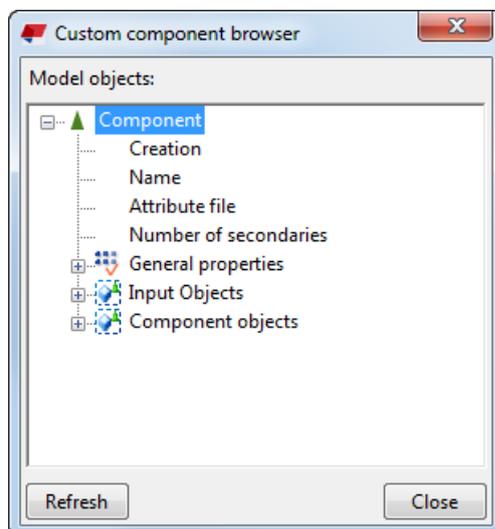
1. In the model, select the custom component by clicking the green component symbol.

NOTE Custom parts do not have a component symbol in the model. To select custom parts, ensure that the **Select components**  selection switch is active.

2. Right-click and select **Edit Custom Component**.

The custom component editor opens. It consists of the following parts:

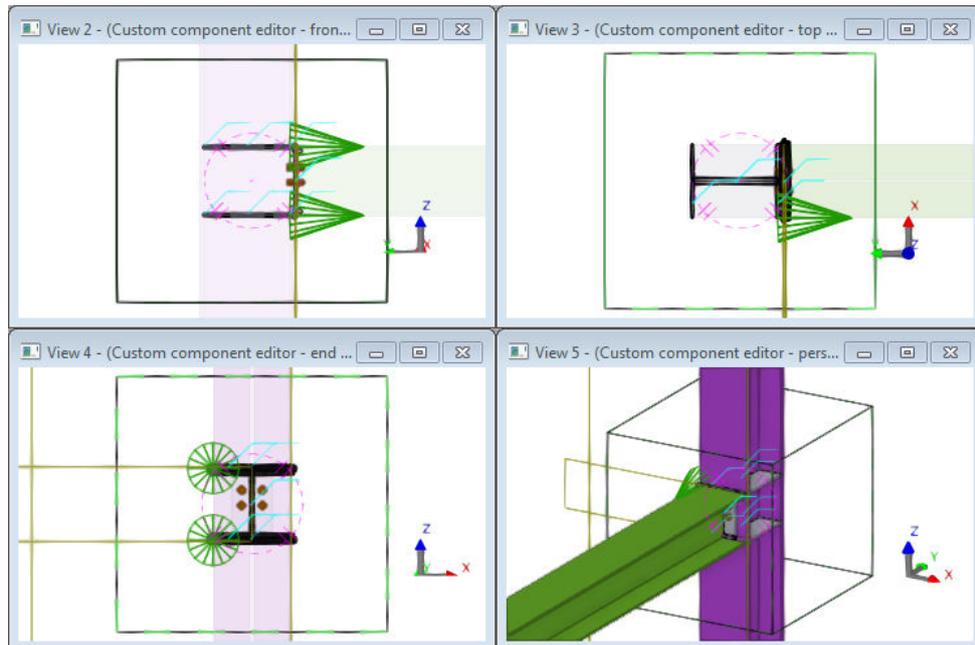
- The **Custom component browser**



- The **Custom component editor** toolbar



- Four different **views** of the custom component



3. Modify the custom component in one of the four custom component views. You can, for example:
 - Add or remove component objects
For example, add extra bolts or stiffeners to the component. Only component objects, not the main or secondary parts, can be modified in the custom component editor.
 - [Bind component objects to a plane \(page 764\)](#)
 - [Add a distance between component objects \(page 774\)](#)
 - [Set object properties using parametric variables \(page 777\)](#)
4. [Save the custom component \(page 823\)](#). Click **Yes** when prompted to replace all occurrences of the custom component in the model. All instances of the custom component are now updated with the changes you made.

See also

[Protect a custom component with a password \(page 762\)](#)

Protect a custom component with a password

You can set a password to prevent others from modifying a custom component. Password-protected custom components can still be added to models as usual.

1. In the model, select a custom component.

2. Right-click the custom component symbol and select **Edit Custom Component**.
3. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
4. Click **Add** to create a new variable.
5. In the **Name** box, enter `Password`.
6. In the **Formula** box, enter the desired password.
7. [Save the custom component. \(page 823\)](#)

The next time someone tries to edit this custom component, they will be asked for the password.

8.7 Add variables to a custom component

Variables are the properties of a custom component. You can create variables in the custom component editor and use them to adapt custom components to changes in the model. Some of the variables appear in the custom component's dialog box, while others are hidden and are only used in calculations.

Variable types

There are two types of variables:

- **Distance variable:** The distance between two planes, or between a point and a plane. A distance variable binds parts together, or works as a reference distance.
- **Parametric variable:** Controls all the other properties in a custom component, such as name, material grade, and bolt size. Parametric variables are also used in calculations.

Distance variables

Use distance variables to bind custom component objects to a plane, so that the component objects stay at a fixed distance even if the surrounding objects change. You can create distance variables manually or automatically.

You can bind the following objects to a plane:

- construction planes
- reference points of parts (only custom component objects)
- reference points of bolt groups
- chamfers
- part and polygon cut handles

- line cuts
- reference points of reinforcing bars
- reference points of reinforcement meshes and strands
- fittings

You can decide which distance variables are shown in the custom component's dialog box. Show the variables if you want to edit their values in the dialog box. Hide the variables if you only use them to bind objects to a plane.

Parametric variables

Use parametric variables to [to set properties for any object the custom component creates \(page 777\)](#). After creating the variable, you will be able to change the value directly in the custom component's dialog box.

You can also create formulas to calculate values. For example, you can calculate the position of a stiffener relative to the beam length.

You can decide which parametric variables are shown in the custom component's dialog box. Show the variables if you want to edit their values in the dialog box. Hide the variables if you only use them in calculations.

Bind component objects to a plane

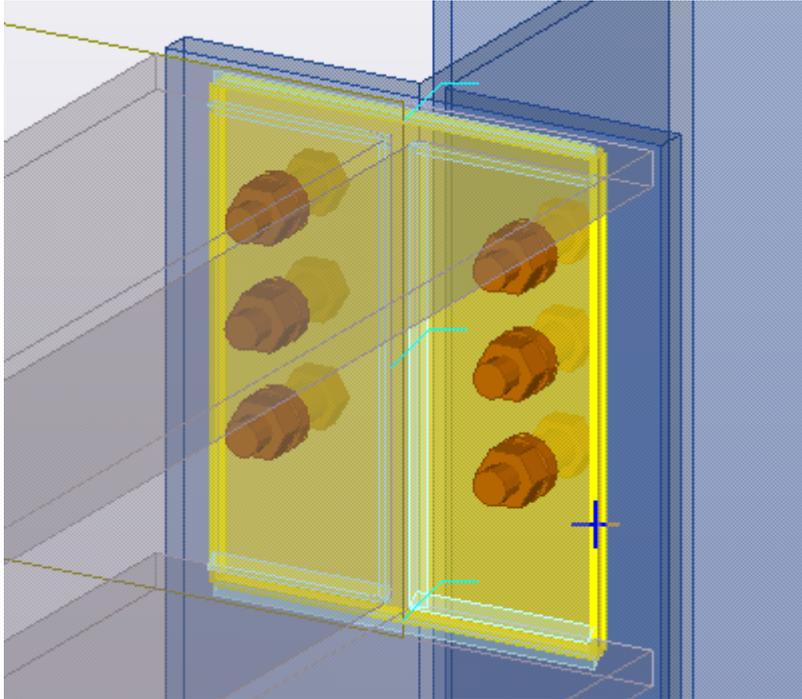
Use *distance variables* to bind component objects to a plane. Binding keeps the custom component at a fixed distance from the plane even if the surrounding objects change. Distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog box.

Bind objects automatically

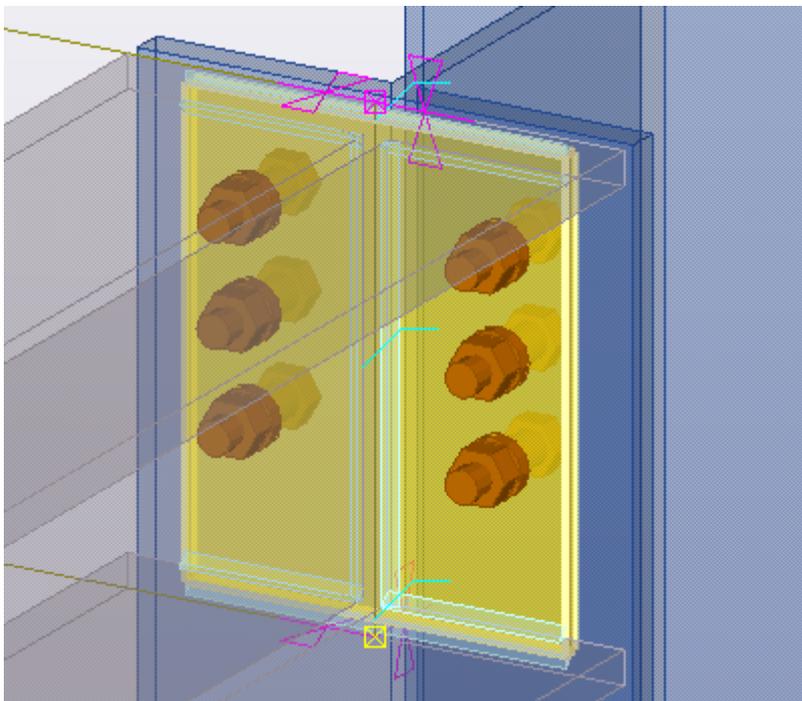
You can bind objects automatically to the main and secondary parts of a connection or detail. The selected objects, or their handles, are bound to existing planes if the objects (or handles) are located exactly on the plane.

NOTE You cannot bind [custom parts \(page 738\)](#) automatically, because they do not have a main part.

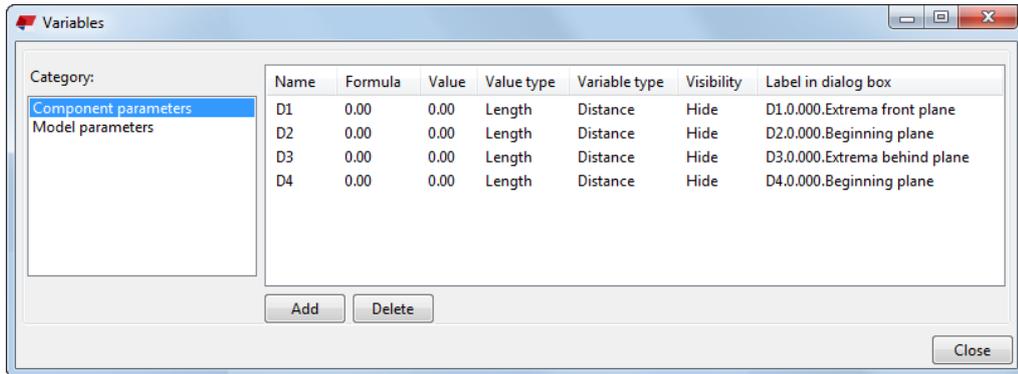
1. In the custom component editor, click the **Create distances variables automatically** button .
2. Select an object that has [handles \(page 303\)](#).



3. Click the middle mouse button to bind the object.
Tekla Structures binds the object from a maximum of three directions to the existing planes.
Tekla Structures displays a distance symbol for each binding. Select the object to see the bindings.



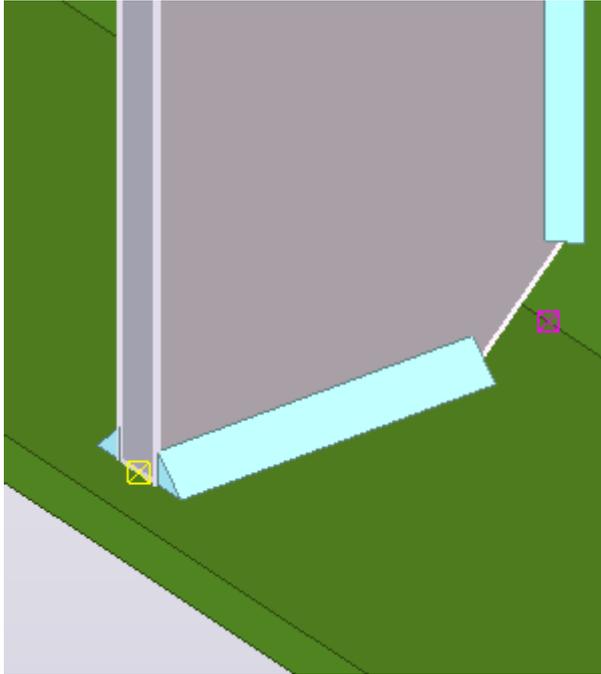
The corresponding distance variables are shown in the [Variables \(page 875\)](#) dialog box:



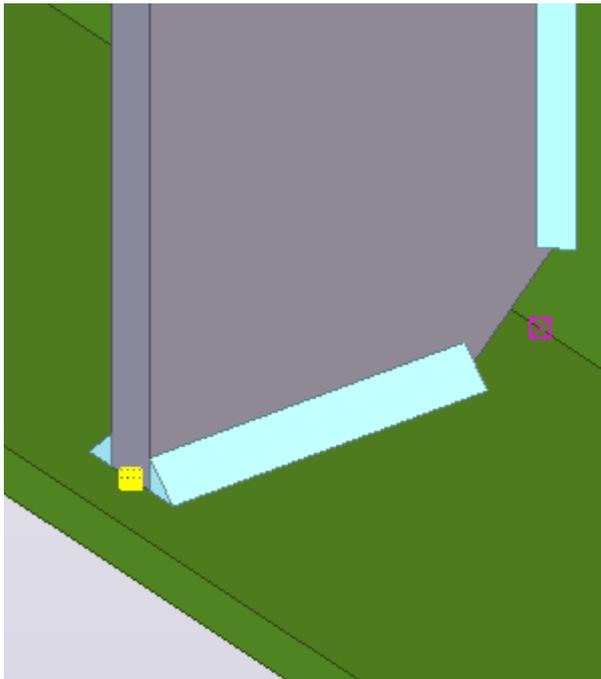
Bind objects manually

Create the bindings manually if you want to bind a custom component from specific handles only. You can bind an object to a maximum of three planes.

1. Ensure that **Direct modification**  is switched off.
The selection of handles is easier when **Direct Modification** is off.
2. Ensure that you are using a model view that shows object faces.
On the **View** tab, click **Rendering**, and use one of the following options:
 - **Parts grayscale** (Ctrl+3)
 - **Parts rendered** (Ctrl+4)
3. In a custom component view, select the custom component to see its [handles \(page 303\)](#).

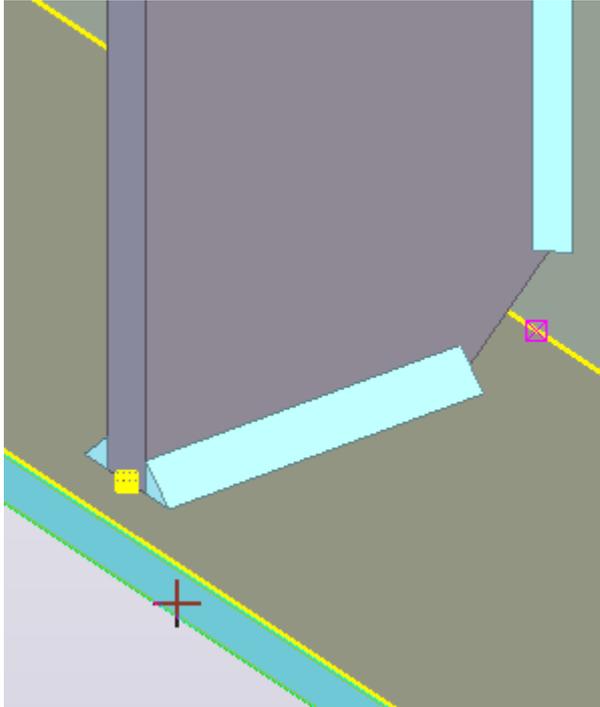


4. Select the handle that you want to bind to a plane.



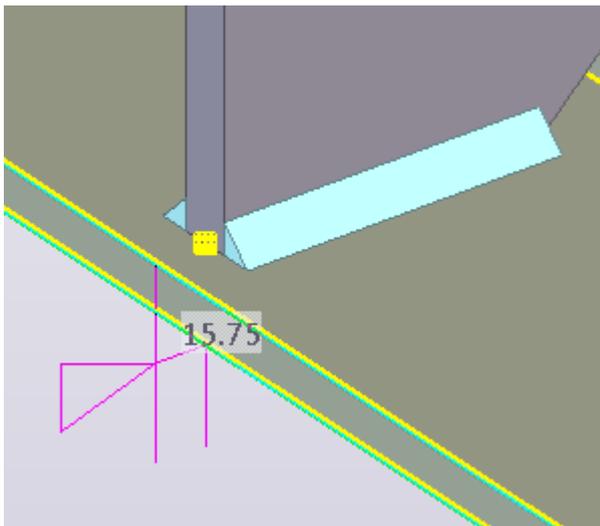
5. In the custom component editor, click the **Add fixed distance** button . You can also right-click and select **Bind to Plane**.
6. Move the mouse pointer in a custom component view to highlight the plane that you want to bind with the handles.

For example:

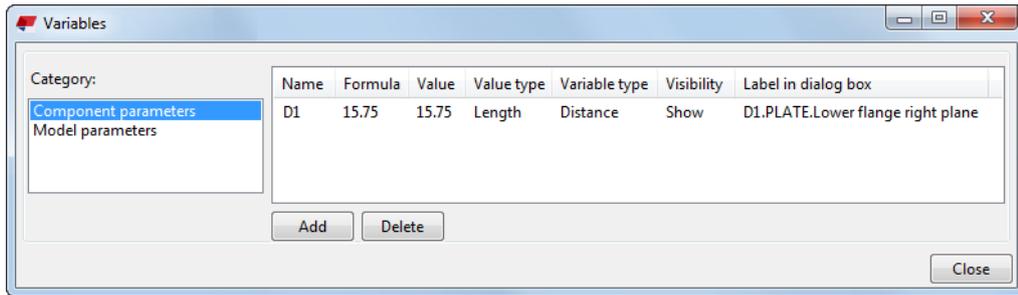


NOTE If you cannot highlight the correct plane, [change the plane type \(page 872\)](#) on the **Custom component editor** toolbar. Boundary and component planes work for most profile types, so try to use them whenever you can.

7. Click the plane to create the binding.
Tekla Structures displays a distance symbol for the binding.



The corresponding distance variable is shown in the **Variables** dialog box:



Test a binding

Test all bindings to see that they work correctly.

1. Double-click the binding symbol in a custom component view.
The **Distance Properties** dialog box opens.



2. In the **Value** box, enter a new value.
3. Click **Modify**.
You should see the binding change in the model.

TIP Alternatively, you can test the binding in the [Variables \(page 875\)](#) dialog box:

- a. Enter a new value in the **Formula** box.
- b. Press **Enter**.

You should see the binding change in the model.

Delete a binding

Bindings cannot be modified, but you can delete the existing bindings and then create new ones to rebind the objects.

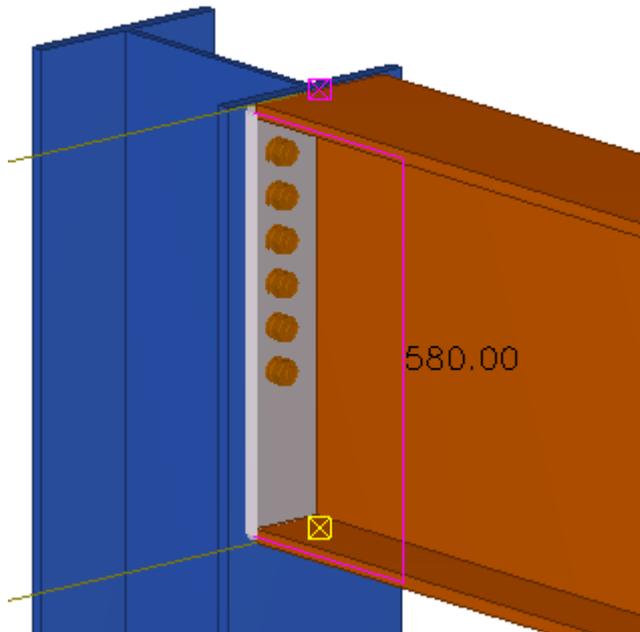
1. Select the binding in a custom component view.
2. Press **Delete**.

You can also select the binding in the [Variables \(page 875\)](#) dialog box and then click the **Delete** button.

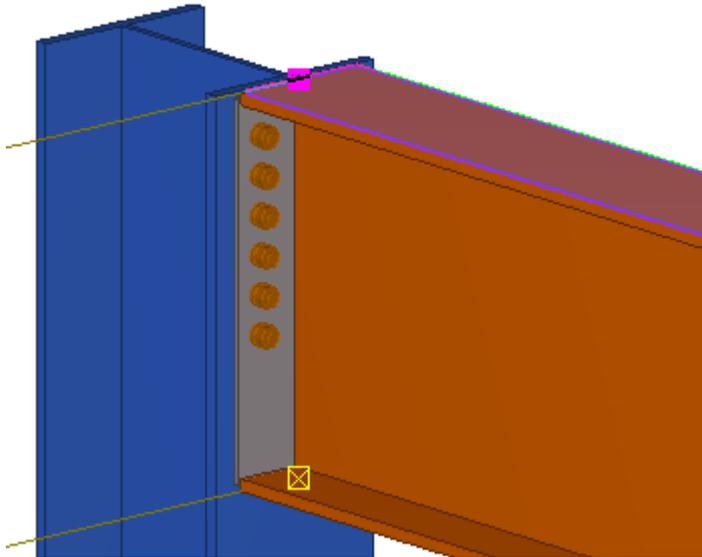
Example: Bind an end plate to a plane

In this example, you will bind the end plate top to the upper side of the beam.

1. Ensure that **Direct modification**  is switched off. The selection of end plate handles is easier when **Direct Modification** is off.
2. In a custom component view, select the end plate to see the end plate handles.



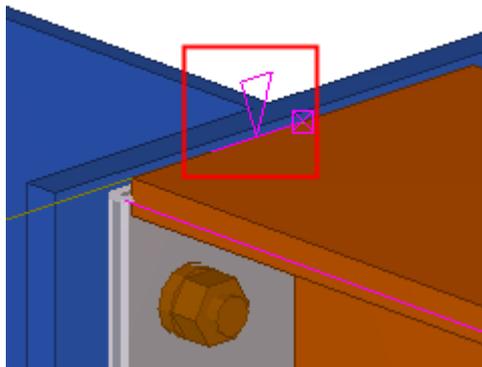
3. Select the top handle of the end plate.
4. Right-click the top handle and select **Bind to Plane**.
5. Move the pointer over the upper side of the beam flange to highlight it.



Here we are using the boundary plane type. If the part profile changes, the boundary plane is always found.

NOTE If you cannot highlight the desired plane, [change the plane type \(page 872\)](#) on the **Custom component editor** toolbar.

6. Click the upper side of the beam flange.
A distance symbol appears in the custom component views.



7. Give a descriptive name for the binding you created:
 - a. In the custom component editor, click the **Display variables** button .

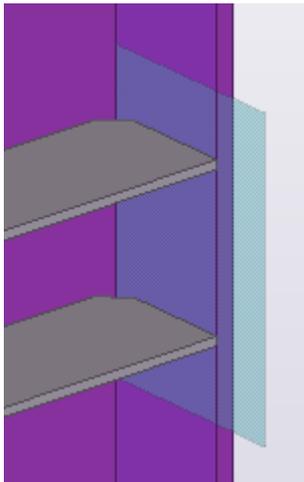
The **Variables** dialog box opens.
 - b. In the **Label in dialog box** box, enter `Plate Top to Flange Top` as the name of the new binding.

Bind component objects using magnetic construction planes or lines

Instead of binding each component object handle to a plane separately, you can use magnetic construction planes and lines. The objects that are directly on a magnetic construction plane (or line) will move with the plane (or line), which means you only need to create one distance variable instead of 8, for example.

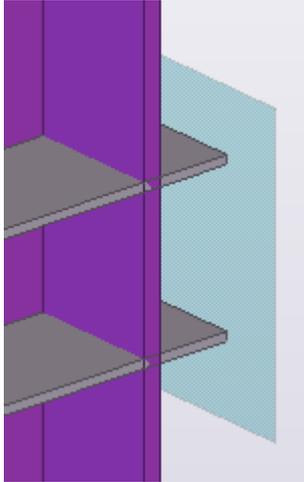
Bind handles using a magnetic construction plane

1. In the custom component editor, click  **Add construction plane**.
2. Pick four points to define the shape of the construction plane.
For example, create a plane that goes through all the handles and chamfers of the custom component.
3. Click the middle mouse button.
Tekla Structures creates a construction plane. For example:



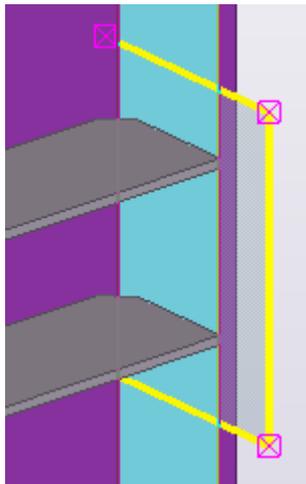
4. Double-click the plane. The plane properties are shown in the property pane.
5. Enter a name for the plane.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.

Now when you move the construction plane, all handles that are on the plane are moved as well:



8. Bind the construction plane to a part face:
 - a. Select the construction plane, right-click and select **Bind to plane**.
 - b. Select a suitable part face.

For example, the inner flange of the column:



Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction plane will follow.

NOTE Only the objects whose reference points are directly on the magnetic construction plane are affected. By default, the magnetic distance is 0.2 mm. To change this setting, use the advanced option `XS_MAGNETIC_PLANE_OFFSET`.

Bind handles using a magnetic construction line

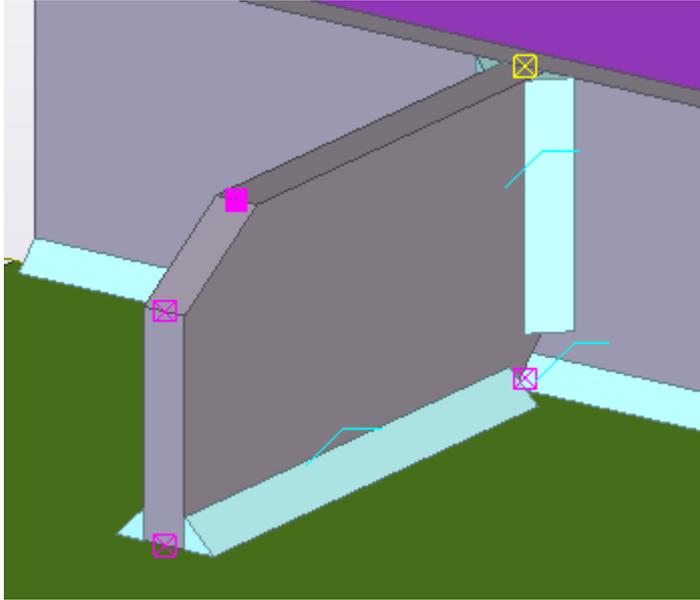
1. In the custom component editor, click  **Add construction line**.
2. Pick the starting point of the construction line.
3. Pick the end point of the construction line.
Tekla Structures creates a construction line.
4. Double-click the line. The line properties are shown in the property pane.
5. Enter a name for the line.
6. In the **Magnetic** list, select **Yes**.
7. Click **Modify**.
Now when you move the construction line, all handles that are on the line are moved as well.
8. Bind the construction line to a part face:
 - a. Select the construction line, right-click and select **Bind to plane**.
 - b. Select a suitable part face.
Tekla Structures displays a distance symbol for the binding. Now if you move the part face, the handles on the magnetic construction line will follow.

Add a distance between component objects

Use *reference distance variables* to add a distance between two points or a point and a plane. The reference distance changes as you move the objects it refers to. You can use reference distances in calculations, for example, to determine the spacing of rungs on a ladder. Reference distance variables automatically get the prefix **D** (distance), which is shown in the **Variables** dialog box.

1. In a custom component view, select a [handle \(page 303\)](#).

This is the starting point for your measurement.

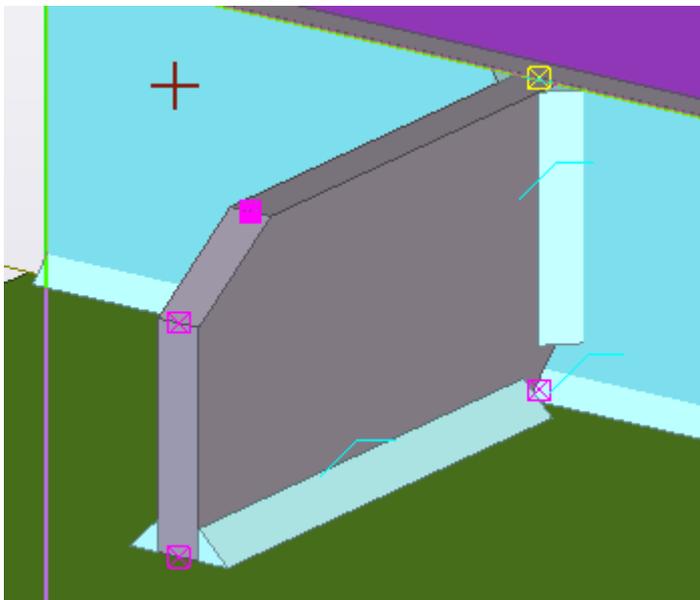


2. In the custom component editor, click the **Add reference distance** button



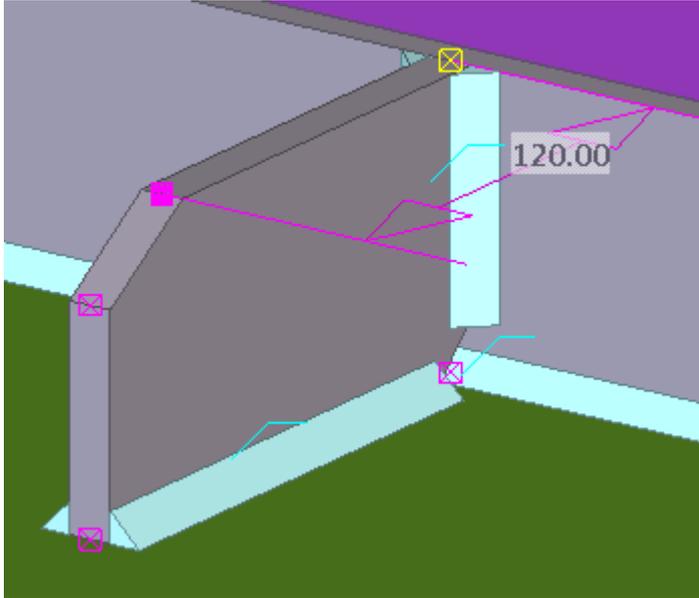
3. Move the mouse pointer in the view to highlight a plane.

This will be the end point for your measurement. If you cannot highlight the correct plane, change the [plane type \(page 872\)](#) on the **Custom component editor** toolbar.

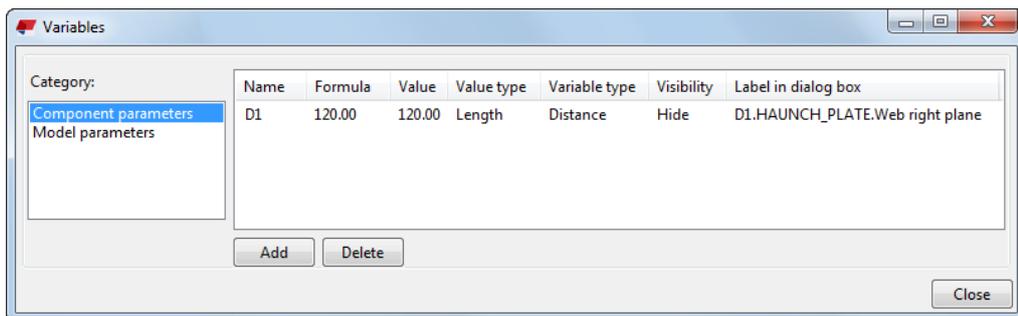


4. Click the plane to select it.

Tekla Structures displays the distance.



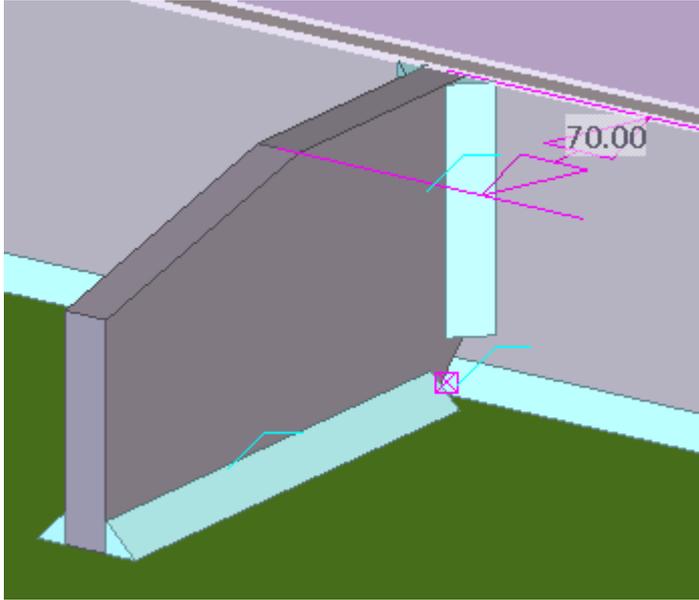
The corresponding reference distance variable is shown in the **Variables** dialog box:



Note that the **Add reference distance** command remains active. You can click more planes if you want to measure other distances.

5. To stop measuring, press **Esc**.
6. To check that the reference distance works correctly, move the handle.

The distance changes accordingly. For example:



Set object properties using parametric variables

Use *parametric variables* to set basic properties (such as name, material, profile, position number, and so on) for any object the custom component creates. Parametric variables automatically get the prefix **P** (parameter), which is shown in the **Variables** dialog box.

In the following example, we will create a variable that sets all welds in a custom component to a given size. After creating the variable, we will be able to change the weld size directly in the custom component's dialog box.

1. In the custom component editor, click the **Display variables** button .
The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. In the **Name** box, enter a name for the variable.
You can also use the default name, such as P1. In our example, we will enter `Weldsize` as the name of the variable.
4. In the **Value type** list, select a suitable [value type \(page 875\)](#).
The type determines what kind of values can be used with this variable. In our example, we will select **Length**, which is suitable for lengths and distances.
5. In the **Formula** box, enter a value or variable formula.
In our example, we will leave this box empty.

6. In the **Label in dialog box** box, enter a descriptive name for the parametric variable.

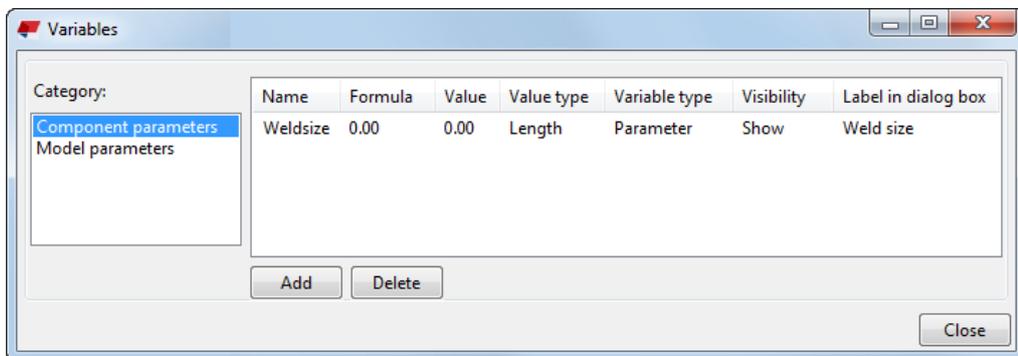
This label will be shown in the custom component's dialog box. In our example, we will enter `Weld size` as the label.

7. In the **Visibility** list, define whether the variable will be visible in the custom component's dialog box.

Hide the variable if you only use it in calculations. Show the variable if you want to be able to edit the value in the custom component's dialog box. In our example, we will select **Show**.

8. Click **Close**.

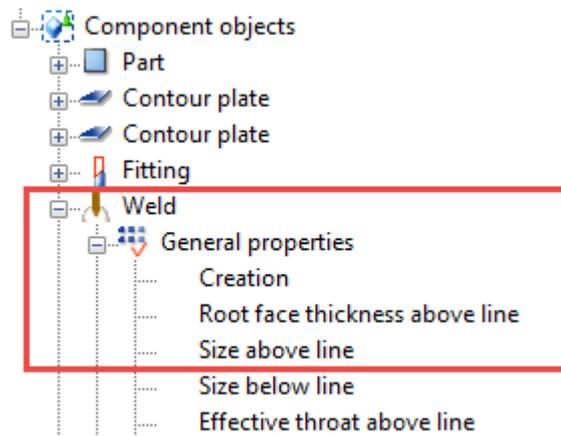
In our example, we have now created a parametric variable with the following settings:



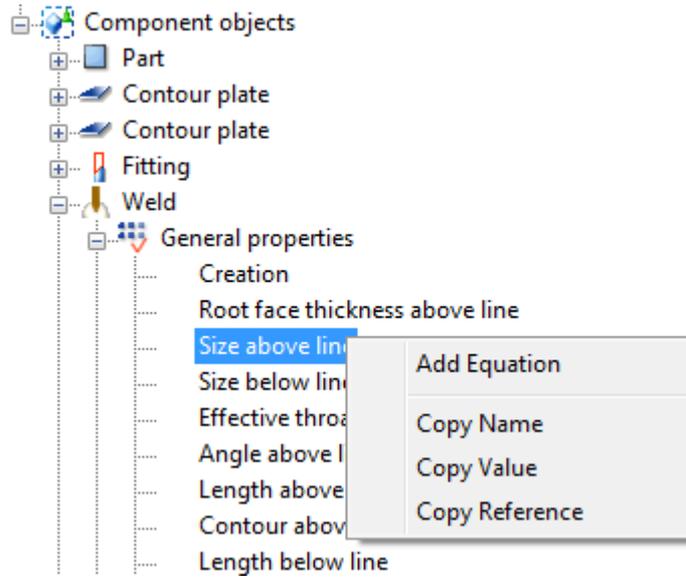
9. In the **Custom component browser**, link the variable to the desired object property.

- a. Select the property.

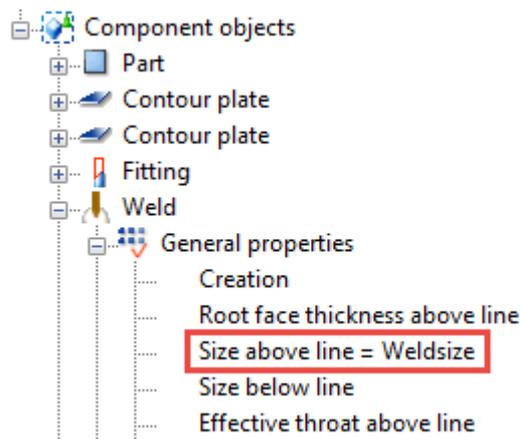
In our example, we will select the **Size above line** property of the uppermost weld.



- b. Right-click the property and select **Add Equation**.



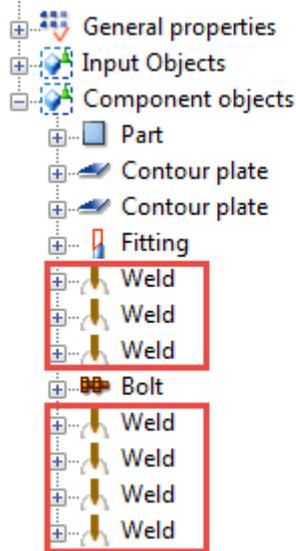
- c. After the equal sign, enter the name of the parametric variable. In our example, we will enter `Weldsize` here.



You can now modify the **Size above line** property by using the **Weld size** box in the custom component's dialog box.

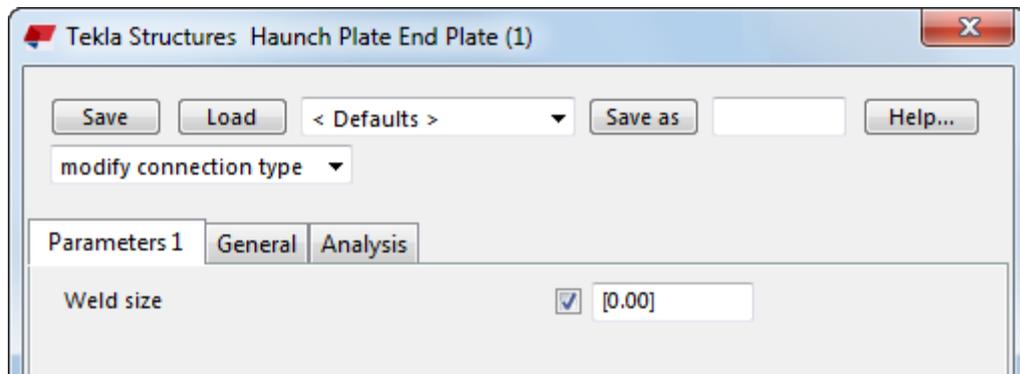
10. Repeat step 9 for any other property of the same type, if needed.

In our example, we will repeat the procedure for the other welds as well, so that they will all be linked to the **Weld size** box in the custom component's dialog box.



11. [Save the custom component. \(page 823\)](#)

The variable is now displayed in the custom component's dialog box, unless you set the visibility of the variable to **Hide** in step 7.



If we change the weld size value now, the size of all welds within the custom component will change accordingly.

See also

[Copy properties and property references from another object \(page 780\)](#)

Copy properties and property references from another object

You can copy properties, such as names and values, from other objects and use them to determine the properties of a custom component. You can also

copy property *references*. The link is dynamic, so when the property changes, the reference reflects the change. For example, you can use a beam length reference in variable formulas. Even if the length changes, the correct value is always used in calculations.

1. In the **Custom component browser**, browse for the object property you want to copy.

To find the required component object more easily, select it in a custom component view. Tekla Structures highlights the selected object in the **Custom component browser**.

2. Right-click the property and select one of the following:

- **Copy Name**

Copies the name of the object. For example, `Material`.

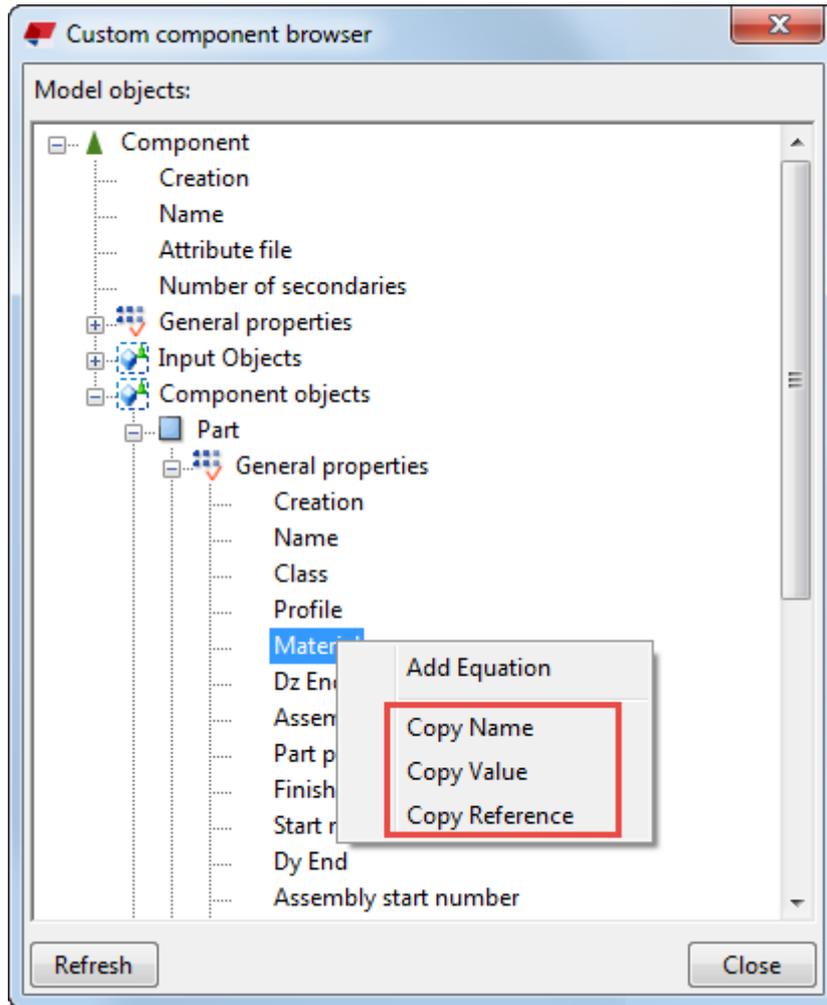
- **Copy Value**

Copies the value the object currently has. For example, `S235JR`.

- **Copy Reference**

Copies the link to the property. For example,

```
fP(Material, "ID57720EEE-0000-000E-3134-363730393237").
```



3. Right-click where you want to insert the object property, and then select **Paste**.

For example, you can paste a reference to the **Formula** box in the [\(page 875\)](#) dialog box to use it in a calculation.

See also

[Examples of parametric variables and variable formulas \(page 797\)](#)

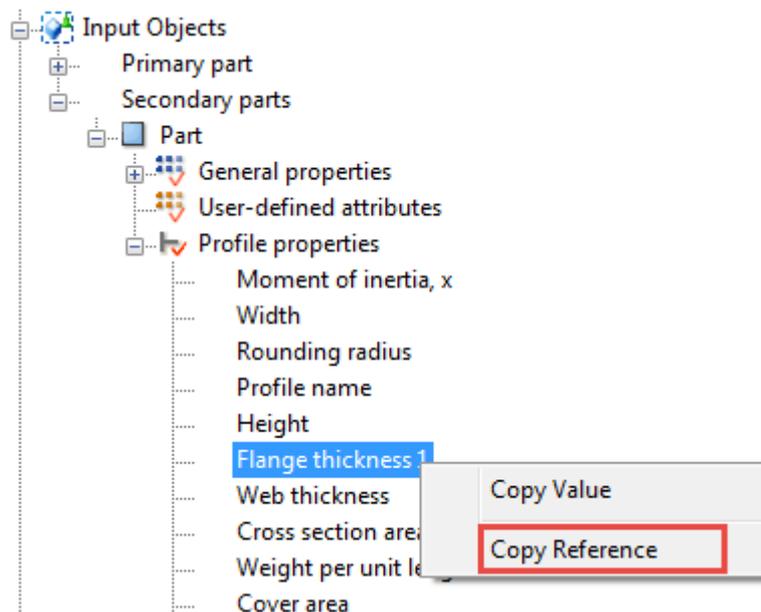
Create a variable formula

Use variable formulas to add more intelligence to your custom components. Variable formulas always begin with the equal sign (=). At its simplest, a formula can be a simple dependency between two variables, stating that P2 equals half of P1 ($P2=P1/2$), for example. To create more complex calculations, you can use functions and operators inside the formula. For example, you can

add mathematical expressions, **if** statements, references to object properties, and so on.

In the following example, we will create a formula that sets the weld size to half the thickness of the secondary part flange. When the component is used in a model, Tekla Structures will use the thickness of the secondary part flange to calculate the size of the weld.

1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. In the **Name** box, enter a name for the variable.
In our example, we will enter w as the name of the variable.
4. In the **Custom component browser**, go to **Input objects** --> **Secondary parts** --> **Part** --> **Profile properties**.
5. Right-click **Flange thickness 1** and select **Copy reference**.

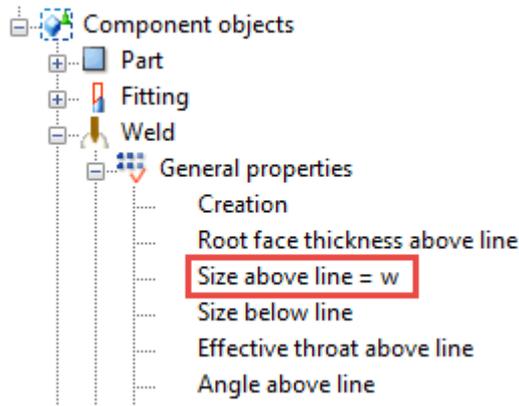


6. In the **Formula** box, type $=$, right-click, and select **Paste**.
Tekla Structures pastes the reference to flange thickness from the clipboard.
7. After the flange thickness formula, enter $*0.5$.
The formula should now read:
`=fP(Flange thickness 1, "GUID")*0.5`
8. Set the other values as follows:
 - a. In the **Value type** list, select **Length**.

b. In the **Visibility** list, select **Hide**.

Name	Formula	Value	Value type	Variable type	Visibility
w	=fP(Flange thickness 1,"ID648C83F9-24EC-442B-9363-AA20EAE834A")*0.5	4.00	Length	Parameter	Hide

9. In the **Custom component browser**, go to **Component objects** --> **Weld** --> **General properties**.
10. Right-click **Size above line**, select **Add equation** and type = w.



Functions in variable formulas

You can use functions to calculate values for parametric variables. Variable formulas always begin with the equal sign (=).

For more information, see [Set object properties using parametric variables \(page 777\)](#).

Arithmetic operators

Use arithmetic operators to combine expressions that return numeric values. You can use the following arithmetic operators:

Operator	Description	Notes
+	addition	Use also to create strings of parameters.
-	subtraction	
*	multiplication	Multiplication is faster than division. =D1 * 0.5 is faster than =D1 / 2
/	division	

Logical and comparison operators

Use logical and comparison operators inside **if** statements. You can use **if-then-else** statements to test a condition and to set the value according to the result.

For example:

```
=if (D1>200) then 20 else 10 endif
```

You can use the following operators inside if statements:

Operator	Description	Example
==	both sides are equal	
!=	sides are not equal	
<	left side is smaller	
<=	left side is smaller or equal	
>	right side is smaller	
>=	right side is smaller or equal	
&&	logical AND both conditions must be true	<pre>=if (D1==200 && D2<40) then 6 else 0 endif</pre> <p>If D1 is 200 and D2 smaller than 40, the result is 6, otherwise 0.</p>
	logical OR only one condition must be true	<pre>=if (D1==200 D2<40) then 6 else 0 endif</pre> <p>If D1 is 200 or D2 is smaller than 40, the result is 6, otherwise 0.</p>

Reference functions

Use reference functions to refer to the property of another object, such as the plate thickness of a secondary part. Tekla Structures refers to the object on the system level, so if the object property changes, so does the reference function value.

You can use the following reference functions:

Function	Description	Example
<code>fTpl ("template attribute", "object GUID")</code>	Returns the template attribute value of an object that has a given object GUID.	<pre>=fTpl ("WEIGHT", "ID50B8559A-0000-010B-3133-353432373038")</pre> <p>returns the weight of an object whose GUID is ID50B8559A-0000-010B-3133-353432373038.</p>

Function	Description	Example
fP("user-defined attribute", "object GUID")	Returns the user-defined attribute value of an object that has a given object GUID.	=fP("comment", "ID50B8559A-0000-010B-3133-353432373038") returns the user-defined attribute comment of an object whose GUID is ID50B8559A-0000-010B-3133-353432373038.
fValueOf("parameter")	Returns the value of the parameter.	If the equation is =P2+"*" +P3, the result is P2*P3 With =fValueOf("P2") + "*" +fValueOf("P3"), where P2=780 and P3=480, the result is 780*480
fRebarCatalogValue(BarGrade, BarSize, Usage, FieldName)	Returns the reinforcing bar catalog value of an object. Usage can be either 2 ("Tie") or 1 ("Main"). FieldName must be one of the following: <ul style="list-style-type: none"> • 0 NominalDiameter • 1 ActualDiameter • 2 Weight • 3 MinRadius • 4 Hook1Radius • 5 Hook1Angle • 6 Hook1Length • 7 HookRadius • 8 Hook2Angle • 9 Hook2Length 	fRebarCatalogValue("A500HW", "10", 1, 2) returns the size, usage, and weight of an object whose reinforcing bar grade is A500HW.

Function	Description	Example
	<ul style="list-style-type: none"> • 10 Hook3Radius • 11 Hook3Angl • 12 Hook3Length • 13 Area 	

ASCII file as a reference function

You can refer to ASCII files to get data. Tekla Structures searches for the files in the following order:

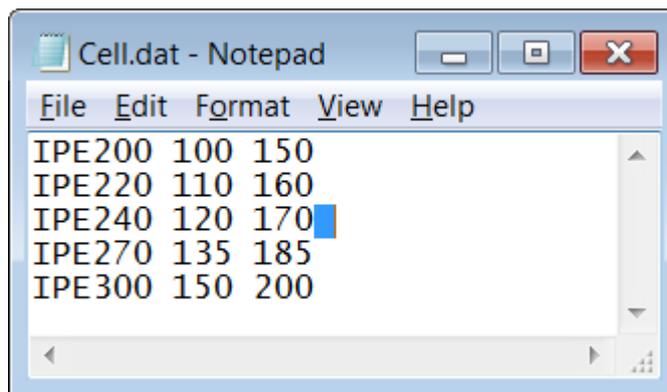
1. model
2. ..\TeklaStructuresModels\<<model>\CustomComponentDialogFiles\
3. project (set with advanced option XS_PROJECT)
4. firm (set with advanced option XS_FIRM)
5. system (set with advanced option XS_SYSTEM)

The format for reading files is the following:

```
fVF("filename", "key_value_of_row", column_number)
```

- Key value of row is a unique text value.
- Column number is an index starting from 1.

NOTE Enter a space at the end of each row in the ASCII file. Otherwise the information is not read correctly.



Example

The =fVF("Overlap.dat", "MET-202Z25", 5) function is in the **Formula** box in the **Variables** dialog box. The function gets the value 16.0 for the profile MET-202Z25, from the Overlap.dat file.

Name	Formula	Value	Value type	Variable type	Visibility
P1	=fVF("Overlap.dat", "MET-202Z25", 5)	16.00	Text	Parameter	Show

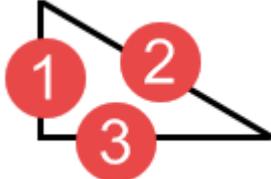
MET-202Z20	201	MET-S202Z20	3	16	1	1	32	32	11
MET-202Z23	201	MET-S202Z23	3	16	1	1	32	32	11
MET-202Z25	201	MET-S202Z25	3	16	1	1	32	32	11
MET-232C16	213	MET-CS232	3	16	2	1	32	32	14
MET-232C18	213	MET-CS232	3	16	2	1	32	32	14
MET-232C20	213	MET-CS232	3	16	2	1	32	32	14

1. Key value of the row (MET-202Z25)
2. Column number (5)

Mathematical functions

Use mathematical functions to create more complex mathematical expressions. You can use the following functions:

Function	Description	Example
fabs(parameter)	Returns the absolute value of the parameter	=fabs (D1) returns 15 if D1 = -15
exp(power)	Returns e raised to the power e is Euler's number.	=exp (D1) returns 7.39 if D1 = 2
ln(parameter)	Returns the natural logarithm of the parameter (base number e)	=ln (P2) returns 2.71 if P2 = 15
log(parameter)	Returns the logarithm of the parameter (base number 10)	=log (D1) returns 2 if D1=100
sqrt(parameter)	Returns the square root of the parameter	=sqrt (D1) returns 4 if D1 = 16
mod(dividend, divider)	Returns the modulo of the division	=mod (D1, 5) returns 1 if D1 = 16
pow(base number, power)	Returns the base number raised to the specified power	=pow (D1, D2) returns 9 if D1 = 3 and D2 = 2

Function	Description	Example
hypot(side1,side2)	Returns the hypotenuse  1. side1 2. hypotenuse 3. side2	=hypot (D1, D2) returns 5 if D1 = 3 and D2 = 4
n!(parameter)	Returns the factorial of the parameter	=n! (P2) returns 24 if P2 = 4 (1*2*3*4)
round(parameter, accuracy)	Returns the parameter rounded off to the given accuracy	=round(P1, 0.1) returns 10.600 if P1 = 10.567
PI	Returns the value of pi to 31 decimal places	=PI returns 3.1415926535897932 384626433832795

Statistical functions

Use statistical functions to calculate sums and averages, and to round values. You can use the following statistical functions:

Function	Description	Example (P1 = 1.4 P2 = 2.3)
ceil()	Returns the smallest whole number greater than or equal to the parameter	=ceil(P1) returns 2
floor()	Returns the largest whole number less than or equal to the parameter	=floor(P1) returns 1
min()	Returns the smallest parameter	=min(P1, P2) returns 1.4
max()	Returns the largest parameter	=max(P1, P2) returns 2.3
sum()	Sum of the parameters	=sum(P1, P2) returns 3.7
sqsum()	Sum of the squared parameters: (parameter1) ² + (parameter2) ²	=sqsum(P1, P2) returns 7.25
ave()	Average of the parameters	=ave(P1, P2) returns 1.85

Function	Description	Example (P1 = 1.4 P2 = 2.3)
sqave()	Average of the squared parameters	=sqave(P1, P2) returns 3.625

Example: Ceil and floor statistical functions

In this example, you have the following parametric variables:

- Beam length: P1 = 3500
- Post spacing: P2 = 450

$P1 / P2 = 7.7778$

You can use the `ceil` and `floor` statistical functions to round the value and then use the rounded value as the number of beam posts:

- `=ceil(P1/P2)` returns 8
- `=floor(P1/P2)` returns 7

Data type conversion functions

Use data type conversion functions to convert values into another data type. You can use the following data type conversion functions:

Function	Description	Example
int()	Converts data to integer	Useful especially for calculating profile dimensions: <code>=int(100.0132222000)</code> returns 100, if decimals are set to 0 in the Options dialog box
double()	Converts data to a double	
string()	Converts data to string	
imp()	Converts imperial units Use this function in calculations instead of imperial units. You cannot use imperial units directly in calculations.	For the following examples, length unit is set to mm and decimals are set to 2 in the Options dialog box. <code>=imp(1, 1, 1, 2)</code> meaning 1 foot 1 1/2 inch returns 342.90 mm <code>=imp(1, 1, 2)</code> meaning 1 1/2 inches returns 38.10 mm <code>=imp(1, 2)</code> meaning 1/2 inches returns 12.70 mm <code>=imp(1)</code> meaning 1 inch returns 25.40 mm <code>=3' / 3"</code> is not possible, but <code>=imp(36) / imp(3)</code> is ok

Function	Description	Example
vwu(value, unit)	Converts the length values and angle values. The available units are: <ul style="list-style-type: none"> • "ft" ("feet", "foot") • "in" ("inch", "inches") • "m" • "cm" • "mm" • "rad" • "deg" 	<pre>=vwu(4.0, "in")</pre> returns 101.60 mm, if length unit is set to mm and decimals are set to 2 in the Options dialog box <pre>=vwu(2.0, "rad")</pre> returns 114.59 degrees, if angle is set to degrees and decimals are set to 2 in the Options dialog box

NOTE The units depend on the settings in **File menu --> Settings --> Options --> Units and decimals** .

String operations

Use string operations to manipulate character strings. Strings must be inside quotation marks in variable formulas.

You can use the following string operations:

Operation	Description	Example (P1 = "PL100*10")
match(parameter1, parameter2)	Returns 1 if parameters are equal and 0 if different. You can also use wildcards *, ?, and [] with the match function.	<pre>=match(P1, "PL100*10")</pre> returns 1 Accept all profiles starting with PFC: <pre>=match(P4, "PFC*")</pre> Accept profiles starting with PFC, and height starts with 2,3,4 or 5: <pre>=match(P4, "PFC[2345]*")</pre> Accept profiles starting with PFC, heights are 200,300,400 or 500 and width starts with 7: <pre>=match(P4, "PFC[2345]00?7")</pre>
length(parameter)	Returns the number of characters in the parameter.	<pre>=length(P1)</pre> returns 8
find(parameter, string)	Returns the order number (starting at zero) of the specified string and -1 if the specified string is not	<pre>=find(P1, "*")</pre> returns 5

Operation	Description	Example (P1 = "PL100*10")
	found from the parameter.	
getat(parameter, n)	Returns the n:th (starting at zero) character from the parameter.	=getat(P1, 1) returns "L"
setat(parameter, n, character)	Sets the n:th (starting at zero) character to the specified character in the parameter.	=setat(P1, 0, "B") returns "BL100*10"
mid(string, n, x)	Returns x characters from the string starting from n:th (starting at zero) character. If you leave out the last argument (x), returns the last part of the string.	=mid(P1, 2, 3) returns "100"
reverse(string)	Reverses the given string.	=reverse(P1) returns "01*001LP"

Example 1

To define profile size PL100*10 with two variables P2 = 100 and P3 = 10, enter the formula as follows:

```
= "PL" + P2 + "*" + P3
```

Example 2

Tekla Structures handles bolt spacings as strings. To define bolt spacing, set **Value type** to **Distance list** and enter the formula as follows:

```
= P1 + " " + P2
```

This results in 100 200, if P1 = 100 (**length**) and P2 = 200 (**length**).

Trigonometric functions

Use trigonometric functions to calculate angles. You can use the following trigonometric functions:

Function	Description	Example
sin()	Returns the sine value	=sin(d45) returns 0.71
cos()	Returns the cosine value	=cos(d45) returns 0.71
tan()	Returns the tangent value	=tan(d45) returns 1.00
asin()	Inverse function of sin(), return value in radians	=asin(1) returns 1.571 rad
acos()	Inverse function of cos(), return value in radians	=acos(1) returns 0 rad

Function	Description	Example
atan()	Inverse function of tan(), return value in radians	=atan(1) returns 0.785 rad
sinh()	Returns the hyperbolic sine value	=sinh(d45) returns 0.87
cosh()	Returns the hyperbolic cosine value	=cosh(d45) returns 1.32
tanh()	Returns the hyperbolic tangent value	=tanh(d45) returns 0.66
atan2()	Returns the angle whose tangent is the quotient of the two numbers. Return value in radians	=atan2(1, 3) returns 0.32

NOTE When you use trigonometric functions in variable formulas, you need to include a prefix to define the unit. If you do not include a prefix, Tekla Structures uses radians as the default unit.

- d is degree. For example, `sin(d180)`
- r is radians (default). For example, `sin(r3.14)` or `sin(3.14)`

Market size function

Use the market size function in a custom component to select a suitable plate dimension (usually plate thickness) from the available market sizes. For example, a plate's thickness should match the web of a beam.

Function	Description	Example
fMarketSize(material, thickness, extrastep)	Returns the next available market size for the material from the <code>marketsize.dat</code> file, based on the thickness you specify. The file must be in the .. <code>\environments</code> <code>\your_environment</code> <code>\profil</code> folder or the system folder. For extrastep enter a number to define the increment to the next size (default is 0).	=fMarketSize("S235JR", 10, 0)

Example

In this example, you have the following data in `marketsize.dat`:

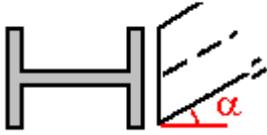
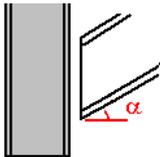
```
S235JR, 6, 9, 12, 16, 19, 22
SS400, 1.6, 2.3, 3.2, 4.5, 6, 9, 12, 16, 19, 22, 25, 28, 32, 38
DEFAULT, 6, 9, 12, 16, 19, 22, 25, 28, 32, 38
```

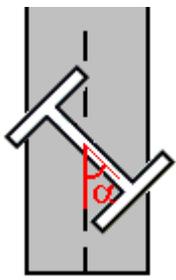
The first item in a row is a material grade followed by available plate thicknesses in millimeters. The DEFAULT line lists the thicknesses available in all other material grades.

With the above data, the function `=fMarketSize("S235JR", 10, 0)` would return 12, and `=fMarketSize("S235JR", 10, 1)` would return 16 (one size up).

Framing condition functions

Use the framing condition functions to return the skew, slope, and cant angle of the secondary beam relative to the main part (column or beam). You can use the following framing condition functions:

Function	Description	Example
<code>fAD("skew", GUID)</code>	Returns the skewed angle of the secondary part whose GUID is given. 	<code>=fAD("skew", "ID50B8559A-0000-010B-3133-353432373038")</code> returns 45 ID50B8559A-0000-010B-3133-353432373038 is the GUID of the secondary part, which is at a 45 degree angle to the main part.
<code>fAD("slope", GUID)</code>	Returns the sloped angle of the secondary part whose GUID is given. 	<code>=fAD("slope", "ID50B8559A-0000-010B-3133-353432373038")</code>

Function	Description	Example
fAD("cant", GUID)	Returns the cant angle of rotated secondary part whose GUID is given. 	=fAD("cant", "ID50B8559A-0000-010B-3133-353432373038")

- NOTE** • These functions do not return positive and negative slope and skew values. It is not possible to determine up or down slope and left or right skew with these functions.
- The maximum skew angle to return is 45 degrees.
 - Tekla Structures calculates the angles in 2D so that slope and skew are isolated from each other. For example, the skew angle is not taken into consideration when calculating the slope angle, which means that the slope angle value stays the same regardless of the secondary part's rotation around the primary part.

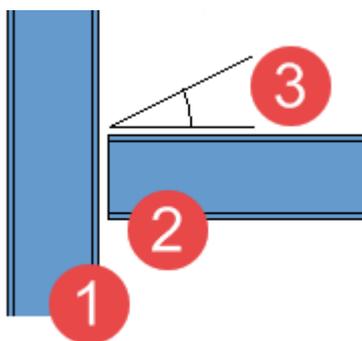
To find out the true 3D slope with the skew included, you can use the following mathematical formula:

$$\text{TRUE_SLOPE} = \text{atan}(\tan(\text{SLOPE}) * \cos(\text{SKEW}))$$

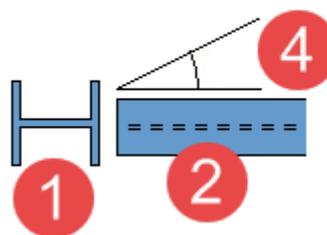
Example 1

The slope and skew are relative to a beam framing into a column.

Side view



Top view



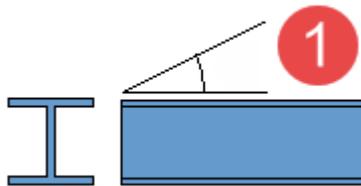
1. Column
2. Beam

3. **Slope**
4. **Skew**

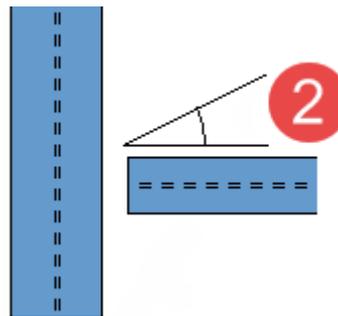
Example 2

With two beams, the **slope** is actually the horizontal skew of the beam framing into the other beam, and the vertical slope of the beam relative to the main is actually the **skew** angle.

Side view



Top view



1. **Skew**
2. **Slope**

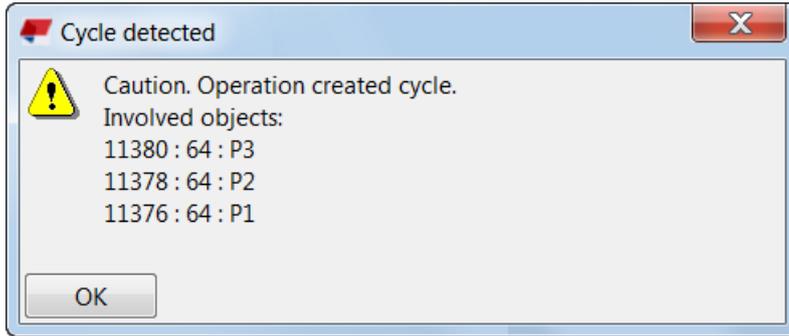
How to avoid cyclic dependencies in formulas

Be careful not to create cyclic dependencies between variables, or else the custom component will not work correctly. A cyclic dependency chain contains formulas that make a variable eventually dependent on itself.

In the following example, variable P1 becomes dependent on itself, through variables P2 and P3:

Name	Formula
P1	=P2
P2	=P3/4
P3	=P1*2

Cyclic dependencies may also occur when binding handles to other objects or when using magnetic construction planes. When you create new formulas, bindings, or magnetic construction planes, Tekla Structures checks if they create cyclic dependency chains in a custom component. If that happens, a warning message "Caution. Operation created cycle." is displayed.



Tekla Structures also writes the message "Cycle detected in parametric solver" into the session history log file and lists the objects involved in the cyclic dependency, to help you find and remove the cyclic dependency. If you do not remove it, the custom component will not work correctly.

Examples of parametric variables and variable formulas

Here you will find some examples that demonstrate how to use parametric variables and variable formulas to create intelligent custom components that adapt to changes in the model.

The examples are independent from each other.

- [Example: Set the end plate material \(page 798\)](#)

In this example, you will link a parametric variable to the end plate material of a component object.

- [Example: Create new component objects \(page 799\)](#)

In this example, you will create a parametric variable that adds bolts to the custom component.

- [Example: Replace sub-components \(page 800\)](#)

In this example, you will create a parametric variable that replaces sub-components with other sub-components.

- [Example: Modify a sub-component by using a component attribute file \(page 802\)](#)

In this example, you will create a parametric variable that modifies a sub-component on the basis of a component attribute file.

- [Example: Define the stiffener position using construction planes \(page 803\)](#)

In this example, you will use construction planes for determining the position of the stiffeners. You will position the stiffeners so that they divide the beam into three equally long sections.

- [Example: Determine the bolt size and bolt standard \(page 806\)](#)
In this example, you will create two parametric variables that determine the bolt size and bolt standard.
- [Example: Calculate the bolt group distance \(page 807\)](#)
In this example, you will create a variable formula that calculates the bolt group distance from the beam flange.
- [Example: Calculate the number of bolt rows \(page 809\)](#)
In this example, you will create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.
- [Example: Link variables to user-defined attributes \(page 811\)](#)
In this example, you will link parametric variables to the user-defined attributes of panels. You can then use the user-defined attributes in view filters to show or hide the panels.
- [Example: Calculate the number of handrail posts using a template attribute \(page 813\)](#)
In this example, you will create a variable formula that calculates the number of handrail posts based on the length template attribute of the beam. The handrail posts were created at both ends of the beam and one of them was copied with the **Array of objects (29)** component.
- [Example: Link an Excel spreadsheet to a custom component \(page 815\)](#)
In this example, you will link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.
- [Examples: Rebar set modifiers in custom components \(page 816\)](#)
In these examples, you will use rebar set modifiers for defining the properties and hooks of rebar set bars in custom components.

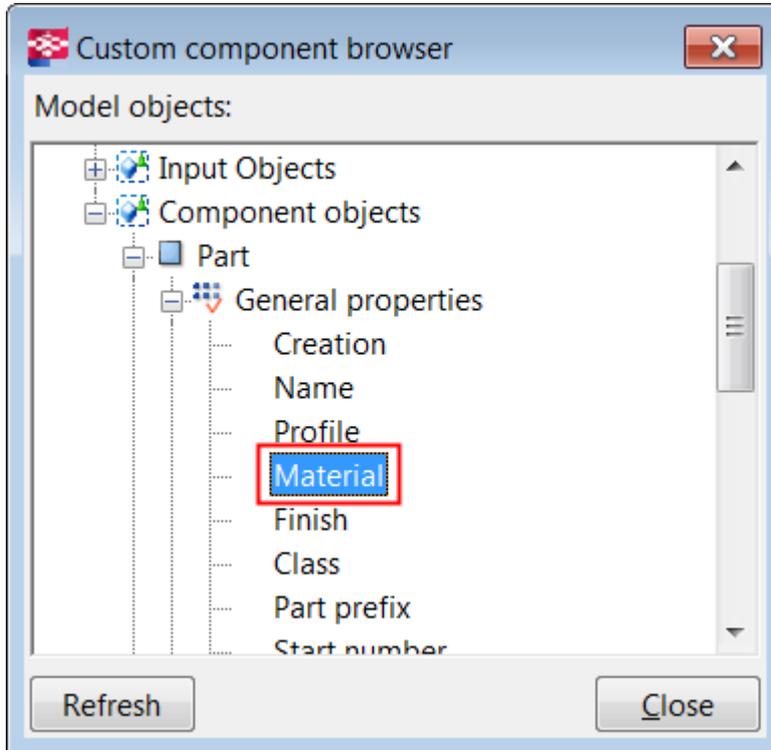
Example: Set the end plate material

In this example, you will link a parametric variable to the end plate material of a component object.

1. In the custom component editor, click the **Display variables** button .
The **Variables** dialog box opens.
2. Click the **Add** button.
A new parametric variable appears.
3. In the **Value type** list, change the variable's value type to **Material**.
4. In the **Label in dialog box** box, enter `End Plate Material`.

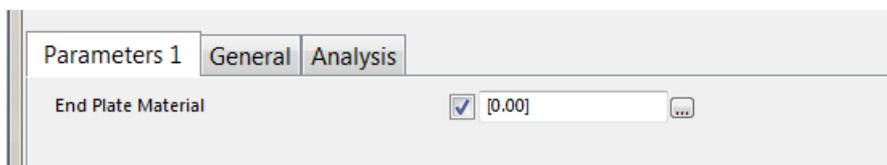
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0.00	0.00	Material	Parameter	Show	End Plate Material

- In the **Custom component browser**, browse for the end plate material.



- Right-click **Material** and select **Add Equation**.
- Enter $P1$ after the equal sign, and then press **Enter**.
- Save the custom component.
- Close the custom component editor.

You can now change the end plate material in the custom component's dialog box.



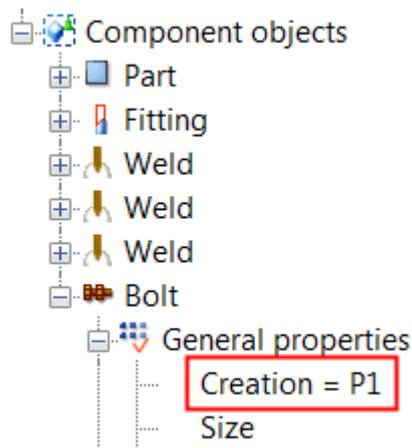
Example: Create new component objects

In this example, you will create a parametric variable that adds bolts to the custom component.

1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. Modify the variable as follows:
 - a. In the **Value type** list, select **Yes/No**.
 - b. In the **Label in dialog box** box, enter `Create bolts`.

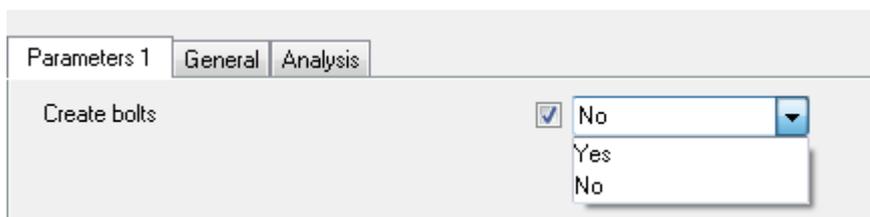
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Create bolts

4. Select the bolt group in a custom component view to highlight it in the **Custom component browser**.
5. In the **Custom component browser**, browse for **Bolt**.
6. Right-click **Creation** and select **Add Equation**.
7. Enter `P1` after the equal sign, and then press **Enter**.



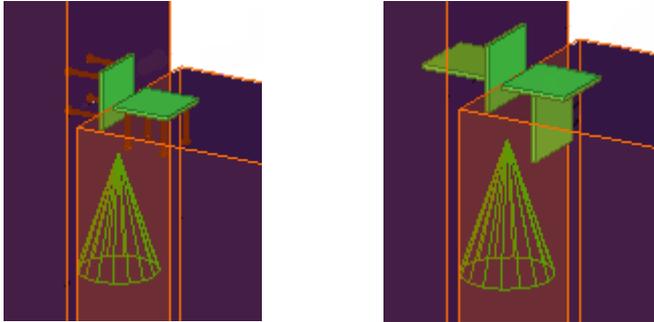
8. Save the custom component.
9. Close the custom component editor.

You now have the following option in the custom component's dialog box:



Example: Replace sub-components

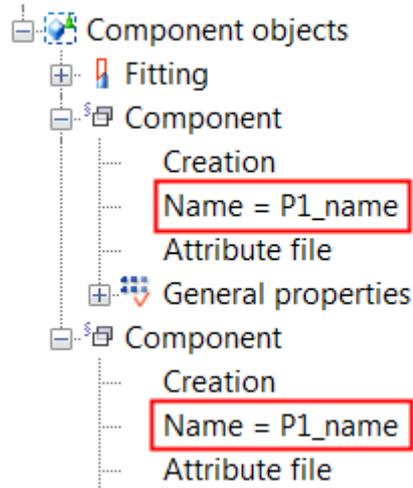
In this example, you will create a parametric variable that replaces sub-components with other sub-components.



1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. Modify the variable as follows:
 - a. In the **Value type** list, select **Component name**.
Tekla Structures automatically adds the suffix `_name` in the variable name. Do not delete the suffix.
 - b. In the **Formula** box, enter the name of the sub-component.
 - c. In the **Label in dialog box** box, enter `Cast-in plate`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	castin1	castin1	Component name	Parameter	Show	Cast-in plate

4. Link the variable to the **Name** property of both sub-components:
 - a. In the **Custom component browser**, browse for the **Name** attribute of the first sub-component.
 - b. Right-click **Name** and select **Add Equation**.
 - c. Enter `P1_name` after the equals sign.
 - d. Repeat steps 4b–4c for the other sub-component.



5. Save the custom component.
6. Close the custom component editor.

You can now change the sub-components by using the **Cast-in-plate** option in the custom component's dialog box.

Example: Modify a sub-component by using a component attribute file

In this example, you will create a parametric variable that modifies a sub-component on the basis of a component attribute file.

1. In the custom component editor, click the **Display variables** button .

The **Variables** dialog box opens.

2. Click **Add** to create a new parametric variable.
3. In the **Value type** list, select **Component attribute file**.

Tekla Structures automatically adds the suffix `_attrfile` in the variable name. Do not delete the suffix.

4. In the **Formula** box, enter the name of the component attribute file.
5. In the **Name** box, ensure that the variable has the same prefix as the variable that is linked to the component name.

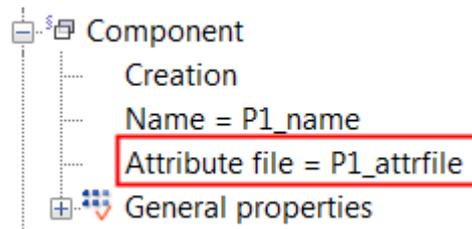
In this example, the prefix is P1.

NOTE The component name and the component attribute file variables must always have the same prefix, otherwise they do not work.

6. In the **Label in dialog box** box, enter `Properties file`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1_name	castin1	castin1	Component name	Parameter	Show	Cast-in plate
P1_attrfile	prop1	prop1	Component attribute file	Parameter	Show	Properties file

- In the **Custom component browser**, browse for the component attribute file property of the sub-component.
- Right-click **Attribute file** and select **Add Equation**.
- Enter `P1_attrfile` after the equal sign, and then press **Enter**.

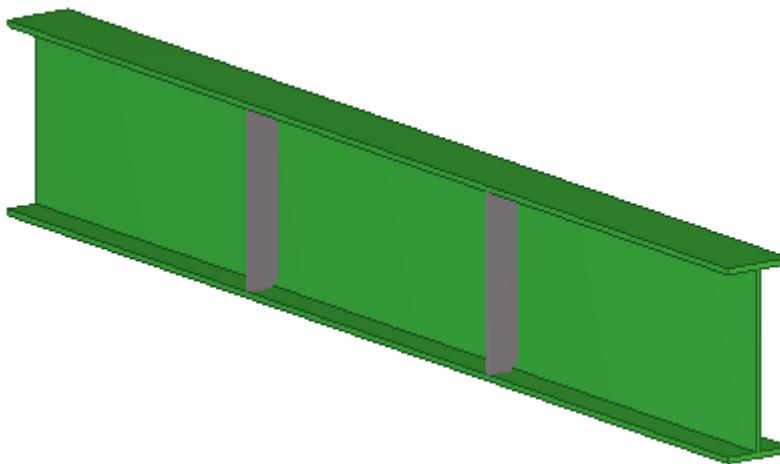


- Save the custom component.
- Close the custom component editor.

You can now modify the sub-component by using the **Properties file** option in the custom component's dialog box.

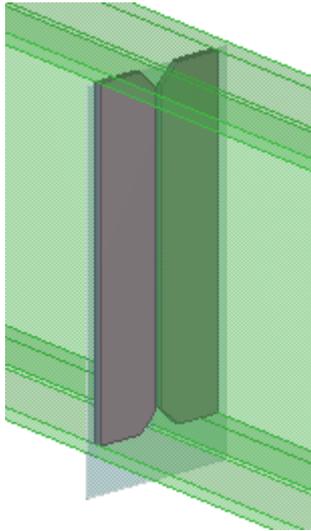
Example: Define the stiffener position using construction planes

In this example, you will use construction planes for determining the position of the stiffeners. You will position the stiffeners so that they divide the beam into three equally long sections.

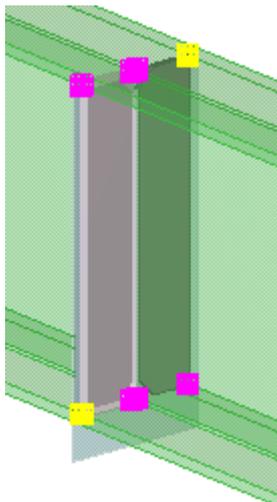


- Ensure that **Direct Modification**  is switched off. The selection of handles is easier when **Direct Modification** is off.

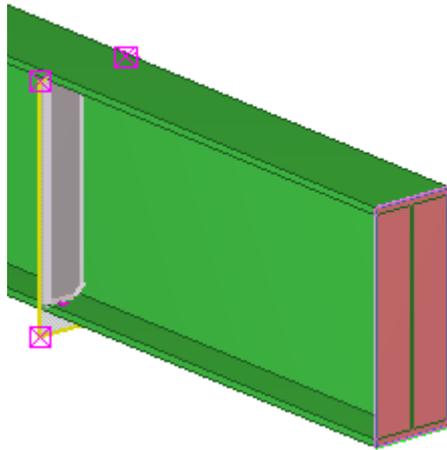
2. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
3. Click **Add** to create a new parametric variable.
4. Get the GUID of the beam.
 - a. On the ribbon, click **Inquire objects** .
 - b. Select the beam.
 - c. In the **Inquire Object** dialog box, check the GUID of the beam.
5. Modify the variable as follows:
 - a. In the **Formula** box, enter `=fTp1 ("LENGTH", "ID4C8B5E24-0000-017D-3132-383432313432")`.
 ID4C8B5E24-0000-017D-3132-383432313432 is the GUID of the beam.
 The value of the variable is now the same as the beam length. If you change the beam length, also the value changes.
 - b. In the **Label in dialog box** box, enter `Beam Length`.
6. Click **Add** to create another parametric variable.
7. Modify the new variable as follows:
 - a. In the **Formula** box, enter `=P1/3`.
 - b. In the **Label in dialog box** box, enter `3rd Points`.
8. Create a construction plane:
 - a. In the custom component editor, click the **Add construction plane** button .
 - b. Pick the required points and then click the middle mouse button to create a construction plane in the center of a stiffener at one end.



9. Bind the stiffener to the construction plane:
 - a. Select the stiffener.
 - b. Hold down **Alt** and use area selection (from left to right) to select all stiffener handles.



- c. Right-click and select **Bind to plane**.
 - d. Bind the stiffener handles to the construction plane.
10. Bind the construction plane to the beam end:
 - a. Select the construction plane.
 - b. Right-click and select **Bind to plane**.
 - c. Bind the construction plane to the beam end.



11. Repeat steps 9–11 for the stiffener at the other end.
12. In the **Formula** box, enter `=P2` for the two distance variables that bind the construction planes to the beam ends.
13. Save the custom component.
14. Close the custom component editor.

If you now change the beam length, the position of the stiffeners changes so that the stiffeners divide the beam into three equally long sections.

Example: Determine the bolt size and bolt standard

In this example, you will create two parametric variables that determine the bolt size and bolt standard.

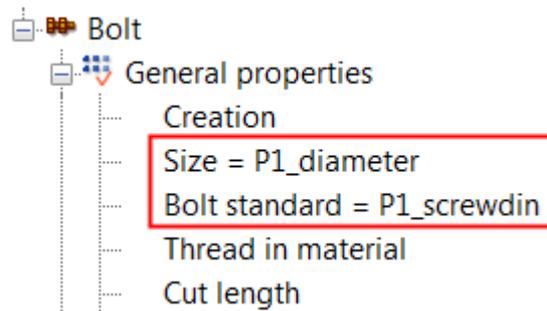
1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** twice to create two new parametric variables.
3. Modify the first variable as follows:
 - In the **Value type** list, select **Bolt size**.
Tekla Structures automatically adds the suffix `_diameter` to the name of the variables. Do not delete the suffix.
 - In the **Label in dialog box** box, enter `Bolt Size`.
4. Modify the second variable as follows:
 - a. In the **Value type** list, select **Bolt standard**.
Tekla Structures automatically adds the suffix `_screwdin` to the name of the variable. Do not delete the suffix.
 - b. In the **Name** box, change the prefix of the second variable so that the prefixes for the two variables are the same.

In this example, the prefix is P1.

Name	Formula	Value	Value type	Variable type	Visibility
P1_diameter	0.00	0.00	Bolt size	Parameter	Show
P1_screwdin	0.00	0.00	Bolt standard	Parameter	Show

NOTE The bolt size and bolt standard variables must always have the same prefix, otherwise they do not work.

- c. In the **Label in dialog box** box, enter `Bolt Standard`.
5. Link the parametric variables to the bolt group properties:
 - a. In the **Custom component browser**, browse for the size property of the component object.
 - b. Right-click **Size** and select **Add Equation**.
 - c. Enter `P1_diameter` after the equal sign, and then press **Enter**.
 - d. Right-click **Bolt standard** and select **Add Equation**.
 - e. Enter `P1_screwdin` after the equal sign, and then press **Enter**.

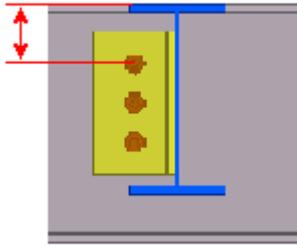


6. Save the custom component.
7. Close the custom component editor.

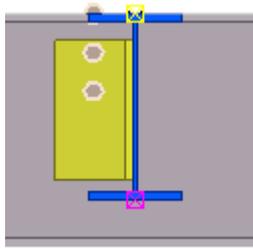
You can now determine the bolt size and bolt standard for the custom component in the custom component's dialog box.

Example: Calculate the bolt group distance

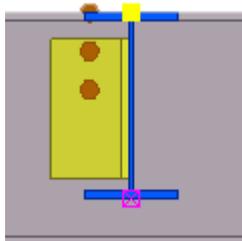
In this example, you will create a variable formula that calculates the bolt group distance from the beam flange.



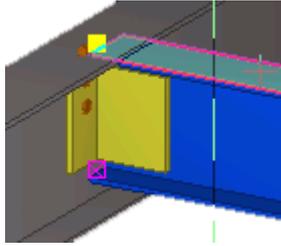
1. Modify the bolt group properties as follows:
 - a. In the custom component editor, double-click the bolt group.
The **Bolt** properties open.
 - b. Clear all values that are in the **Offset from** section.
 - c. Click **Modify**.
The bolt group moves to the same level with the start point handle of the bolt group.



2. Bind the bolt group to the beam flange:
 - a. In the custom component editor, select the bolt group.
 - b. Select the yellow top handle.



- c. Right-click the handle and select **Bind to plane**.
 - d. Select the top flange of the beam.



A new distance variable appears in the **Variables** dialog box.

3. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
4. Click **Add** to create a new parametric variable.
5. Modify the variable as follows:
 - a. In the **Formula** box, enter a distance value.
 - b. In the **Label in dialog box** box, enter `Vertical distance to bolt`.
6. In the **Formula** box, enter `--P1` for the distance variable.

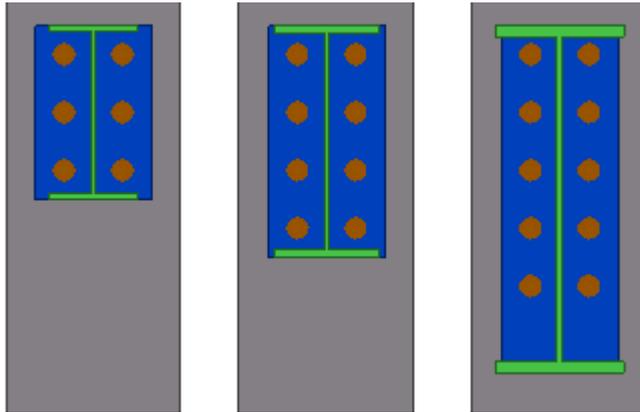
Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	=P1	-75.00	Length	Distance	Hide	D1.BOLT.BEAM
P1	75.00	75.00	Length	Parameter	Show	Vertical distance to bolt

7. Save the custom component.
8. Close the custom component editor.

You can now determine the bolt group distance from the beam flange by changing the **Vertical distance to bolt** value in the custom component's dialog box.

Example: Calculate the number of bolt rows

In this example, you will create a variable formula that calculates the number of bolt rows based on the beam height. You will use `if` statements in the calculations.



1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. In the **Value type** list, select **Number**.
4. In the **Custom component browser**, browse for height property of the beam.
5. Right-click **Height** and select **Copy Reference**.
6. In the **Formula** box, enter the following `if` statement for the parametric variable:

```
=if (fP(Height,"ID50B8559A-0000-00FD-3133-353432363133")< 301) then 2
else (if (fP(Height,"ID50B8559A-0000-00FD-3133-353432363133")>501) then 4
else 3 endif) endif
```

In the formula,

`fP(Height, "ID50B8559A-0000-00FD-3133-353432363133")` is the beam height reference copied from the **Custom component browser**. The variable gets its value in the following way:

- If the beam height is under 301 mm, the value is 2.
 - If the beam height is over 501 mm, the value is 4.
 - If the beam height is between 300 and 500 mm, the value is 3.
7. Click **Add** to create another parametric variable.
 8. In the **Value type** list, select **Distance list** for the new variable.
 9. In the **Formula** box, enter `=P1+"*"+100` for the new variable.

In the formula, 100 is the bolt spacing and the P1 value is the number of bolt rows.

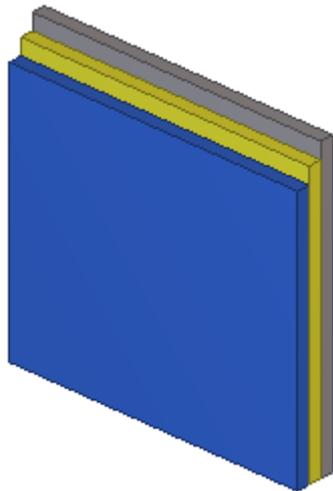
Name	Formula	Value	Value type
P1	=if (fP(Height,"ID50B8559A-0000 ...	2	Number
P2	=P1+"*" +100	2*100.00	Distance list

10. In the **Custom component browser**, browse for **Bolt group distance x**.
11. Right-click **Bolt group distance x** and select **Add Equation**.
12. Enter P2 after the equal sign, and then press **Enter**.
13. Save the custom component.
14. Close the custom component editor.

When you now change the beam height, the number of bolt rows changes as well.

Example: Link variables to user-defined attributes

In this example, you will link parametric variables to the user-defined attributes of panels. You can then use the user-defined attributes in view filters to show or hide the panels in the model.



1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Click **Add** to create a new parametric variable.
3. Modify the variable as follows:
 - a. In the **Value type** list, select **Text**.

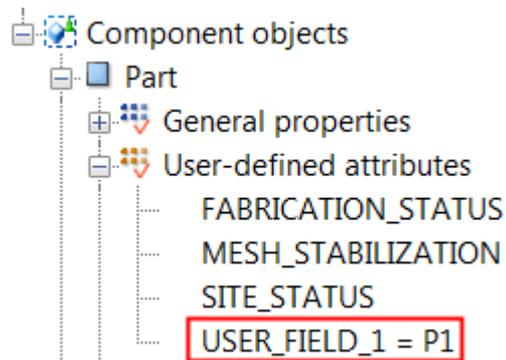
- b. In the **Formula** box, enter `Type1`.
 - c. In the **Label in dialog box** box, enter `Panel1`.
4. In the **Custom component browser**, browse for the user-defined attributes of the first panel.

You will link the **P1** variable to the **USER_FIELD_1** attribute. However, the attribute is not visible in the **Custom component browser**.
5. Make the user-defined attribute visible in the **Custom component browser**:
 - a. Double-click the first panel.

The panel properties open in the property pane.
 - b. Click **More**.

The dialog box for user-defined attributes opens.
 - c. Go to the **Parameters** tab.
 - d. Enter text in the **User field 1** box.
 - e. Click **Modify**.
6. In the **Custom component browser**, click **Refresh**.

USER_FIELD_1 appears under **User-defined attributes** in the **Custom component browser**.
7. Link **P1** to **USER_FIELD_1**.
 - a. Right-click **USER_FIELD_1** and select **Add Equation**.
 - b. Enter `P1` after the equal sign, and then press **Enter**.

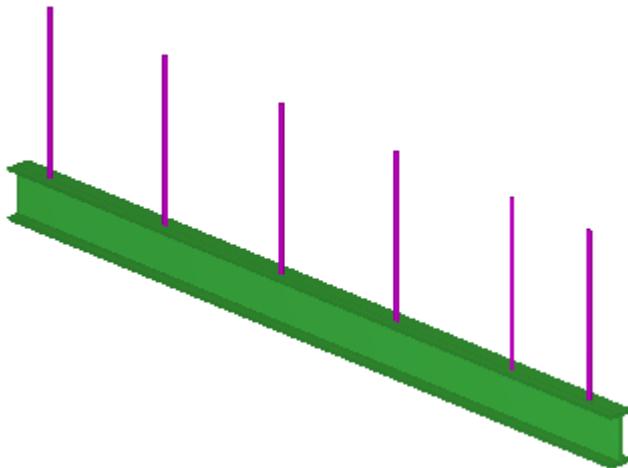


8. Create two new parametric variables and link them to the user-defined attributes of the other two panels.
9. Save the custom component.
10. Close the custom component editor.

You can now create a [view filter \(page 158\)](#) in the model to hide or show panels using the **User field 1** attribute and the **Formula** values you entered for the parametric variables in the filter.

Example: Calculate the number of handrail posts using a template attribute

In this example, you will create a variable formula that calculates the number of handrail posts based on the length template attribute of the beam. The handrail posts were created at both ends of the beam and one of them was copied with the **Array of objects (29)** component.



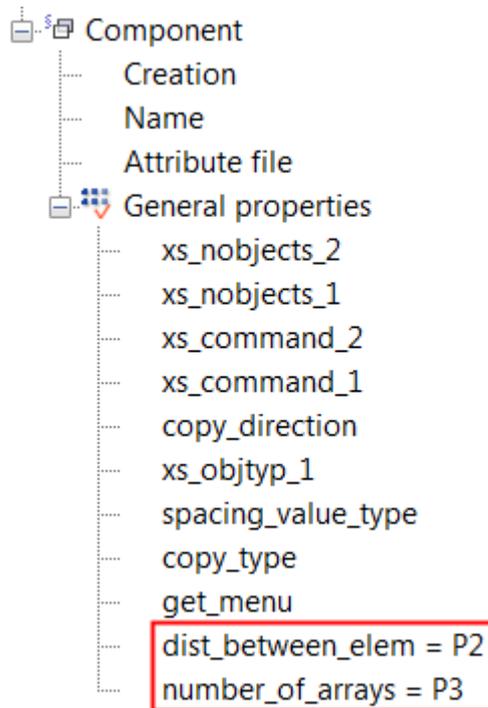
1. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
2. Create three new parametric variables by clicking **Add**.
3. Modify the variable **P1** as follows:
 - In the **Formula** box, enter 250.
 - In the **Label in dialog box** box, enter `End Distance`.
4. Modify the variable **P2** as follows:
 - In the **Formula** box, enter 900.
 - In the **Label in dialog box** box, enter `Spacing`.
5. Modify the variable **P3** as follows:
 - In the **Value type** box, select **Number**.
 - In the **Label in dialog box** box, enter `Number of Posts`.
6. Inquire the GUID of the beam:
 - a. On the ribbon, click **Inquire objects** .
 - b. Select the beam.
 - c. Check the GUID of the beam in the **Inquire Object** dialog box.

7. In the **Formula** box of the **P3** variable, enter

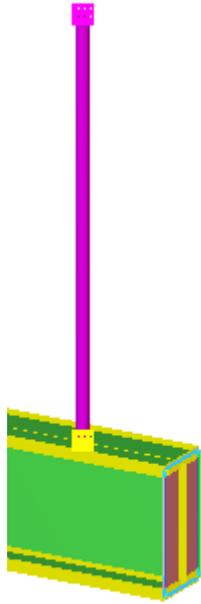
$$= (\text{fTp1}(\text{"LENGTH"}, \text{"ID50B8559A-0000-010B-3133-353432373038"}) - (\text{P1} * 2)) / \text{P2}.$$

$\text{fTp1}(\text{"LENGTH"}, \text{"ID50B8559A-0000-010B-3133-353432373038"})$ is the length template attribute of the beam and $\text{ID50B8559A-0000-010B-3133-353432373038}$ is the GUID of the beam.

The number of the posts is calculated as follows: First the end distances are subtracted from the beam length, and then the result is divided by the post spacing.
8. In the **Custom component browser**, link the variables **P2** and **P3** to the properties of **Array of objects (29)**.
 - a. Right-click **dist_between_elem** and select **Add Equation**.
 - b. Enter P2 after the equal sign, and then press **Enter**.
 - c. Right-click **number_of_arrays** and select **Add Equation**.
 - d. Enter P3 after the equal sign, and then press **Enter**.



9. Bind the first post to the beam end.
 - a. Select the post in the custom component view.
 - b. Hold down **Alt** and use area selection (from left to right) to select the post handles.
 - c. Right-click and select **Bind to Plane**.



10. Bind the last post to the other beam end by following the instructions in step 9.
11. Modify the distance variables as follows:
 - a. In the **Formula** box, enter =P1.
 - b. In the **Visibility** list, select **Hide**.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	250.00	250.00	Length	Parameter	Show	End Distance
P2	900.00	900.00	Length	Parameter	Show	Spacing
P3	=(fTPl(...	4	Number	Parameter	Show	Number of Posts
D1	=P1	250.00	Length	Distance	Hide	D1.COLUMN.BEAM
D2	=P1	250.00	Length	Distance	Hide	D2.COLUMN.BEAM
D3	=P1	250.00	Length	Distance	Hide	D3.COLUMN.BEAM
D4	=P1	250.00	Length	Distance	Hide	D4.COLUMN.BEAM

12. Save the custom component.
13. Close the custom component editor.

You can now change the spacing and the end distance of the handrail posts in the custom component dialog box. Tekla Structures calculates the number of posts based on the spacing, end distance, and length of the beam.

Example: Link an Excel spreadsheet to a custom component

In this example, you will link a parametric variable to an Excel spreadsheet. For example, you can use Excel spreadsheets to check connections.

1. Create an Excel spreadsheet.
The name of the spreadsheet file must be `component_"component_name".xls`. For example, `component_stiffener.xls` for a custom component whose name is `stiffener`.
2. Save the Excel spreadsheet in the model folder: `.. \<model> \exceldesign\`. Alternatively, you can save the spreadsheet in the folder defined with the `XS_EXTERNAL_EXCEL_DESIGN_PATH` advanced option.
3. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
4. Click **Add** to create a new parametric variable.
5. Modify the variable as follows:
 - a. In the **Value type** list, select **Yes/No**.
 - b. In the **Name** box, enter `use_externaldesign`.
 - c. In the **Label in dialog box** box, enter `Use external design`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
<code>use_externaldesign</code>	0	0	Yes/No	Parameter	Show	Use external design

6. Save the custom component.
7. Close the custom component editor.

The custom component dialog box now contains the **Use external design** option.

Examples: Rebar set modifiers in custom components

You can use rebar set modifiers in custom components. You can define the rebar set bar and modifier properties using parametric variables.

For each modifier property that you want to parametrize, the corresponding **Apply** property is also needed. By using the **Apply** property you can override an existing property value with an empty value. Clearing an existing value is not possible without the **Apply** property.

Example: Define the class and size of rebar set bars using a property modifier

In this example, you will use a rebar set property modifier for defining the class and size of certain rebar set bars in a custom component. You will define

the class and size using parametric variables and modifier-specific **Apply** properties.

The parametric variable for class will be defined so that if the class is set to 0, the class value is not applied, but the original class of the rebar set is used.

1. In the model, select a [previously created custom component \(page 747\)](#) that contains a rebar set and a property modifier.

NOTE Custom parts do not have a component symbol in the model.

To select custom components, ensure that the **Select**

components  selection switch is active.

-
2. Right-click and select **Edit Custom Component**.

3. In the custom component editor, click the **Display variables** button .

4. In the **Variables** dialog box, create and define parametric variables as follows:

- a. Click **Add** three times to create three new parametric variables.

The variable names will be P1, P2, and P3.

- b. Modify the variable **P1** for class number input as follows:

- In the **Value type** list, select **Number**.
- In the **Label in dialog box** box, enter `Class`.

- c. Modify the variable **P2** for **Apply** property control as follows:

- In the **Formula** box, enter `=if (P1==0) then 0 else 1 endif`.

This means that if P1 (**Class**) is set to 0, the class property is not applied when you use the custom component. If P1 is set to any other value, then the class property is applied.

- In the **Value type** list, select **Yes/No**.
- In the **Visibility** list, select **Hide**.

This means that the variable **P2** will not be visible in the custom component dialog box.

- In the **Label in dialog box** box, enter `Apply class`.

- d. Modify the variable **P3** for bar size input as follows:

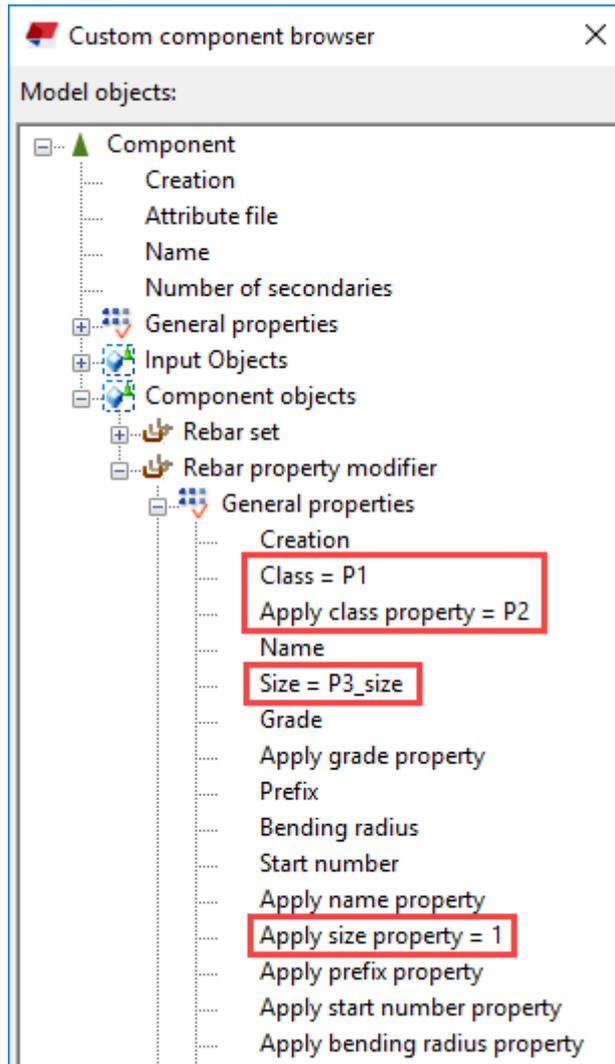
- In the **Name** box, change the name to `P3_size`.
- In the **Value type** list, select **Rebar size**.
- In the **Label in dialog box** box, enter `Bar size`.
- In the **Formula** box, replace zero with a valid bar size value.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Number	Parameter	Show	Class
P2	=if (P1==0) then 0 else 1 endif	0	Yes/No	Parameter	Hide	Apply class
P3_size	12	12	Rebar size	Parameter	Show	Bar size

5. In the **Custom component browser**, link the parametric variables to the property modifier properties:
 - a. Browse for **Component objects** --> **Rebar property modifier** --> **General properties**.
 - b. Right-click **Class**, select **Add Equation**, enter P1 after the equal sign (=), and then press **Enter**.

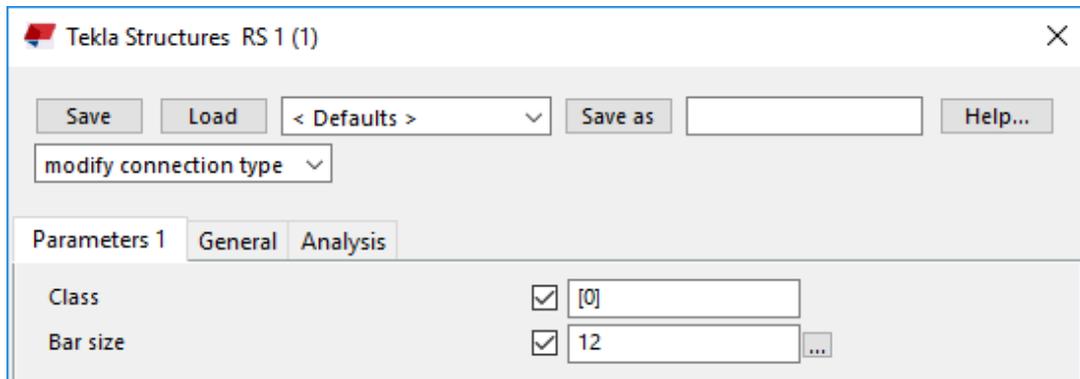
Similarly, link the other variables and properties as follows:

- **Apply class property** = P2
- **Size** = P3_size
- **Apply size property** = 1



6. [Save and close \(page 823\)](#) the modified custom component.

Now you have the following properties available in the custom component's dialog box and you can modify the class and size of those rebar set bars that are affected by the property modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to [modify \(page 761\)](#) it in the custom component editor.

Example: Create and modify rebar hooks using an end detail modifier

In this example, you will use a rebar set end detail modifier for creating hooks at certain rebar set bar ends in a custom component. You will define the hook properties using parametric variables and modifier-specific **Apply** properties.

1. In the model, select a [previously created custom component \(page 747\)](#) that contains a rebar set and an end detail modifier.

NOTE Custom parts do not have a component symbol in the model.

To select custom components, ensure that the **Select**

components  selection switch is active.

2. Right-click and select **Edit Custom Component**.
3. In the custom component editor, click the **Display variables** button .
4. In the **Variables** dialog box, create and define parametric variables as follows:
 - a. Click **Add** four times to create four new parametric variables. The variable names will be P1, P2, P3, and P4.
 - b. Modify the variable **P1** for hook type input as follows:
 - In the **Value type** list, select **Rebar hook type**.
 - In the **Label in dialog box** box, enter `Hook type`.

- In the **Formula** box, enter 4 for a custom hook.

The different hook types are identified with numbers: 1 = 90 degree hook, 2 = 135 degree hook, 3 = 180 degree hook, 4 = custom hook.

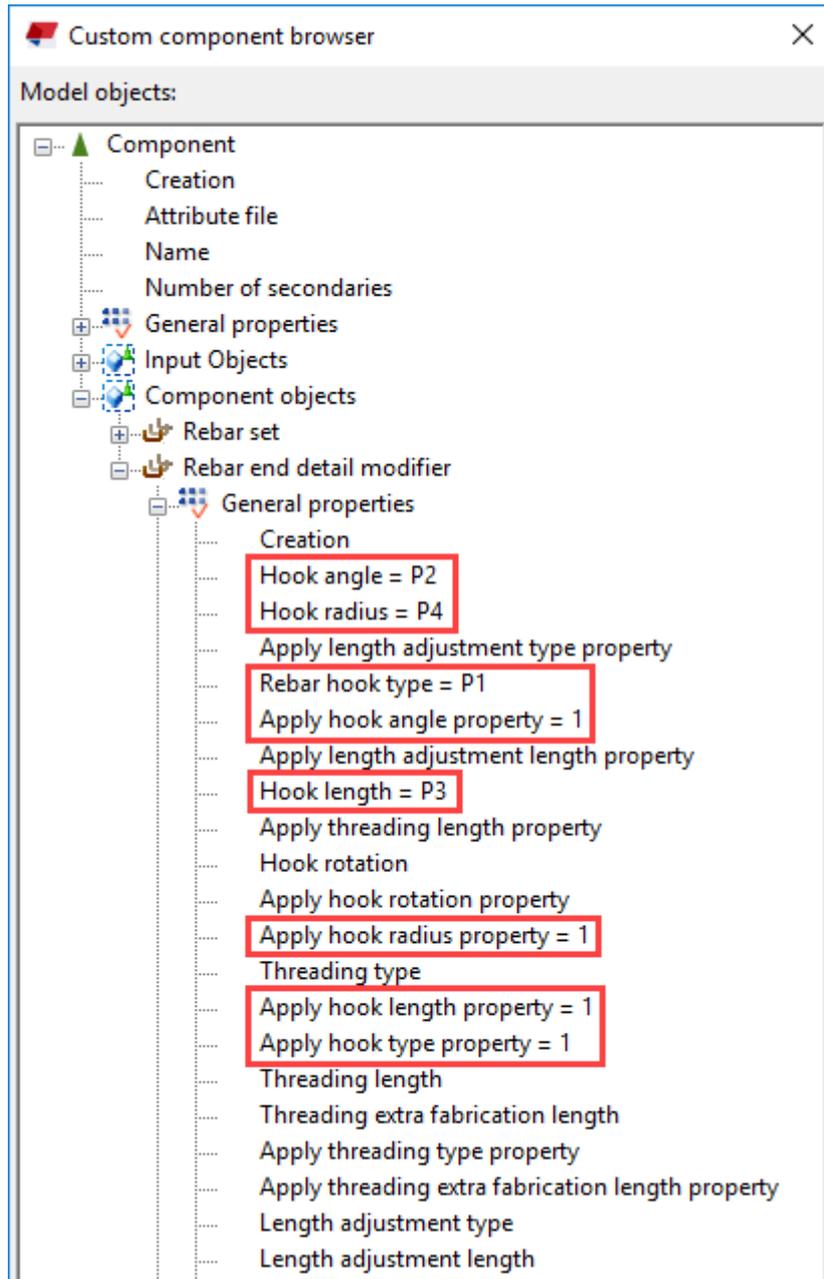
- Modify the variable **P2** for hook angle input as follows:
 - In the **Value type** list, select **Number**.
Note that although **Angle** is available as a value type, the **Number** option must be used for the hook angle.
 - In the **Label in dialog box** box, enter `Custom hook angle`.
- Modify the variable **P3** for hook length input as follows:
 - In the **Value type** list, select **Number**.
 - In the **Label in dialog box** box, enter `Custom hook length`.
- Modify the variable **P4** for hook radius input as follows:
 - In the **Value type** list, select **Number**.
 - In the **Label in dialog box** box, enter `Custom hook radius`.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	4	4	Rebar hook type	Parameter	Show	Hook type
P2	0	0	Number	Parameter	Show	Custom hook angle
P3	0	0	Number	Parameter	Show	Custom hook length
P4	0	0	Number	Parameter	Show	Custom hook radius

- In the **Custom component browser**, link the parametric variables to the end detail modifier properties:
 - Browse for **Component objects --> Rebar end detail modifier --> General properties**.
 - Right-click **Hook angle**, select **Add Equation**, enter `P2` after the equal sign (=), and then press **Enter**.

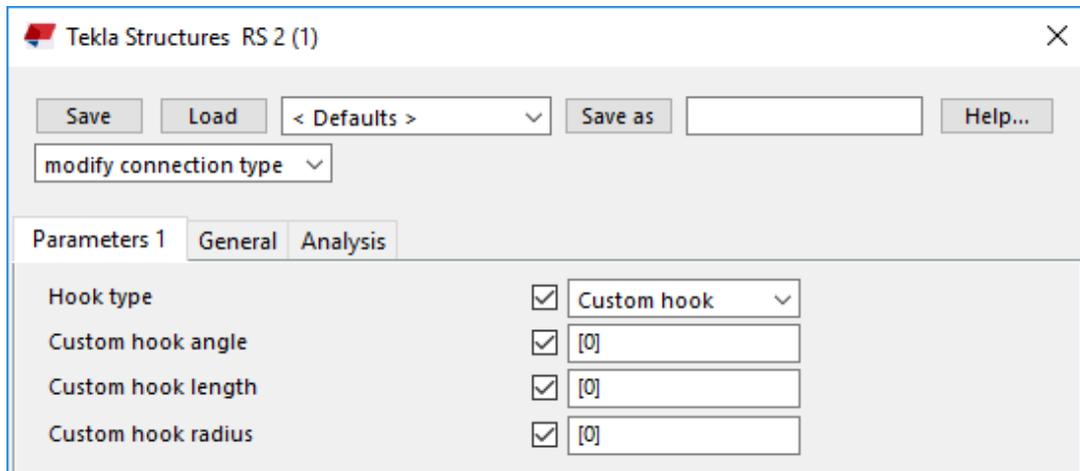
Similarly, link the other variables and properties as follows:

- **Hook radius** = `P4`
- **Rebar hook type** = `P1`
- **Apply hook angle property** = `1`
- **Hook length** = `P3`
- **Apply hook radius property** = `1`
- **Apply hook length property** = `1`
- **Apply hook type property** = `1`



6. [Save and close \(page 823\)](#) the modified custom component.

Now you have the following properties available in the custom component's dialog box and you can modify the hooks of those rebar set bars that are affected by the end detail modifier:



You can use the component in locations similar to where the component was originally created. This component is not adaptive and Tekla Structures does not adjust the component dimensions to suit any changes in the model. To make the custom component adaptive, you need to [modify \(page 761\)](#) it in the custom component editor.

8.8 Save a custom component

After modifying a custom component, save the changes.

To	Do this
Save changes to all copies of the custom component	<ol style="list-style-type: none"> In the custom component editor, click the Save component button . In the Save confirmation dialog box, click Yes. Tekla Structures saves the changes and applies them to all copies of the custom component in the model.
Save the component with a new name	<ol style="list-style-type: none"> In the custom component editor, click the Save with new name button . Enter a new name for the component.
Save and close the component	<ol style="list-style-type: none"> In the custom component editor, click the Close button .

To	Do this
	2. In the Close custom component editor message, click Yes . If you click No , the custom component editor closes without saving the changes.

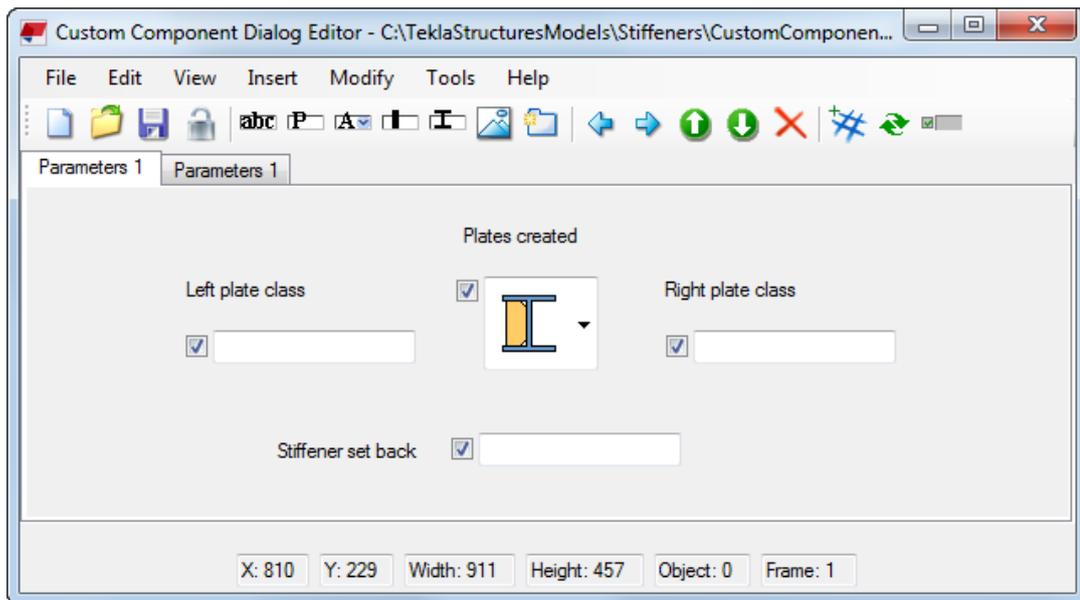
See also

[Tips for sharing custom components \(page 882\)](#)

8.9 Modify the dialog box of a custom component

Tekla Structures automatically creates a dialog box for each custom component you define. You can customize the dialog box by using the **Custom Component Dialog Editor** tool.

To open the dialog editor, select a custom component in the model, right-click, and select **Edit Custom Component Dialog Box**.



To	Do this
View and edit object properties	1. Select a dialog box element. For example, a text box. 2. Click Modify --> Properties . Now you can view and edit the current properties of the dialog box element. For example, you can check that you have the

To	Do this
	<p>correct text box under each label in the dialog box.</p> <p>Alternatively, you can double-click the dialog box element. If the dialog box element will not open for viewing and editing, try double-clicking the space right underneath the check box:</p> 
Add a dialog box element	<p>Click Insert and select a suitable element from the list. The options are:</p> <ul style="list-style-type: none"> • Tab Page: add a new tab • Label: add a label for a text box or list • Parameter: add a text box • Attribute: add a list • Part: add some basic part properties • Profile: add some basic profile properties • Picture: add an illustrative image of the custom component
Add an image	<ol style="list-style-type: none"> 1. Click Insert --> Picture to show the contents of the Image Folder set in Tools --> Options. 2. Select an image. The image must be in the bitmap (.bmp) format. 3. Click Open. 4. Drag the image to the desired location.
Add a tab	<ol style="list-style-type: none"> 1. Click Insert --> Tab Page . 2. Double-click the new tab. 3. Enter a new name, and then press Enter.

To	Do this
	<p>NOTE Each tab may contain up to 25 fields. If you have more than 25 fields visible, Tekla Structures automatically creates another tab.</p>
Show or hide the pixel grid	<p>Click .</p> <p>Tekla Structures displays a pixel grid that makes it easier to align elements in the dialog box.</p>
Move a dialog box element	<p>Drag the dialog box element to a new location.</p> <p>You can also use the keyboard shortcuts Ctrl+X (cut), Ctrl+C (copy), and Ctrl+V (paste). For example, to move a dialog box element to another tab: select the dialog box element, press Ctrl+X, go to another tab, and press Ctrl+V.</p>
Select multiple dialog box elements	<p>Hold down the Ctrl key and click the dialog box elements, or use area selection.</p>
Rename a tab or text box label	<ol style="list-style-type: none"> 1. Double-click the tab or text box label. 2. Type a new name. 3. Press Enter.
Remove a dialog box element	<ol style="list-style-type: none"> 1. Select the dialog box element you want to remove. 2. Press Delete.
Remove a tab	<ol style="list-style-type: none"> 1. Select the tab. 2. Right-click and select Delete.
Add images to a list	<ol style="list-style-type: none"> 1. Select the list element. 2. Click Modify --> Properties. 3. Click Edit Values. 4. Click Browse Add. 5. Select the image you want to use and click Open.

To	Do this
	6. Repeat steps 4–5 for any other images you want to use. 7. Click OK to save the changes.
Save the changes	Click File --> Save .

See also

[Dialog editor settings \(page 857\)](#)

[Modify custom component input files manually \(page 827\)](#)

[Example: Modify the dialog box of a stiffener detail \(page 835\)](#)

Custom component input files

Each custom component has an input file that defines the contents of the custom component's dialog box.

When you create a new custom component, Tekla Structures automatically creates an input file for the component. The input file is located in the `\CustomComponentDialogFiles` folder under the model folder. The input file has the same name as the custom component, and the file name extension is `.inp`.

When you [modify a custom component \(page 761\)](#), you will lose any changes you have made to the input file. However, when you modify the custom component, Tekla Structures automatically creates a backup copy of the input file. The backup copy has the file name extension `.inp_bak`, and it is located in the `\CustomComponentDialogFiles` folder under the model folder. Tekla Structures displays a notification when the backup file is created.

See also

[Lock or unlock the custom component input file \(page 830\)](#)

Modify custom component input files manually

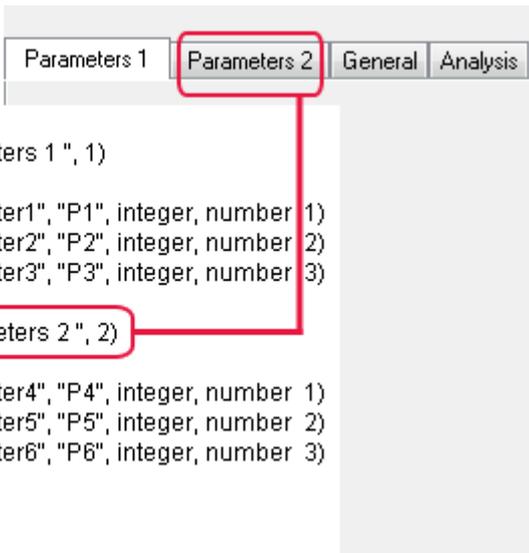
If you are an advanced user, you can modify the dialog box input (`.inp`) files manually in a text editor. Be careful when modifying an input file, as errors may cause the dialog box to disappear.

Note that the **General** tab is reserved for predefined general properties. You cannot rename the **General** tab or add more parameters to it.

Add new tabs

1. Open the `.inp` file in a text editor.

2. Add a new tab definition, as shown below:



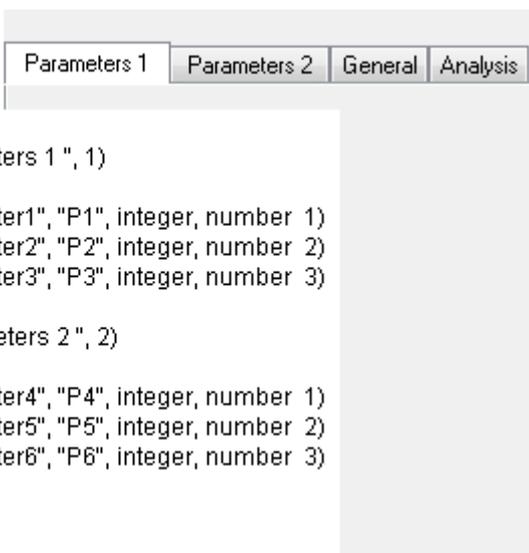
```
{
  tab_page("", " Parameters 1 ", 1)
  {
    parameter("Parameter1", "P1", integer, number 1)
    parameter("Parameter2", "P2", integer, number 2)
    parameter("Parameter3", "P3", integer, number 3)
  }
  tab_page("", " Parameters 2 ", 2)
  {
    parameter("Parameter4", "P4", integer, number 1)
    parameter("Parameter5", "P5", integer, number 2)
    parameter("Parameter6", "P6", integer, number 3)
  }
}
```

3. Save the `.inp` file.

NOTE The fourth tab is reserved for the **General** properties, so you cannot add your own parameters to it.

Add text boxes

1. Open the `.inp` file in a text editor.
2. Add `parameter` elements and enclose them in curly brackets, as shown below:

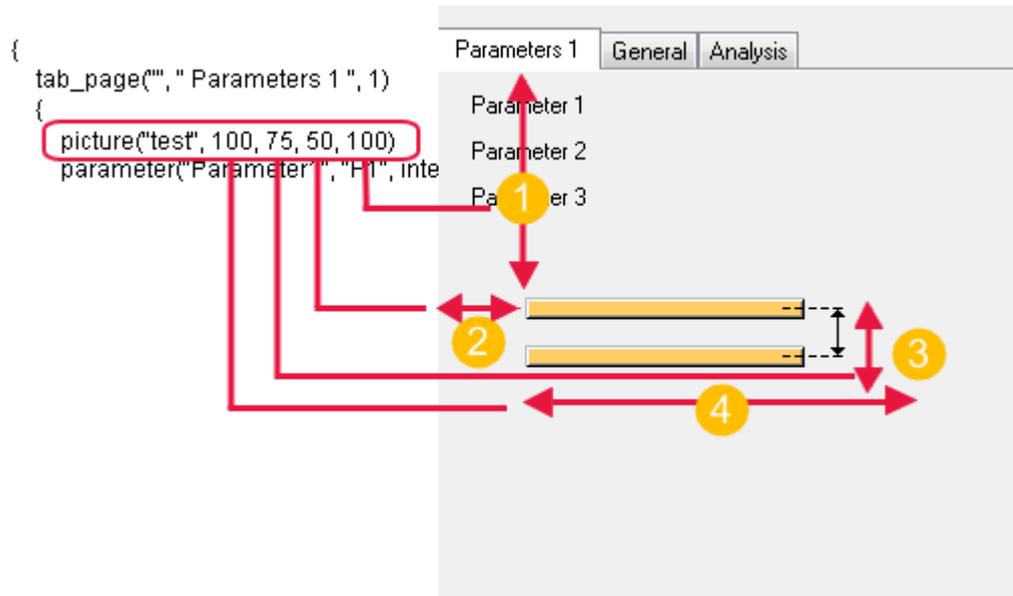


```
{
  tab_page("", " Parameters 1 ", 1)
  {
    parameter("Parameter1", "P1", integer, number 1)
    parameter("Parameter2", "P2", integer, number 2)
    parameter("Parameter3", "P3", integer, number 3)
  }
  tab_page("", " Parameters 2 ", 2)
  {
    parameter("Parameter4", "P4", integer, number 1)
    parameter("Parameter5", "P5", integer, number 2)
    parameter("Parameter6", "P6", integer, number 3)
  }
}
```

3. Save the `.inp` file.

Add images

1. Create an image and save it in bitmap (.bmp) format in the ..\Tekla Structures\- 2. Open the .inp file in a text editor.
- 3. Add an image definition, as shown below:



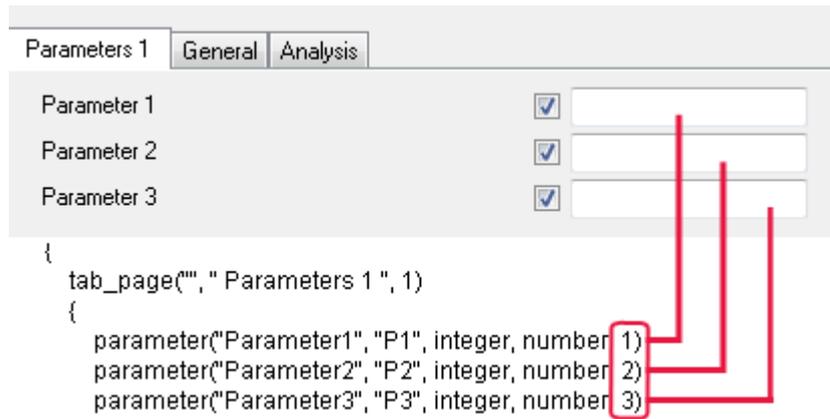
- (1) $y = 100$
- (2) $x = 50$
- (3) height = 75
- (4) width = 100

4. Save the .inp file.

Change the order of boxes

1. Open the .inp file in a text editor.
2. Change the last number in the parameter definition.

The boxes are listed from top to bottom, as shown below:



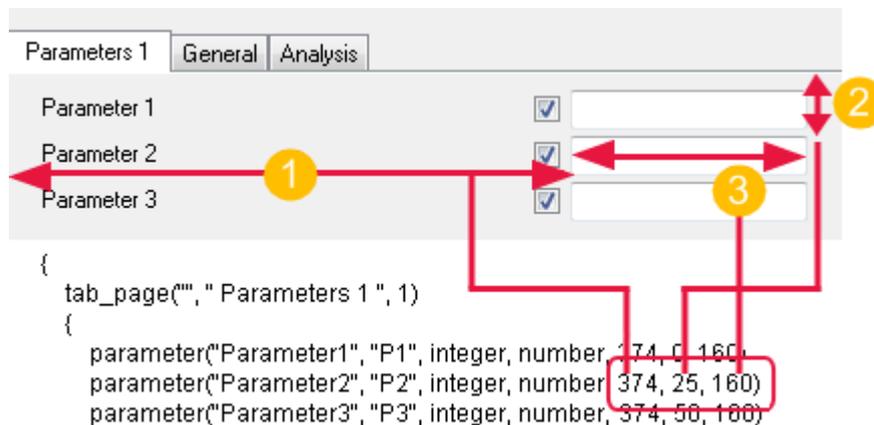
3. Save the `.inp` file.

Change the location of boxes

You can define an exact location for each text box.

1. Open the `.inp` file in a text editor.
2. Define the exact location of the box using three values: the x coordinate, y coordinate, and width of the box.

For example:



(1) $x = 374$

(2) $y = 25$

(3) width = 160

3. Save the `.inp` file.

Lock or unlock the custom component input file

You can lock the custom component's input file to prevent accidental modifications. If the file is unlocked, and someone else updates the custom component in the custom component editor, all your modifications to the dialog box will be lost.

1. In the model, select the custom component whose [input file \(page 827\)](#) you want to lock or unlock.
2. Right-click and select **Edit Custom Component Dialog Box**.
3. In the dialog editor, click the **Lock/Unlock** button .

If someone modifies the custom component in the custom component editor when the `.inp` file is locked, the `.inp` file will not be updated. You can still modify the dialog box in the **Custom Component Dialog Editor** even if the `.inp` file is locked.

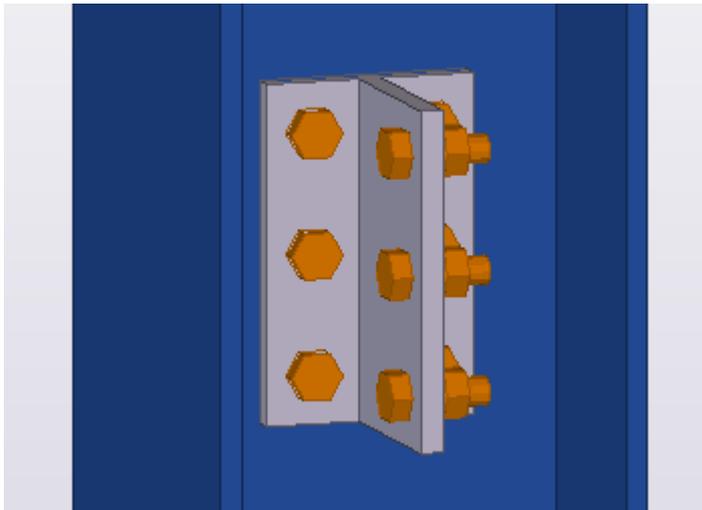
Example: Create a toggle group

Create a toggle group if you want to add a group of check boxes without labels.

In this example, we will add a check box for each bolt group in a custom component. When the component is used in a model, the user can select which bolts to create by selecting the desired check boxes.

1. Create a custom component that contains bolts.

For example, we created a custom tee connection that creates one bolt group and three single bolts:



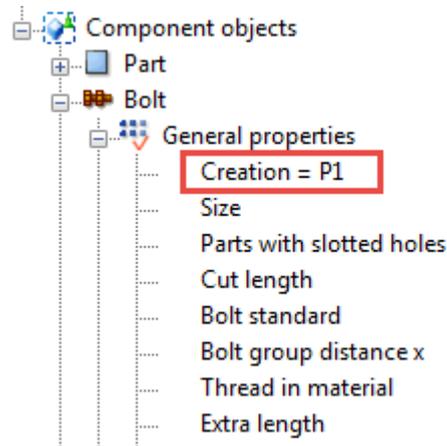
2. Create parametric variables that control the creation of bolts.

With toggle groups, the **Value type** of these variables must be **Yes/No**. For example, we created three variables P1, P2, and P3, one for each single bolt in the custom tee connection.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
P1	0	0	Yes/No	Parameter	Show	Parameter1
P2	0	0	Yes/No	Parameter	Show	Parameter2
P3	0	0	Yes/No	Parameter	Show	Parameter3

3. [Link the variables \(page 777\)](#) to the **Creation** property of the bolts.

For example, link the variable P1 to the **Creation** property of the first bolt, the variable P2 to the **Creation** property of the second bolt, and so on.



4. Save the custom component.
5. In the model, click **File --> Open the model folder** to open the current model folder.
6. Go to the `\CustomComponentDialogFiles` folder.
7. Open the `.inp` file in a text editor.
8. [Add an image definition \(page 829\)](#).

For example:

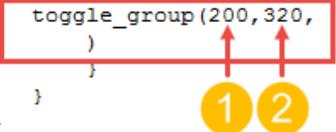
```
page("TeklaStructures","")
{
  detail(1, "tee")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      picture("CustomTee",100,100,200,100) /*Bolts*/
    }
  }
}
```

If you use a custom image, save it in bitmap (.bmp) format in the ..
\TeklaStructures\

9. Add a `toggle_group` element to define the toggle group origin, that is, the position of the toggle group in the custom component dialog box.

Use the x and y coordinate values to define the position. For example:

```
page("TeklaStructures","")
{
  detail(1, "tee")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      picture("CustomTee",100,100,200,100) /*Bolts*/
      toggle_group(200,320,
    )
  }
}
```



(1) x = 200

(2) y = 320

10. Within the `toggle_group` element, add a line for each check box you want to add.

Use the same parametric variables that you created in step 2.

```

page("TeklaStructures","")
{
  detail(1, "tee")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      picture("CustomTee",100,100,200,100) /*Bolts*/
      toggle_group(200,320,
        "P1", 160, -165, "0",
        "P2", 160, -135, "0",
        "P3", 160, -105, "0")
    }
  }
}

```

The two numeric values after the variable name are offsets from the toggle group origin. For example, the first definition "P1", 160, -165, "0" means that the check box for the variable P1 is located 160 steps right and 165 steps up from the toggle group origin.

	Direction	Negative values	Positive values
X		left	right
Y		up	down

```

page("TeklaStructures","")
{
  detail(1, "tee")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      picture("CustomTee",100,100,200,100) /*Bolts*/
      toggle_group(200,320,
        "P1", 160, -165, "0",
        "P2", 160, -135, "0",
        "P3", 160, -105, "0")
    }
  }
}

```

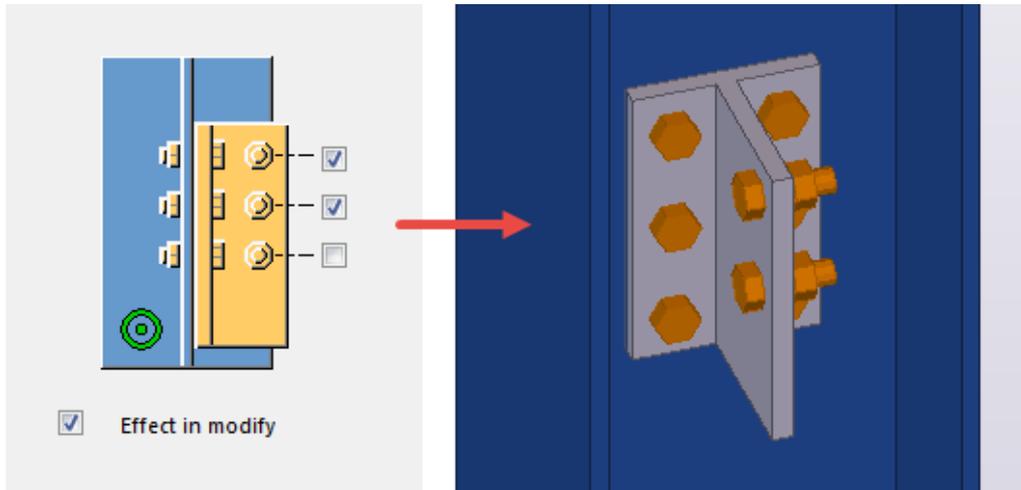
(1) offset in X direction

(2) offset in Y direction

11. Save the .inp file.

12. Close and reopen the model for the change to take effect.

Now when you select and clear check boxes in the dialog box, the number of bolts changes accordingly in the model. For example:

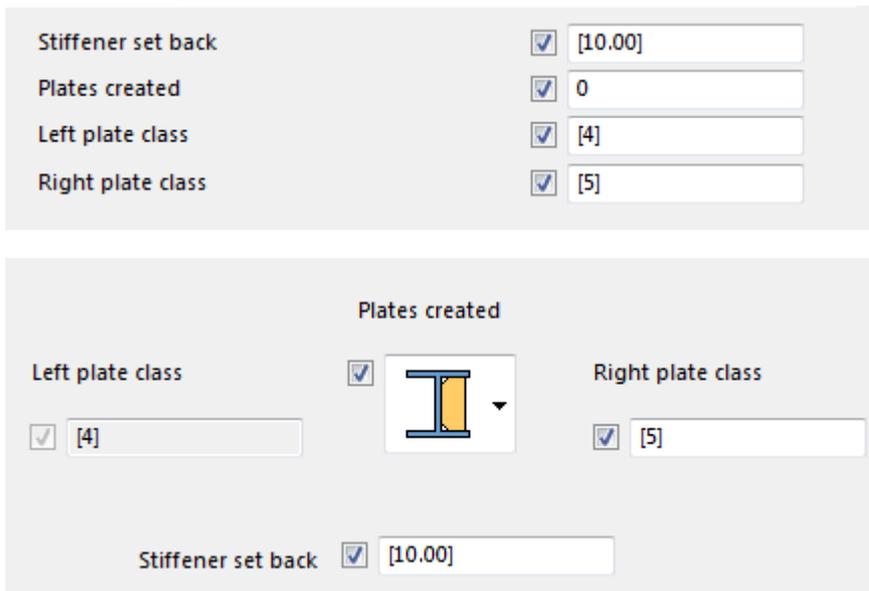


NOTE Tekla Structures automatically adds the **Effect in modify** label and check box for each toggle group you create.

Example: Modify the dialog box of a stiffener detail

In this example, we will edit the dialog box of a custom stiffener detail to make it easier to adjust the settings later on.

When we begin, the dialog box looks like the following:



Do this

1. Create a custom stiffener detail with all the needed variables that control the creation of stiffener plates.
2. Add a list with images.
3. Arrange text boxes and labels.
4. Dim unavailable options.

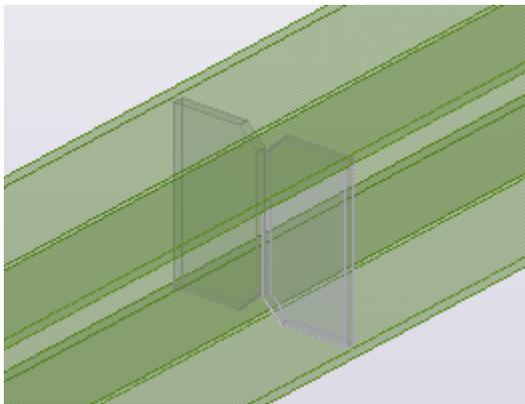
Example: Create a custom stiffener detail with variables

In this example, we will create a stiffener detail with variables that control the shape and position of the stiffeners.

Create a basic stiffener detail

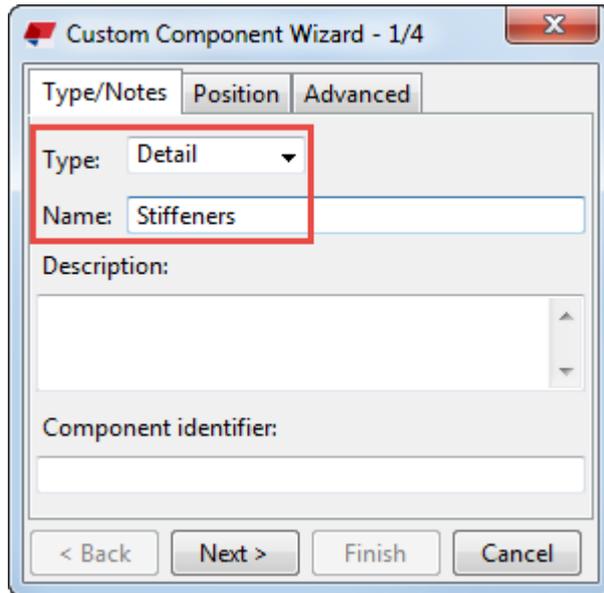
In this example, we will create a basic stiffener detail.

1. Create a beam with two stiffeners.

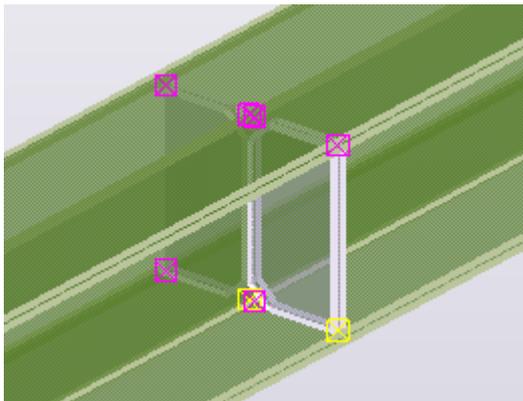


TIP To create the stiffeners, you can use the **Stiffeners (1003)** component and then explode the component.

2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. Click the **Access advanced features** button  and select **Define custom component**.
The **Custom Component Wizard** dialog box opens.
4. In the **Type** list, select **Detail**.
5. In the **Name** box, type `Stiffeners`.

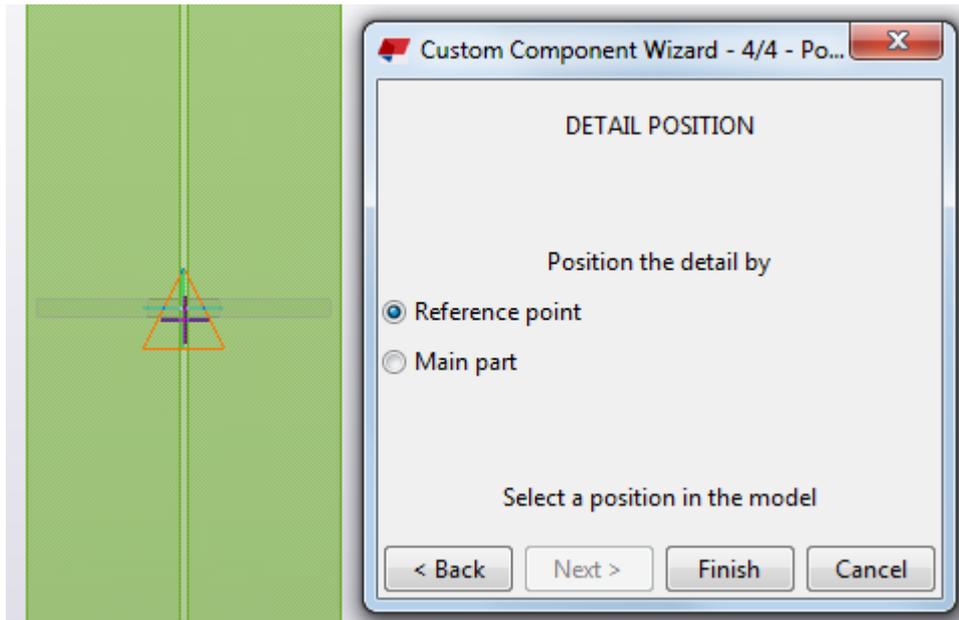


6. Click **Next**.
7. Select the stiffeners and the beam as the objects that form the custom component.



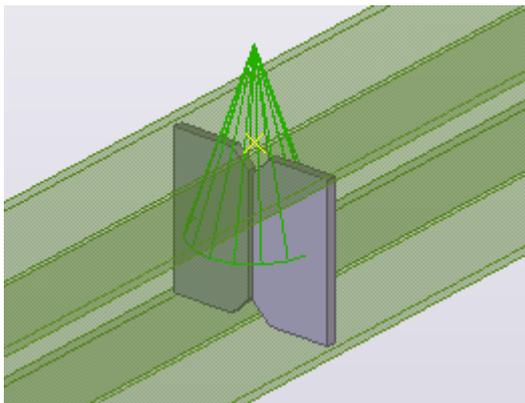
8. Click **Next**.
9. Select the beam as the main part.
10. Click **Next**.
11. Select the middle point of the beam as the reference point.

TIP Switch to the [plane view \(page 43\)](#) to select the middle point more easily.



12. Click **Finish** to finish creating the stiffener detail.

Tekla Structures displays a component symbol for the new custom component and adds the stiffener detail to the component catalog.



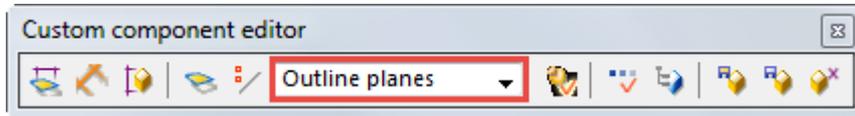
Create bindings to control the stiffener shape

In this example, we will bind custom component handles to a plane to control the shape of the stiffeners.

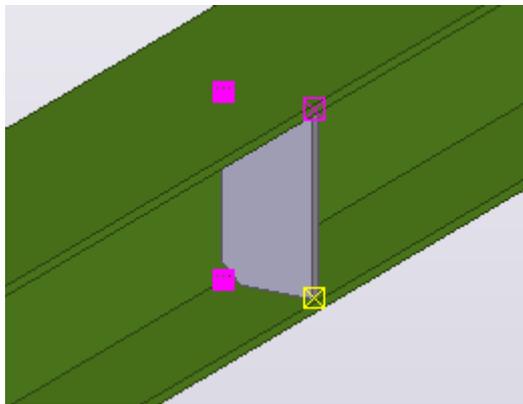
1. Open the stiffener detail in the custom component editor.
 - a. Right-click the custom component in the model.
 - b. Select **Edit Custom Component**.

The custom component editor opens showing the custom component editor toolbar, the component browser, and four views of the custom component.

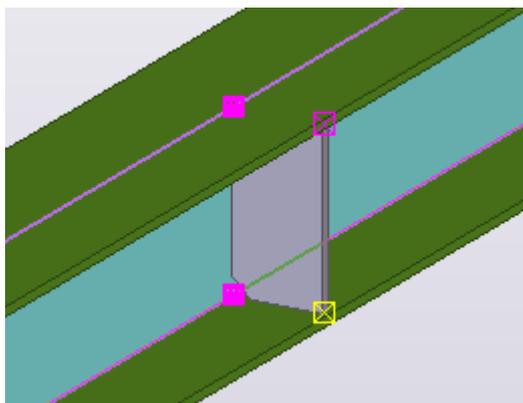
2. On the **View** tab, click **Rendering** --> **Parts rendered**.
Part surfaces and available planes can be selected only when they are rendered.
3. On the custom component editor toolbar, select **Outline planes** from the list.



4. In the custom component editor, select the stiffener on the right.
5. Bind the two inside handles of the stiffener to the beam web.
 - a. Select the two handles next to the beam web.

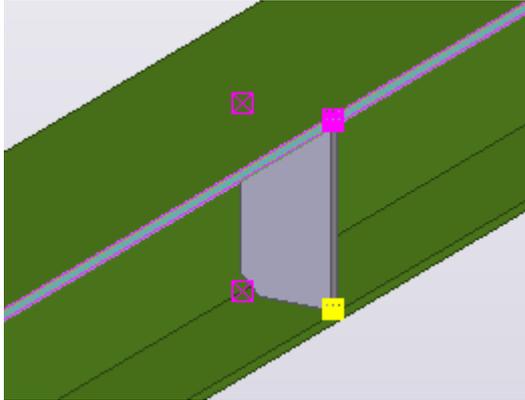


- b. Right-click and select **Bind to Plane**.
 - c. Move the pointer over the face of the web to highlight it.



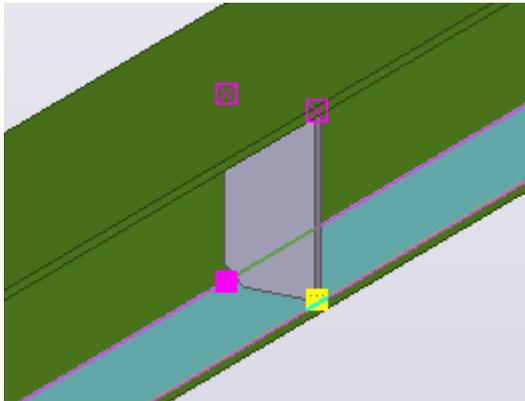
- d. Click the web to bind the handles.
6. Bind the two outside handles of the stiffener to the face of the top flange.

Use the same method as in step 5.



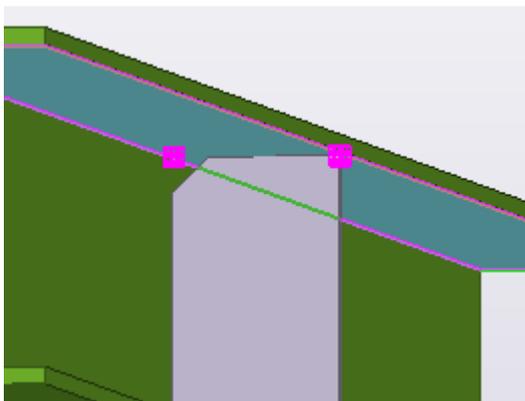
7. Bind the two bottom handles of the stiffener to the inside face of the bottom flange.

Use the same method as in step 5.



8. Bind the two top handles of the stiffener to the inside face of the top flange.

Use the same method as in step 5.



9. Repeat steps 4 -11 for the stiffener on the left.

10. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
11. Click **Add** to create a new parametric variable P1.
12. Modify the variable P1 as follows:
 - a. In the **Formula** box, enter 10.
 - b. In the **Label in dialog box** box, enter `Stiffener set back`.
13. In the **Formula** box, enter `=P1` for all variables that got values during the binding of the handles.

For example:

Name	Formula	Value	Value type
D1	0.00	0.00	Length
D2	0.00	0.00	Length
D3	10.00	10.00	Length
D4	10.00	10.00	Length

The variable P1 now controls the distances of these variables.

14. In the **Visibility** list, set the variable P1 to **Show** and the other variables to **Hide**.

We have now created distance variables that control the stiffener shape.

Name	Formula	Value	Value type	Variable type	Visibility	Label in dialog box
D1	0.00	0.00	Length	Distance	Hide	D1.PLATE.Web right plane
D2	0.00	0.00	Length	Distance	Hide	D2.PLATE.Web right plane
D3	=P1	10.00	Length	Distance	Hide	D3.PLATE.Upper flange right plane
D4	=P1	10.00	Length	Distance	Hide	D4.PLATE.Upper flange right plane
D5	0.00	0.00	Length	Distance	Hide	D5.PLATE.Lower flange top right plane
D6	0.00	0.00	Length	Distance	Hide	D6.PLATE.Lower flange top right plane
D7	0.00	0.00	Length	Distance	Hide	D7.PLATE.Upper flange bottom right plane
D8	0.00	0.00	Length	Distance	Hide	D8.PLATE.Upper flange bottom right plane
D9	0.00	0.00	Length	Distance	Hide	D9.PLATE.Web left plane
D10	0.00	0.00	Length	Distance	Hide	D10.PLATE.Web left plane
D11	=P1	10.00	Length	Distance	Hide	D11.PLATE.Upper flange left plane
D12	=P1	10.00	Length	Distance	Hide	D12.PLATE.Upper flange left plane
D13	0.00	0.00	Length	Distance	Hide	D13.PLATE.Lower flange top left plane
D14	0.00	0.00	Length	Distance	Hide	D14.PLATE.Lower flange top left plane
D15	0.00	0.00	Length	Distance	Hide	D15.PLATE.Upper flange bottom left plane
D16	0.00	0.00	Length	Distance	Hide	D16.PLATE.Upper flange bottom left plane
P1	10.00	10.00	Length	Parameter	Show	Stiffener set back

Create bindings to control the stiffener position

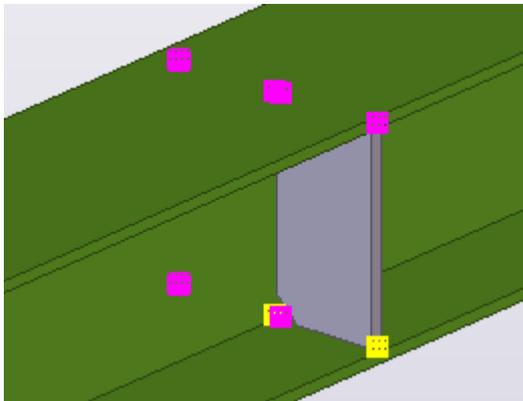
In this example, we will bind custom component handles to a plane to control the position of the stiffeners.

1. Open the stiffener detail in the custom component editor.
 - a. Right-click the custom component in the model.
 - b. Select **Edit Custom Component**.

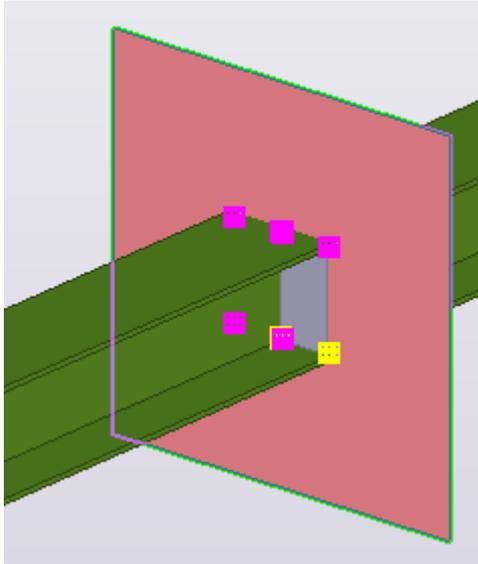
The custom component editor opens showing the custom component editor toolbar, the component browser, and four views of the custom component.
2. On the custom component editor toolbar, select **Component planes** from the list.



3. Select all the handles of both stiffeners.



4. Right-click and select **Bind to Plane**.
5. Bind the handles to the vertical component plane.



We have now created distance variables that control the position of the stiffeners.

Create variables to control the stiffener thickness

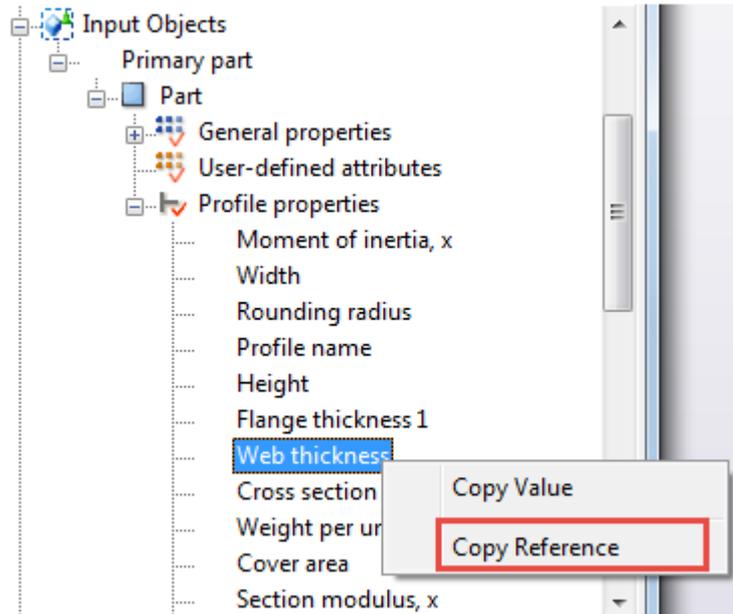
In this example, we will control the stiffener thickness so that it is one and a half times the web thickness rounded up to the nearest available plate thickness. The available thickness values are 10, 12, and 16 mm.

1. Open the stiffener detail in the custom component editor.
 - a. Right-click the custom component in the model.
 - b. Select **Edit Custom Component**.

The custom component editor opens showing the custom component editor toolbar, the component browser, and four views of the custom component.

2. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
3. Click **Add** to create a new parametric variable P2.
4. Modify the variable P2 as follows:
 - a. In the **Formula** box, enter $=1.5*$.
 - b. In the **Visibility** list, select **Hide**.
 - c. In the **Label in dialog box** box, enter `Plate calculation`.
5. Select the beam in the custom component editor to highlight the beam (primary part) in the custom component browser.
6. In the **Custom component browser**, select **Web thickness** of the primary part.

- Right-click and select **Copy Reference**.

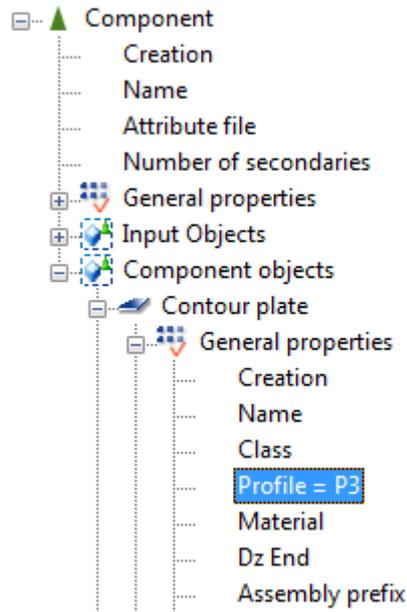


- Paste the reference value to **Formula** after $=1.5*$.

P2 $=1.5*FP(\text{Web thickness}, \text{"ID1B0F6981-F56C-4619-88FF-7D30F42A8F2E"})$ 12.75 Length

NOTE A reference function refers to the property of an object, such as the web thickness of a part. If the object property changes, so does the reference function value.

- Click **Add** to create a new parametric variable P3.
- Modify the variable P3 as follows:
 - In the **Value type** list, select **Number**.
 - In the **Formula** box, enter `=if (P2 < 12 && P2 > 10) then 12 else if (P2 > 12) then 16 else 10 endif endif.`
 This means that if P2 is less than 12 and greater than 10, the thickness is 12. If P2 is greater than 12, the thickness is 16. If none of these conditions are met, the thickness is 10.
- In the **Custom component browser**, link the variable P3 to the **Profile** property of the first contour plate.



12. Repeat step 11 for the second contour plate.

We have now created and linked all required variables that control the stiffener thickness according to the web thickness.

Create variables to control the creation of stiffener plates

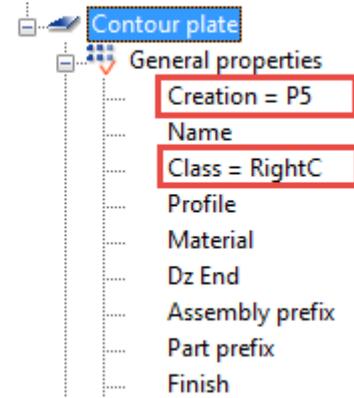
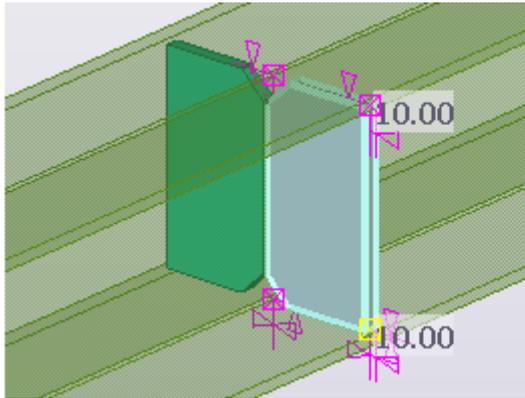
In this example, we will create five variables to control which stiffener plates are created and what is the class of the plates.

1. Open the stiffener detail in the custom component editor.
 - a. Right-click the custom component in the model.
 - b. Select **Edit Custom Component**.

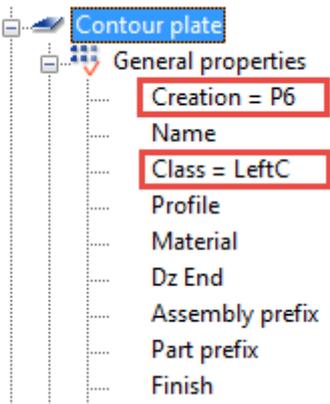
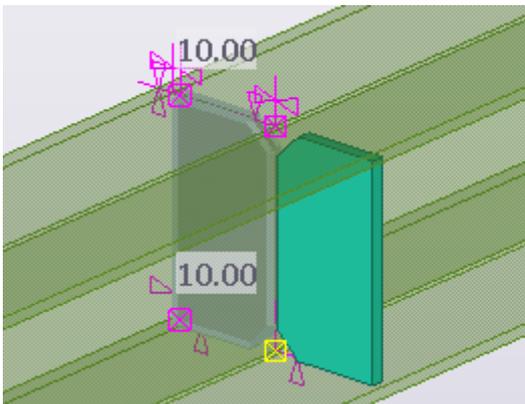
The custom component editor opens showing the custom component editor toolbar, the component browser, and four views of the custom component.

2. In the custom component editor, click the **Display variables** button . The **Variables** dialog box opens.
3. Click **Add** to create a new parametric variable P4.
4. Modify the variable P4 as follows:
 - a. In the **Formula** box, enter 2.
 - b. In the **Value type** list, select **Number**.
 - c. In the **Visibility** list, select **Show**.
 - d. In the **Label in dialog box** box, enter `Plates created`.
5. Click **Add** to create a new parametric variable P5.

6. Modify the variable P5 as follows:
 - a. In the **Formula** box, enter `=if P4==0 then 0 else 1 endif.`
 - b. In the **Value type** list, select **Yes/No**.
 - c. In the **Visibility** list, select **Hide**.
 - d. In the **Label in dialog box** box, enter `Do not create right.`
7. Click **Add** to create a new parametric variable P6.
8. Modify the variable P6 as follows:
 - a. In the **Formula** box, enter `=if P4==1 then 0 else 1 endif.`
 - b. In the **Value type** list, select **Yes/No**.
 - c. In the **Visibility** list, select **Hide**.
 - d. In the **Label in dialog box** box, enter `Do not create left.`
9. Click **Add** to create a new parametric variable P7.
10. Modify the variable P7 as follows:
 - a. Rename P7 as `LeftC`.
 - b. In the **Formula** box, enter `4`.
 - c. In the **Value type** list, select **Number**.
 - d. In the **Visibility** list, select **Show**.
 - e. In the **Label in dialog box** box, enter `Left plate class.`
11. Click **Add** to create a new parametric variable P8.
12. Modify the variable P8 as follows:
 - a. Rename P8 as `RightC`.
 - b. In the **Formula** box, enter `5`.
 - c. In the **Value type** list, select **Number**.
 - d. In the **Visibility** list, select **Show**.
 - e. In the **Label in dialog box** box, enter `Right plate class.`
13. In the **Custom component browser**, link the variables P5 and RightC to the right stiffener plate.



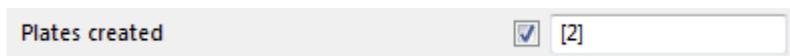
14. Link the variables P6 and LeftC to the left stiffener plate.



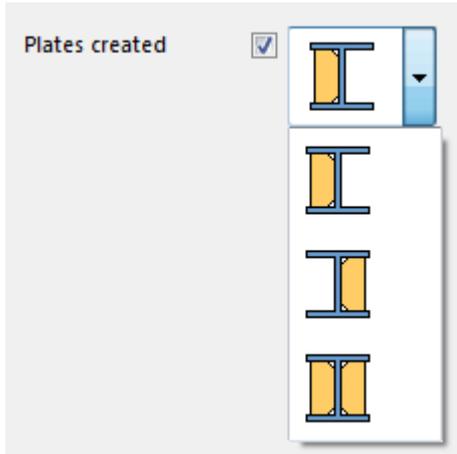
Example: Add a list with images

In this example, we will add an illustrative list in the stiffener dialog box. You can do this either in the custom component dialog editor or by editing the input (.inp) file manually.

When we begin, the dialog box has the text box shown below, and the user needs to know the values (0 is left, 1 is right, and 2 is both plates) that control the creation of stiffener plates.



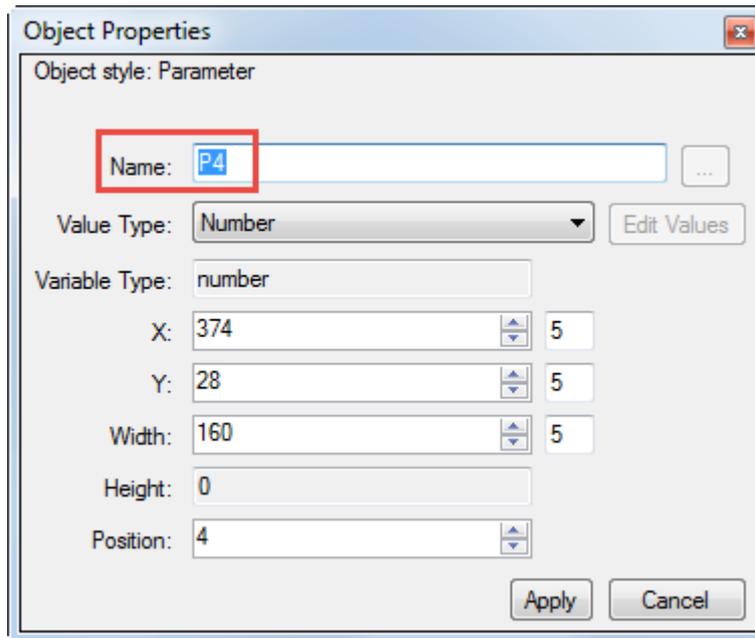
We will replace the text box with a list that is easier to use:



Add a list by using the dialog editor

1. [Create a custom stiffener detail \(page 836\)](#) with all the needed variables that control which stiffener plates are created.
In our example, the variable is called **Plates created**.
2. Open the stiffener dialog box for editing.
 - a. In the model, select the custom stiffener detail.
 - b. Right-click and select **Edit Custom Component Dialog Box**.
3. Check the name of the parametric variable that controls the plate creation.
 - a. In the dialog editor, double-click the **Plates created** box.
The **Object Properties** dialog box opens.
 - b. Check the name of the parametric variable.

In our example, the name is P4.



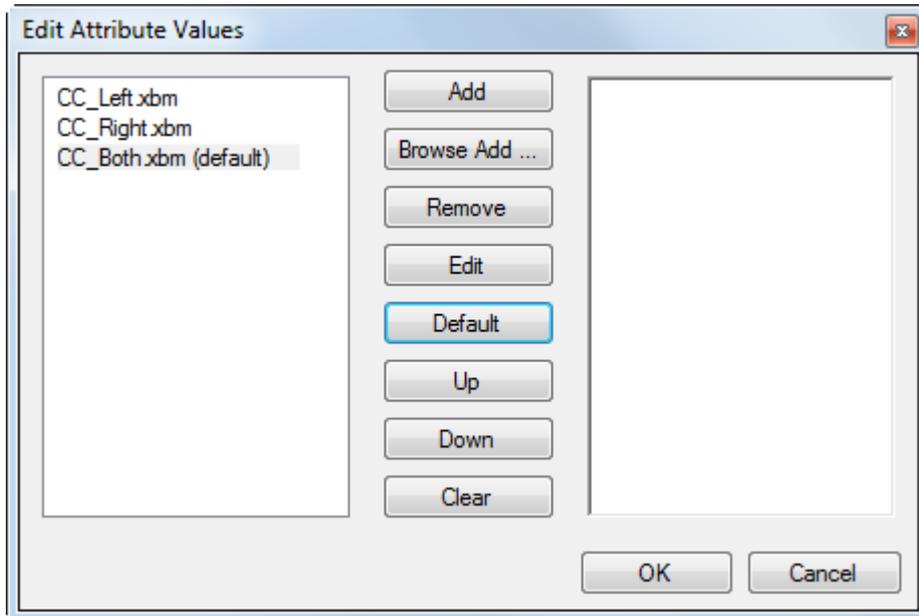
- c. Click **Cancel** to close the dialog box.
4. Select the **Plates created** text box and press **Delete**.
5. Click **Insert --> Attribute** to add a new attribute list.
6. Drag the attribute list to a suitable location, next to the **Plates created** label.
7. Select the attribute list and then click **Modify --> Properties** to edit its properties.
8. Enter P4 as the **Name** of the attribute.

Now the attribute list is linked to the parametric variable that controls the plate creation.

9. Click **Edit Values** to add the list items.
10. In the **Edit Attribute Values** dialog box, add an image for the left plate.
 - a. Click **Browse Add**.
 - b. Browse for a suitable image.

If you make new images, make sure they are in the bitmap (.bmp) format. Save the images in the ..\ProgramData\Trimble\Tekla Structures\<version>\Bitmaps folder.
 - c. Click **Open**.
11. Repeat step 9 to add an image for the right plate, and then for both plates.

- In the **Edit Attribute Values** dialog box, select the image of both plates and then click **Default** to make the attribute the default value.



- Click **OK**.
- Click **Apply** in the **Object Properties** dialog box, and then click **Cancel** to close the dialog box.
- In the dialog editor, click **File** --> **Save** to save the changes.
- Close and reopen the model for the change to take effect.

Add a list by editing the .inp file

- [Create a custom stiffener detail \(page 836\)](#) with all the needed variables that control which stiffener plates are created.
In our example, the variable is called **Plates created**.
- In the model, click **File** --> **Open the model folder** to open the current model folder.
- Go to the `\CustomComponentDialogFiles` folder.
- Open the `.inp` file in a text editor.

```

Stiffeners.inp - Notepad
File Edit Format View Help
page("TeklaStructures","")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      parameter("Stiffener set back", "P1", distance, number, 1)
      parameter("Plates created", "P4", integer, number, 2)
      parameter("Left plate class", "LeftC", integer, number, 3)
      parameter("Right plate class", "RightC", integer, number, 4)
    }
  }
}

```

- Remove the following line:

```
parameter("Plates created", "P4", integer, number, 2)
```

- Add a new **Plates created** attribute with the following settings:

```

page("TeklaStructures","")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      parameter("Stiffener set back", "P1", distance, number, 1)
      parameter("Left plate class", "LeftC", integer, number, 3)
      parameter("Right plate class", "RightC", integer, number, 4)
      attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
    }
  }
}

```

- Add a new P4 attribute with the following settings:

```

page("TeklaStructures","")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      parameter("Stiffener set back", "P1", distance, number, 1)
      parameter("Left plate class", "LeftC", integer, number, 3)
      parameter("Right plate class", "RightC", integer, number, 4)
      attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
      attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
      {
        value ("Left", 0)
        value ("Right", 0)
        value ("Both", 1)
      }
    }
  }
}

```

The list now contains three options, and **Both** is the default value. The list options are linked to the variable P4 that controls the creation of the stiffener plates.

8. Edit the line numbers so that there are no empty rows between the variables in the dialog box.

```

page("TeklaStructures", "")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      parameter("Stiffener set back", "P1", distance, number, 1)
      parameter("Left plate class", "LeftC", integer, number, 2)
      parameter("Right plate class", "RightC", integer, number, 3)
      attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
      attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
      {
        value ("Left", 0)
        value ("Right", 0)
        value ("Both", 1)
      }
    }
  }
}

```

9. Browse for the images you want to use in the dialog box.

If you make new images, make sure they are in the bitmap (.bmp) format. Save the images in the ..\ProgramData\Trimble\Tekla Structures \<version>\Bitmaps folder.

10. Replace the option texts with the actual file names of the images, but with the file name extension .xbm.

```

page("TeklaStructures", "")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      parameter("Stiffener set back", "P1", distance, number, 1)
      parameter("Left plate class", "LeftC", integer, number, 2)
      parameter("Right plate class", "RightC", integer, number, 3)
      attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
      attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
      {
        value ("CC_Left.xbm", 0)
        value ("CC_Right.xbm", 0)
        value ("CC_Both.xbm", 1)
      }
    }
  }
}

```

11. Save the .inp file.
12. Close and reopen the model for the change to take effect.

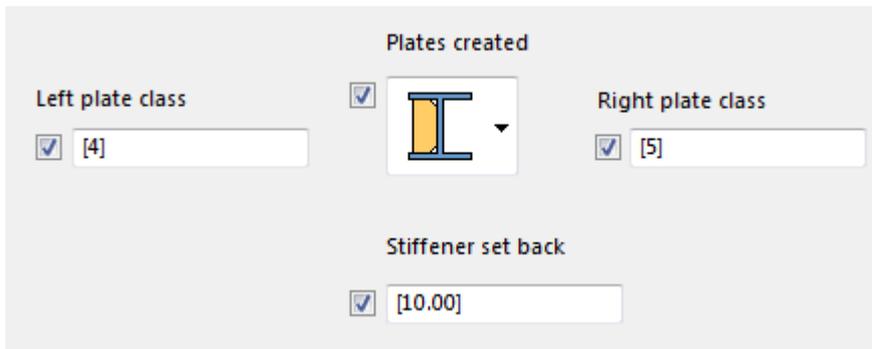
Example: Arrange text boxes and labels

In this example, we will arrange the text boxes and labels around a list in the dialog box. You can do this either in the custom component dialog editor or by editing the input (.inp) file manually.

When we begin, the dialog box looks like the following:



We will arrange the dialog box elements more nicely, in the following manner:



Arrange the elements by using the dialog editor

1. [Create a custom stiffener detail \(page 836\)](#) with all the needed variables that control the creation of stiffener plates.
2. Open the stiffener dialog box for editing.
 - a. In the model, select the custom stiffener detail.
 - b. Right-click and select **Edit Custom Component Dialog Box**.
3. Drag the **Plates created** label above the list with images.
4. Drag the **Left plate class** label and the corresponding text box to the left side of the list.
5. Drag the **Right plate class** label and the corresponding text box to the right side of the list.
6. Drag the **Stiffener set back** label and the corresponding text box underneath the list.
7. In the dialog editor, click **File** --> **Save** to save the changes.
8. Close and reopen the model for the change to take effect.

Arrange the elements by editing the .inp file

1. [Create a custom stiffener detail \(page 836\)](#) with all the needed parametric variables that control the creation of stiffener plates.

2. In the model, click **File --> Open the model folder** to open the current model folder.
3. Go to the \CustomComponentDialogFiles folder.
4. Open the .inp file in a text editor.
5. Edit the file as follows:

```

page("TeklaStructures", "")
{
    detail(1, "Stiffeners")
    {
        tab_page("", " Parameters 1 ", 1)
        {
            attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
            attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90)
            {
                value("CC_Left.xbm", 0)
                value("CC_Right.xbm", 0)
                value("CC_Both.xbm", 1)
            }
            attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
            attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
            parameter("", "LeftC", integer, number, 146, 192, 160)
            parameter("", "RightC", integer, number, 522, 194, 160)
            parameter("", "P1", distance, number, 357, 289, 160)
            attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
        }
    }
}

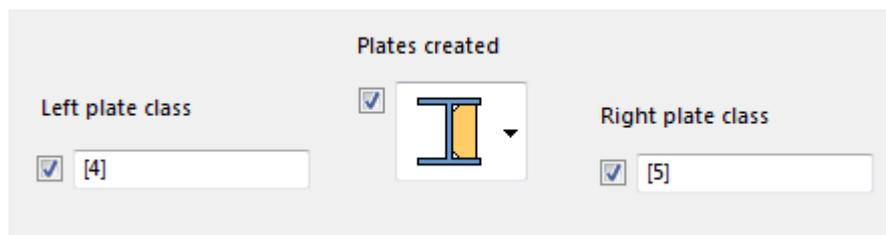
```

6. Save the .inp file.
7. Close and reopen the model for the change to take effect.

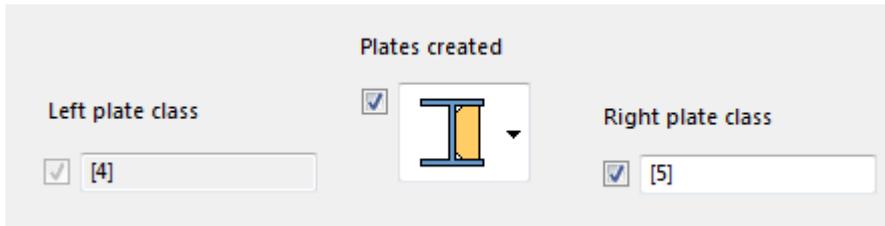
Example: Dim unavailable options

In this example, we will dim the unavailable options in the stiffener dialog box based on conditions. You can do this either in the custom component dialog editor or by editing the input (.inp) file manually.

When we begin, all the options are available:

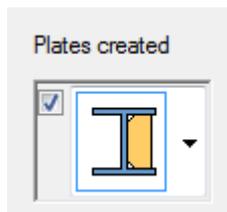


We will define that the **Left plate class** text box is unavailable if only the right plate is created, and vice versa.

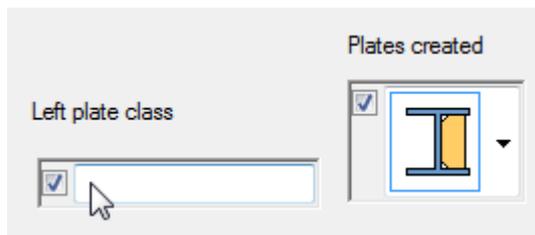


Dim unavailable options by using the dialog editor

1. [Create a custom stiffener detail \(page 836\)](#) with all the needed parametric variables that control the creation of stiffener plates.
2. Open the stiffener dialog box for editing.
 - a. In the model, select the custom stiffener detail.
 - b. Right-click and select **Edit Custom Component Dialog Box**.
3. Define that the **Left plate class** text box must be dimmed if only the right stiffener plate is created.
 - a. In the **Plates created** list, select the image for the right plate class. Note that a blue selection border must be displayed for the image:



- b. Hold down the **Ctrl** key and click the **Left plate class** text box.

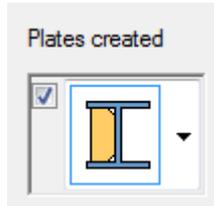


- c. Click the **Toggle visibility**  button. The **Left plate class** text box is now dimmed:

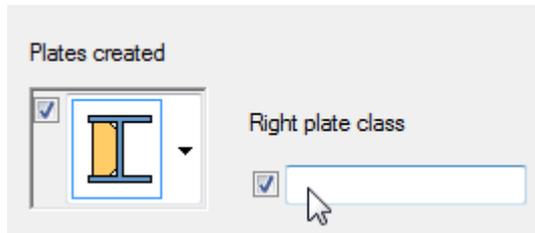


4. Unselect the **Left plate class** text box by clicking the **Right plate class** text box.
5. Define that the **Right plate class** text box must be dimmed if only the left stiffener plate is created.

- a. In the **Plates created** list, select the image for the left plate class.
Note that a blue selection border must be displayed for the image:



- b. Hold down the **Ctrl** key and select the **Right plate class** text box.



- c. Click the **Toggle visibility**  button.

The **Right plate class** text box is now dimmed:



6. In the dialog editor, click **File** --> **Save** to save the changes.
7. Close and reopen the model for the change to take effect.

Dim unavailable options by editing the .inp file

1. [Create a custom stiffener detail \(page 836\)](#) with all the needed parametric variables that control the creation of stiffener plates.
2. In the model, click **File** --> **Open the model folder** to open the current model folder.
3. Go to the `\CustomComponentDialogFiles` folder.
4. Open the `.inp` file in a text editor.
5. Add the following line to the end of the attribute P4 line:

```

"toggle_field:LeftC=0;RightC=1"

page("TeklaStructures", "")
{
  detail(1, "Stiffeners")
  {
    tab_page("", " Parameters 1 ", 1)
    {
      attribute("", "Plates created", label, "%s", none, none, "0", "0", 334, 118)
      attribute("P4", "", option, "%s", none, none, "0.0", "0.0", 360, 151, 90, "toggle_field:LeftC=0;RightC=1")
      {
        value("CC_Left.xbm", 0)
        value("CC_Right.xbm", 0)
        value("CC_Both.xbm", 1)
      }
      attribute("", "Left plate class", label, "%s", none, none, "0", "0", 125, 157)
      attribute("", "Right plate class", label, "%s", none, none, "0", "0", 497, 160)
      parameter("", "LeftC", integer, number, 146, 192, 160)
      parameter("", "RightC", integer, number, 522, 194, 160)
      parameter("", "P1", distance, number, 357, 289, 160)
      attribute("", "Stiffener set back", label, "%s", none, none, "0", "0", 330, 255)
    }
  }
}

```

The logic is the following:

Selecting the image **CC_left** returns 0, **CC_right** returns 1, and **CC_both** returns 2.

```
toggle_field:RightC=1
```

When 0 (left) is selected, **RightC** is dimmed.

```
toggle_field:LeftC=0
```

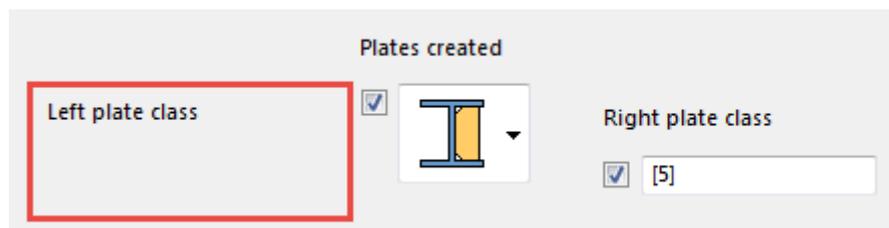
When 1 (right) is selected, **LeftC** is dimmed.

6. Save the .inp file.
7. Close and reopen the model for the change to take effect.

TIP If you want to hide unavailable options instead of dimming them in the stiffener dialog box, add an exclamation mark in the conditions:

```
"toggle_field:!LeftC=0;!RightC=1"
```

The option is now completely hidden when unavailable:



Dialog editor settings

Click **Tools** --> **Options** in the **Custom Component Dialog Editor** to view and modify basic settings in the dialog editor. Click **Tools** --> **Change Language** to change the language of the dialog editor.

Option	Description
Image folder	The location of the image folder. To restore the default folder setting, click Default .
Project folder	The location of the project folder. When you create a completely new input file by clicking File --> New and then save it, the file is saved in the project folder. Note that existing input files are saved under the model folder.
Parameter width	The default width for text boxes.
Attribute width	The default width for lists.
Grid spacing X Grid spacing Y	The spacing of the pixel grid (page 824) in the X and Y directions. The default value is 5.
Snap to grid	Select to show or hide the pixel grid.

Option	Description
Language	<p>Select a language from the list. Close and reopen the dialog editor for the change to take effect. You have the following options:</p> <ul style="list-style-type: none"> • Auto: the dialog editor follows the language of the Tekla Structures user interface • English • Dutch • French • German • Italian • Spanish • Japanese • Chinese Simplified • Chinese Traditional • Czech • Portuguese Brazilian • Hungarian • Polish • Russian

Option	Description
	• Korean

See also

[Modify the dialog box of a custom component \(page 824\)](#)

8.10 Add a custom component to a model

Use the **Applications & components** catalog to add your custom component to a model.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. To search for a component, browse the catalog or enter a search term in the search box. Custom components have the following symbols in the catalog:

Type	Symbol
Custom part	
Custom connection or seam	
Custom detail	

3. Select the custom component you want to add.
4. Follow the instructions on the status bar to add the custom component in the model.
5. To modify the properties, double-click the custom component in the model.

Example: Add a custom connection to a model

In this example, you will add a previously created custom end plate connection to a model. Because you have not modified the custom component to adapt to different situations in the model, you need to add it to a similar location where it was created. Otherwise the custom component may not work as required.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. In the catalog, select the custom end plate connection you want to add. Tekla Structures displays instructions on the status bar.
3. Select the column as the main part.

4. Select the beam as the secondary part.
Tekla Structures adds the end plate connection to the model.

See also

[Add or move a custom part in the model \(page 860\)](#)

8.11 Add or move a custom part in the model

Use the direct modification handles and dimensions when you add or move custom parts. If you are unable to select custom parts in the model, ensure

that the **Select components**  selection switch is active.

NOTE This method cannot be used when adding custom parts to faces that have cuts or edge chamfers. You need to hide the cutting parts and edge chamfer objects from the view before you add custom parts on cut or chamfered faces using direct modification.

We do not recommend using this method with custom parts that are parametric, and in which the input points define the dimensions of the custom part. The preview is simplified, based on the default custom part dimensions, and snapping has a different focus than usually.

1. Ensure that  **Direct modification** is switched on.
2. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
3. In the catalog, select the custom part you want to add.
4. Move the mouse pointer over part faces and edges in the model, and see how the custom part turns over and adjusts to the part faces.

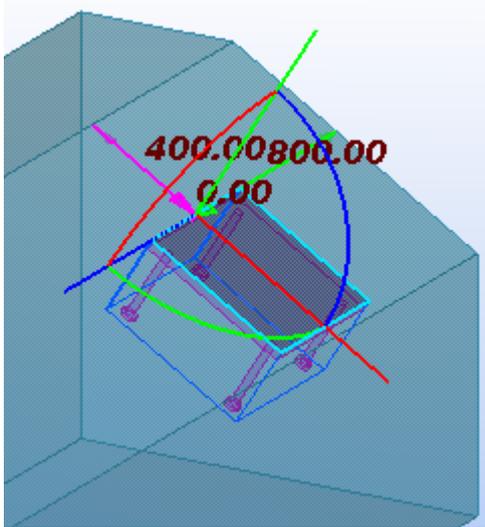
If you are adding the custom part to another object, Tekla Structures shows location dimensions to the nearest object edges.

If you are adding a custom part that has only one input point, press **Tab** to rotate it in 90-degree steps around the work plane Y axis.

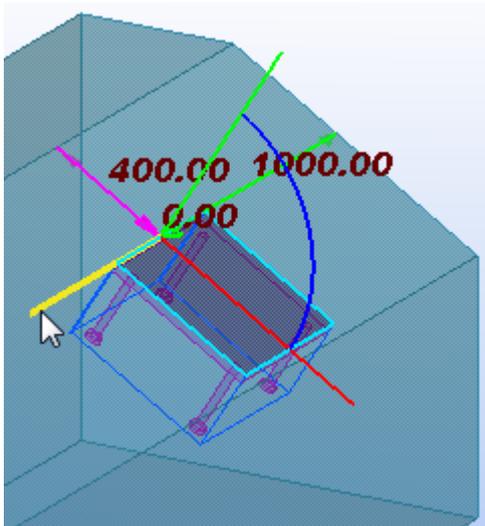
5. Depending on the number of custom part input points, pick one or two points to place the custom part in the model.

Tekla Structures shows the coordinate axes, rotation handles, and location dimensions that you can use to fine-tune the location and rotation of the

custom part. The handles are red, green, and blue, according to the local coordinate system of the custom part.

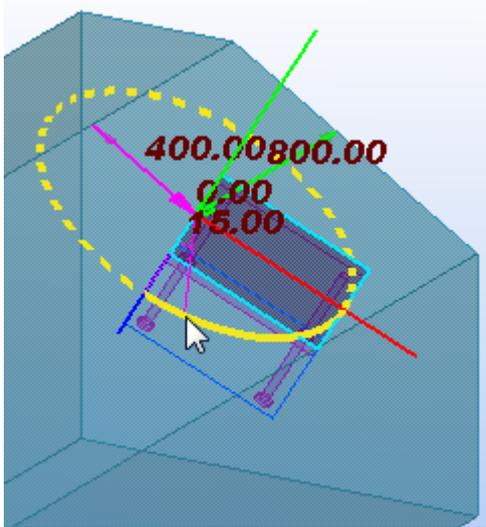


6. Click the middle mouse button to confirm the location and rotation. Tekla Structures adds the custom part to the model.
7. To move the custom part along any of its coordinate axes, drag the relevant axis handle to a new location.



8. To rotate the custom part around any of its coordinate axes, drag the relevant rotation handle to a new location.

Press **Tab** to rotate the custom part in 90-degree steps in the direction of the selected rotation handle.



9. To move or rotate the custom part by specifying a distance or angle:
 - a. Select an axis handle, a rotation handle, or a dimension arrowhead.
 - b. Type the value by which you want the dimension to change.
When you start typing, Tekla Structures displays the **Enter a Numeric Location** dialog box.
 - c. Click **OK** to confirm the new dimension.
10. To stop modifying, press **Esc**.

See also

[Resize and reshape model objects \(page 103\)](#)

8.12 Import and export custom components

You can import and export custom components as `.uel` files between models.

TIP You can share your custom components in Tekla Warehouse, and also download custom components made by other users.

Export custom components

You can export custom components in a `.uel` file.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. In the catalog, select the custom components you want to export.
3. Right-click the selection, and then select **Publish**.
4. Browse for the folder where you want to save the file.
5. Enter a name for the export file.

The file name extension is `.uel`. Do not change the file name after exporting the custom component. If the file name differs from the name in the **Applications & components** catalog, it may be difficult to find the right component later on.

6. Click **Save** to export the custom components.

TIP If you want to export custom components as separate files, select the custom components in the **Applications & components** catalog, right-click, and then select **Publish separately**.

Import custom components

You can import previously made custom components to another model.

1. Click the **Applications & components** button  in the side pane to open the **Applications & components** catalog.
2. Click the **Access advanced features** button , and then select **Import**.
3. Browse for the folder that contains the export file.

The location depends on where you saved the file when you exported the custom component.
4. Select the export file.
5. Click **Open** to import the custom components.

TIP You can import custom components to a new model automatically by using the `XS_UEL_IMPORT_FOLDER` advanced option. Export all custom components to certain folders and enter these folders as the value for the `XS_UEL_IMPORT_FOLDER` advanced option to easily import the custom components to new models.

8.13 Custom component settings

Here you will find more information about the various custom component properties and plane types.

- [Custom component properties \(page 864\)](#)
You must define these properties when you create new custom components. You can change some of these properties when you modify an existing custom component.
- [Default properties of a custom component \(page 867\)](#)
Each custom component has a dialog box that you can modify. By default, the dialog box has a **Position** tab for custom parts and a **General** tab for custom connections, details, and seams.
- [Plane types \(page 872\)](#)
When you create distance variables for a custom component, you must select a plane type. The plane type defines what planes you can select.
- [Variable properties \(page 875\)](#)
Use the **Variables** dialog box to define properties for distance and parametric variables.

Custom component properties

You must define these properties when you create new custom components with the **Custom Component Wizard**. You can change some of these properties when you modify an existing custom component.

For more information, see [Create a custom component \(page 747\)](#) and [Modify a custom component \(page 761\)](#).

Type/Notes tab properties

On the **Type/Notes** tab, you have the following options:

Option	Description
Type	Select the type of the custom component. Type affects how you insert the custom component in the model. Type also defines if the custom component connects to existing parts.
Name	Enter a unique name for the custom component.
Description	Enter a short description for the custom component. Tekla Structures shows the description in the Applications & components catalog.

Option	Description
Component identifier	Enter an additional name or reference for the component, for example a design code reference. This can be shown in general arrangement and assembly drawings, and in lists. To show this in drawings, include Code in the Connection Mark Properties dialog box.

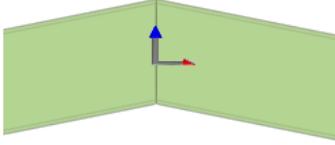
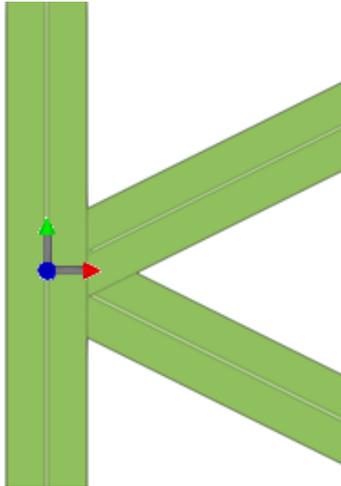
Position tab properties

On the **Position** tab, you have the following options:

Option	Description	Note
Up direction	Sets the default up direction.	Not available for parts.
Position type	The position (or origin) of the component, relative to the main part.	Not available for details and parts.

You can define the position for custom connections and seams. You have the following options:

Option	Description	Example
Middle	Where the center lines of the main and secondary parts intersect.	
Box plane	Where the main part bounding box and the center line of the secondary part intersect.	
Collision plane	Where the main part and the center line of the secondary part intersect.	

Option	Description	Example
Endend plane	Where the center line of the secondary part hits the end of the main part.	
Gusset plane	Where the center lines of the main part and the first secondary part intersect. The x direction is perpendicular to the center line of the main part.	

Advanced tab properties

On the **Advanced** tab, you have the following options:

Option	Description	Note
Detail type	<p>Determines on which side of the main part the component is located. The options are:</p> <ul style="list-style-type: none"> • Intermediate detail Tekla Structures creates all components on the same side of the main part • End detail Tekla Structures creates all components on the side of the main part closest to the details <p>Only affects asymmetric components.</p>	Only available for details and seams

Option	Description	Note
Definition point position in relation to primary part	Determines the position you pick to create the detail, relative to the main part.	Only available for details
Definition point position in relation to secondary part	Determines where the component is created, relative to the secondary part.	Only available for connections and seams
Allow multiple instances of connection between same parts	Select this option to create many components to the same main part, in different locations.	Only available for connections and seams
Exact positions	<p>Select this option to position the seam based on the positions that you pick in the model.</p> <p>Clear the check box to let Tekla Structures use automatic seam recognition to position the seam. This is useful especially with warped seams.</p>	Only available for seams
Use the center of the bounding box in positioning	Select to position the custom part based on the center of its bounding box (the box that surrounds the actual part profile).	Only available for parts

Default properties of a custom component

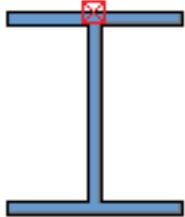
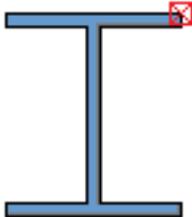
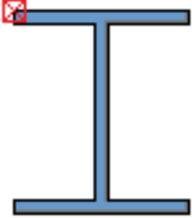
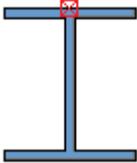
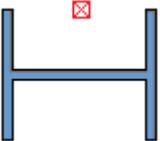
Each custom component has a dialog box that you can modify. By default, the dialog box has a **Position** tab for custom parts and a **General** tab for custom connections, details, and seams.

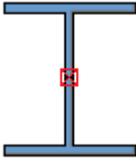
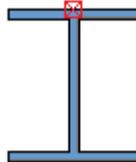
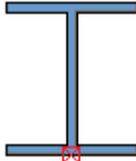
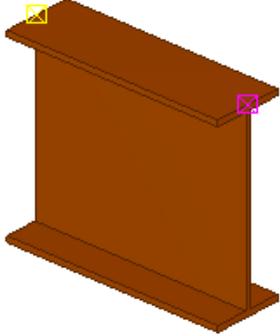
For more information, see [Modify the dialog box of a custom component \(page 824\)](#).

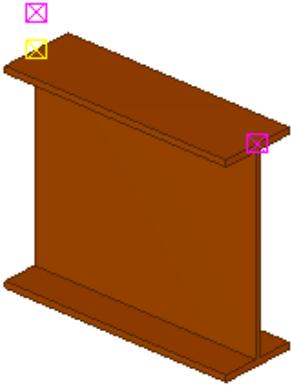
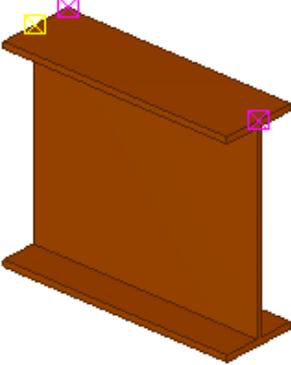
To view the current properties, double-click the custom component in the model.

Default properties of custom parts

By default, the dialog box of a custom part has the following options:

Option	Description	Example
On plane	Changes part location on the work plane.	Middle 
		Right 
		Left 
Rotation	Rotates the part in steps of 90 degrees.	Top and Below 
		Front and Back 

Option	Description	Example
At depth	Changes part location perpendicular to the work plane.	Middle 
		Front 
		Behind 
Show third handle	Sets the third handle of a nested custom part visible in the desired direction. You can bind the third handle in the desired direction and thus force the part to follow the rotation of another part.	None 

Option	Description	Example
		<p data-bbox="1066 277 1161 309">Above</p>  <p data-bbox="1066 748 1230 779">On the left</p> 

Default properties of custom connections, details, and seams

By default, the dialog box of a custom connection, detail, or seam has the following options:

Option	Description	Note
Up direction	Indicates how the component is rotated around the secondary part, relative to the current work plane. If there are no secondary parts, Tekla Structures rotates the connection around the main part.	
Position in relation to primary part	The creation point of the component, relative to the main part.	Only available for details.

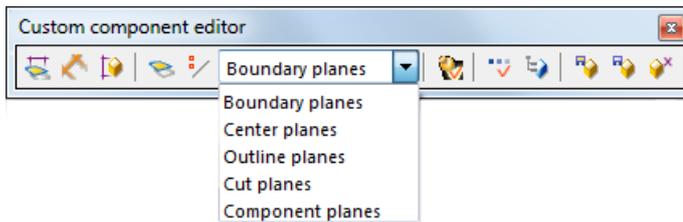
Option	Description	Note
Position in relation to secondary part	Tekla Structures automatically places the component according to the selected option.	By default, only available for seams. If you want to use this property in connections, select the Allow multiple instances of connection between same parts check box on the Advanced tab when you create the component.
Place to picked positions	Select this option to place the seam at the points you pick.	Only available for seams.
Detail type	<p>Determines on which side of the main part the component is located. The options are:</p> <ul style="list-style-type: none"> • Intermediate detail Tekla Structures creates all components on the same side of the main part. • End detail Tekla Structures creates all components on the side that is closest to the details. <p>Only affects asymmetric components.</p>	Only available for details.
Locked	Select Yes to prevent other users from modifying the properties.	
Class	The class of the parts that the custom component creates.	
Connection code	Identifies the component. You can display this connection code in connection marks in drawings.	
AutoDefaults rule group	The rule group used for setting the connection properties.	

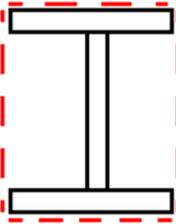
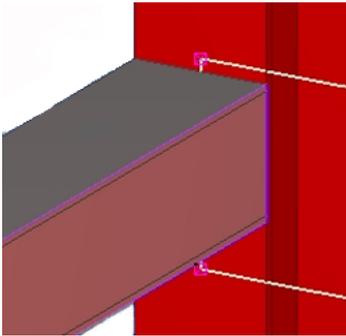
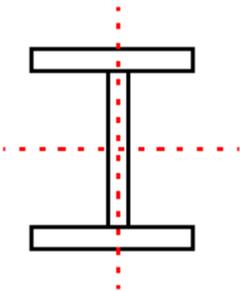
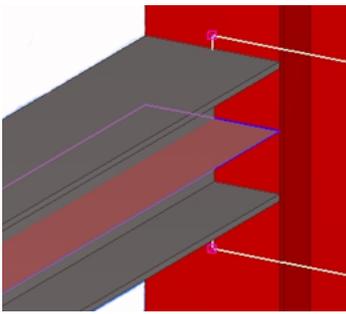
Option	Description	Note
AutoConnection rule group	The rule group Tekla Structures uses to select the connection.	

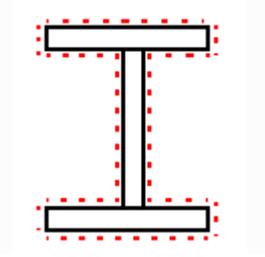
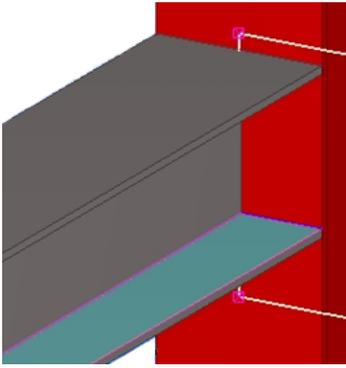
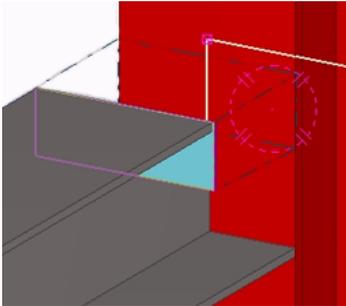
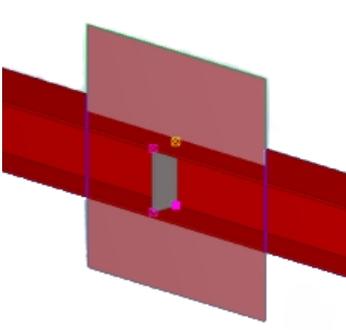
Plane types

When you add distance variables to a custom component, you must select a plane type. The plane type defines what planes you can select.

You have the following options:



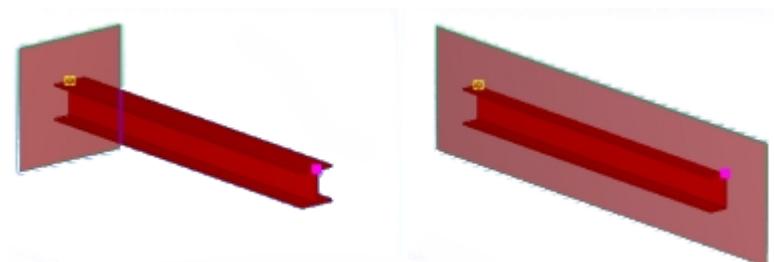
Plane type	Description	Example
Boundary planes	You can select the edges of a bounding box that surrounds the profile. 	
Center planes	You can select the center planes of a profile. 	

Plane type	Description	Example
Outline planes	You can select the outer and inner surfaces of a profile. 	
Cut planes	If the part contains line, part, or polygon cuts, this option enables you to select cut surfaces. Fittings cannot be selected.	
Component planes	What you can select depends on the component type and the Position type of the custom component.	

Examples of component planes

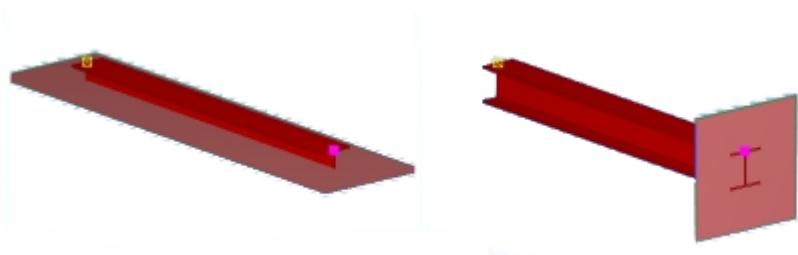
See below for examples of possible component planes. What you can select depends on the component type and the **Position type** of the custom component.

Part component planes

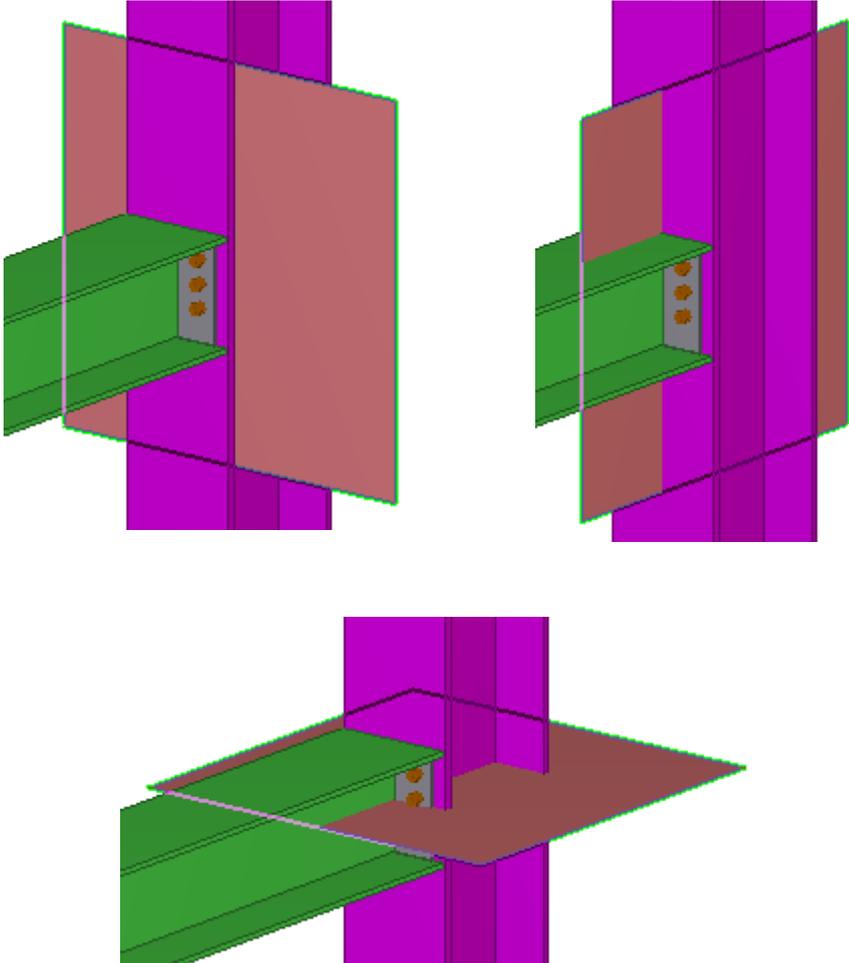


Custom components

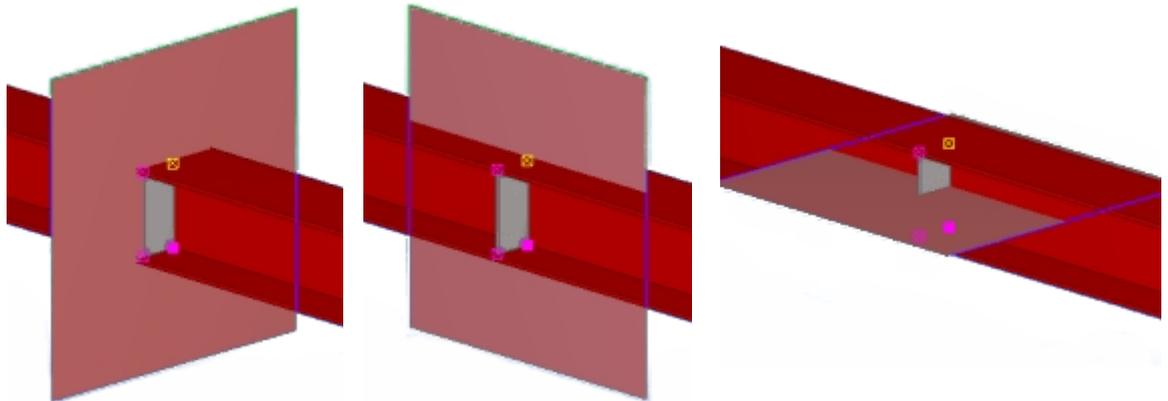
Custom component settings



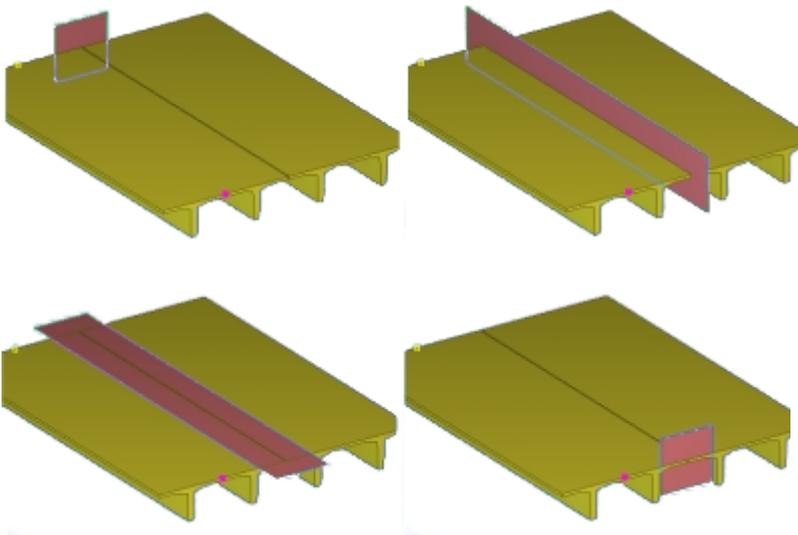
Connection component planes



Detail component planes



Seam component planes



Variable properties

Use the **Variables** dialog box to view, modify, and create parametric variables, and to view fixed and reference distance variables.

Tekla Structures uses variables with [custom components \(page 763\)](#), sketched cross sections, and parametric modeling. The examples below are given for

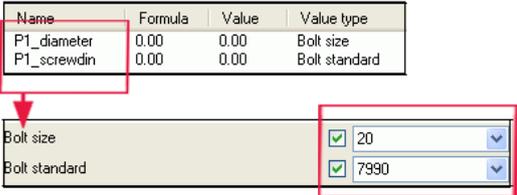
custom components, but the same principles apply also to sketched cross sections and parametric modeling.

Option	Description
Category	Component parameters lists all variables in the component. Model parameters lists variables in the current model (such as bindings between a part's end point and a grid plane).
Name	A unique name of a variable. Use this name to refer to the variable in the custom component editor. The maximum length is 19 characters.
Formula	Use this box to enter a value or a formula (page 782) . Formulas begin with =.
Value	Shows the current value of Formula .
Value type	Select a value type from the list. The type determines what kind of value you can enter for the variable.
Variable type	This property can be either Distance or Parametric .
Visibility	Use this setting to control the visibility of a variable. Set to Show to display the variable in the custom component dialog box.
Label in dialog box	The name of the variable that Tekla Structures displays in the custom component dialog box. The maximum length is 30 characters.

Value types

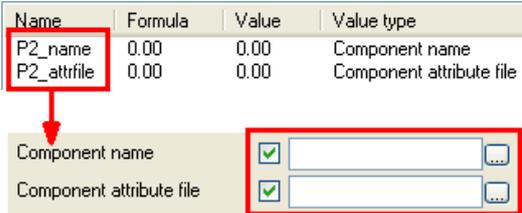
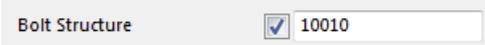
You have the following options for the value type:

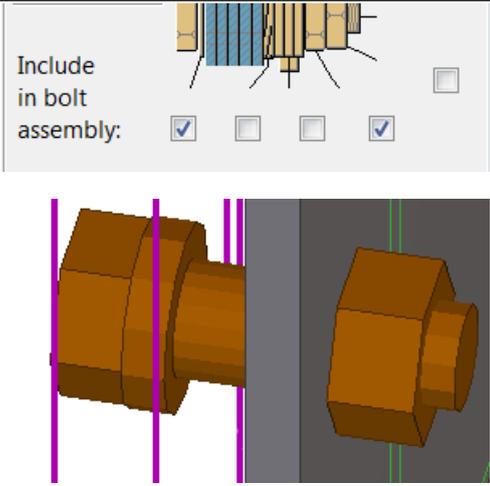
Option	Description
Number	A whole (integer) number. Use for quantity and multiplier.
Length	A decimal (floating point) number. Use for lengths and distances. Length numbers have unit (mm, inch, etc.) and they are rounded to two decimal places.

Option	Description												
Text	A text (ASCII) string.												
Factor	A decimal value without a unit. You can set the number of decimals for the value type in File menu --> Settings --> Options --> Units and decimals .												
Angle	A decimal number type for storing angles, stored to one decimal place, in radians.												
Material	A data type associated with the material catalog. Use to select a material from the standard material dialog.												
Profile	A data type associated with the profile catalog. Use to select a profile from the standard profile dialog.												
Bolt size Bolt standard	<p>Data types linked to the bolt catalog. Bolt size works with Bolt standard. They have a fixed naming format: P_x_diameter and P_x_screwdin. Do not change the fixed name.</p> <p>To show values for these in the component's dialog box, x must be the same for both, for example, P₁_diameter and P₁_screwdin.</p>  <table border="1" data-bbox="858 1240 1262 1317"> <thead> <tr> <th>Name</th> <th>Formula</th> <th>Value</th> <th>Value type</th> </tr> </thead> <tbody> <tr> <td>P1_diameter</td> <td>0.00</td> <td>0.00</td> <td>Bolt size</td> </tr> <tr> <td>P1_screwdin</td> <td>0.00</td> <td>0.00</td> <td>Bolt standard</td> </tr> </tbody> </table>	Name	Formula	Value	Value type	P1_diameter	0.00	0.00	Bolt size	P1_screwdin	0.00	0.00	Bolt standard
Name	Formula	Value	Value type										
P1_diameter	0.00	0.00	Bolt size										
P1_screwdin	0.00	0.00	Bolt standard										
Bolt type	<p>For determining the bolt type (site/workshop) in the custom component dialog box. Linked to the Bolt type property of bolts in the Custom component browser.</p> 												
Stud size Stud standard Stud length	<p>Data types linked to the bolt catalog. Stud size, Stud standard and Stud length work together. They have a fixed naming format: P_x_size,</p>												

Option	Description												
	<p>Px_standard and Px_length. Do not change the fixed names.</p> <p>To show values for these in the component's dialog box, x must be the same for all of them. For example, P9_size, P9_standard, and P9_length.</p> <table border="1" data-bbox="855 577 1289 651"> <thead> <tr> <th>Name</th> <th>Formula</th> <th>Value</th> <th>Value type</th> </tr> </thead> <tbody> <tr> <td>P9_size</td> <td>6.35</td> <td>6.35</td> <td>Stud size</td> </tr> <tr> <td>P9_standard</td> <td>NELSON</td> <td>NELSON</td> <td>Stud standard</td> </tr> </tbody> </table> 	Name	Formula	Value	Value type	P9_size	6.35	6.35	Stud size	P9_standard	NELSON	NELSON	Stud standard
Name	Formula	Value	Value type										
P9_size	6.35	6.35	Stud size										
P9_standard	NELSON	NELSON	Stud standard										
Distance list	<p>Use with options that have several length values, such as bolt spacings.</p> <p>Use space as a separator between the distances.</p> 												
Weld type	<p>A data type for selecting the weld type.</p> 												
Chamfer type	<p>A data type for determining the shape of a chamfer.</p> <p>For more information, see Create part chamfers (page 368).</p>												
Welding site	<p>A data type for determining the welding place: workshop or building site.</p>												
Rebar grade Rebar size Rebar bending radius	<p>Data types linked to reinforcement catalog. Rebar grade, Rebar size, and Rebar bending radius work together. They have a fixed naming format: Px_grade, Px_size, and Px_radius. Do not change the fixed name.</p> <p>To show values for these in the component's dialog box, x must be the same for all, for example, P1_grade, P1_size, and P1_radius.</p>												

Option	Description																						
	<table border="1"> <thead> <tr> <th data-bbox="847 277 970 304">Name</th> <th data-bbox="970 277 1082 304">Formula</th> <th data-bbox="1082 277 1177 304">Value</th> <th data-bbox="1177 277 1375 304">Value type</th> </tr> </thead> <tbody> <tr> <td data-bbox="847 304 970 331">P1_grade</td> <td data-bbox="970 304 1082 331">0.00</td> <td data-bbox="1082 304 1177 331">0.00</td> <td data-bbox="1177 304 1375 331">Rebar grade</td> </tr> <tr> <td data-bbox="847 331 970 358">P1_size</td> <td data-bbox="970 331 1082 358">0.00</td> <td data-bbox="1082 331 1177 358">0.00</td> <td data-bbox="1177 331 1375 358">Rebar size</td> </tr> <tr> <td data-bbox="847 358 970 385">P1_radius</td> <td data-bbox="970 358 1082 385">0.00</td> <td data-bbox="1082 358 1177 385">0.00</td> <td data-bbox="1177 358 1375 385">Rebar bending radius</td> </tr> </tbody> </table> <table border="1"> <tbody> <tr> <td data-bbox="847 421 1082 448">Rebar grade</td> <td data-bbox="1082 421 1375 448"><input checked="" type="checkbox"/> <input type="text"/></td> </tr> <tr> <td data-bbox="847 448 1082 474">Rebar size</td> <td data-bbox="1082 448 1375 474"><input checked="" type="checkbox"/> <input type="text"/></td> </tr> <tr> <td data-bbox="847 474 1082 501">Rebar bending radius</td> <td data-bbox="1082 474 1375 501"><input checked="" type="checkbox"/> <input type="text"/></td> </tr> </tbody> </table>	Name	Formula	Value	Value type	P1_grade	0.00	0.00	Rebar grade	P1_size	0.00	0.00	Rebar size	P1_radius	0.00	0.00	Rebar bending radius	Rebar grade	<input checked="" type="checkbox"/> <input type="text"/>	Rebar size	<input checked="" type="checkbox"/> <input type="text"/>	Rebar bending radius	<input checked="" type="checkbox"/> <input type="text"/>
Name	Formula	Value	Value type																				
P1_grade	0.00	0.00	Rebar grade																				
P1_size	0.00	0.00	Rebar size																				
P1_radius	0.00	0.00	Rebar bending radius																				
Rebar grade	<input checked="" type="checkbox"/> <input type="text"/>																						
Rebar size	<input checked="" type="checkbox"/> <input type="text"/>																						
Rebar bending radius	<input checked="" type="checkbox"/> <input type="text"/>																						
Rebar hook type	Used for rebar set end detail modifiers, to specify the hook type.																						
Rebars to split	Used for rebar set splitters, to specify how the bars are to be split (1/1, 1/2, and so on).																						
Rebar stagger type	Used for rebar set splitters, to specify the stagger type (left/right/middle).																						
Rebar lap side	Used for rebar set splitters, to specify the side of the overlap (left/right/middle).																						
Rebar lap placement	Used for rebar set splitters, to determine whether the lapping bars are parallel to each other or on top of each other.																						
Rebar lap type	Used for rebar set splitters, to determine whether the reinforcing bars are kept straight at lap splices by offsetting entire bars, or placed slanted by offsetting bar ends.																						
Reinforcement mesh	For determining meshes in custom components. Linked to the Catalog name property of reinforcement meshes in the Custom component browser .																						
Cross bar location	Used for rebar meshes, to determine whether the crossing bars are located above or below the longitudinal bars.																						
Component name Component attribute file	Use Component name for replacing a sub-component inside a custom component with another sub-component. Linked to the Name property of objects in the Custom component browser . Use Component attribute file for setting the properties of a sub-																						

Option	Description
	<p>component inside a custom component.</p> <p>Component name and Component attribute file work together. They have a fixed naming format: P_x_name and P_x_attrfile. Do not change the fixed name.</p> <p>To show values for these in the component's dialog box, x must be the same for both, for example, P₂_name and P₂_attrfile.</p> 
Yes/No	<p>For determining whether or not Tekla Structures creates an object in a custom component. Linked to the Creation property of objects in the Custom component browser.</p> 
Bitmask	<p>For defining bolt assembly (nuts and washers) and parts with slotted holes. Linked to the Bolt structure and Parts with slotted holes properties of bolts in the Custom component browser.</p> <p>The value is a five-digit series of ones and zeros. This relates to the check boxes in the bolt properties. 1 means that a check box is selected, 0 means that a check box is clear.</p> <p>In the example below, the value of 10010 means that a bolt with a washer and a nut is created in the bolt assembly.</p> 

Option	Description
	

8.14 Hints and tips for using custom components

Here you will find some useful tips on how to create and use custom components more efficiently.

- [Tips for creating custom components \(page 881\)](#)
Follow these guidelines when creating new custom components.
- [Tips for sharing custom components \(page 882\)](#)
Follow these guidelines when sharing custom components with colleagues.
- [Tips for updating custom components to a new version \(page 882\)](#)
When you start using a new version of Tekla Structures, always check that custom components created in older versions work correctly in the new version.

Tips for creating custom components

Follow these guidelines when creating new custom components.

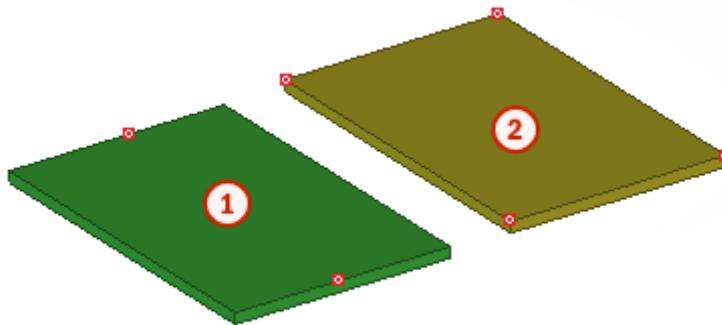
- **Enter short, logical names for custom components.**
Use the description field to describe the component and to explain what it does.
- **Create simple components for specific situations.**
Simple components are easier and faster to model, and also much easier to use. Avoid creating a single, complex component which you will use for every possible purpose.

- **Consider creating a separate component model.**

Use that model when you create and test custom components.

- **Use the simplest part you can.**

For example, if all you need is a rectangular shape, use a rectangular plate, not a contour plate. Rectangular plates only have two handles, so you only need to create a few bindings to manipulate them. Contour plates require more bindings because they have four handles.



1. Rectangular plate
2. Contour plate

- **Model parts only as accurately as you need.**

If the only part information required is a part mark in a general arrangement drawing, plus a quantity on a materials list, create a simple bar or plate. If you need to include the part in a detailed view later on, simply re-model the part more accurately at that point.

- **Model embeds as custom parts and include them in components.**

Tips for sharing custom components

Follow these guidelines when sharing custom components with colleagues.

- **Use Tekla Warehouse to share and store custom components.**
- **Provide essential information.**

If you distribute your component to other users, remember to list the profiles it works with.

- **Use [fixed profiles \(page 316\)](#) whenever possible.**
- **If your custom component contains user-defined profile cross sections, remember to include them when you copy the custom component to a new location.**

Tips for updating custom components to a new version

When you start using a new version of Tekla Structures, always check that custom components created in older versions work correctly in the new version.

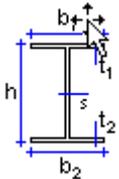
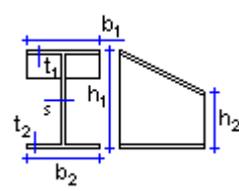
When you edit custom components created with an older version of Tekla Structures, and the new version contains improvements that require an update, Tekla Structures asks whether you want to update the component. If you do not update the component, it works in the same manner as in the version where it was originally created, but you do not gain the benefits of the new improvements.

If you choose to update the component, you need to check and sometimes recreate dimensions depending on the improvements. When you delete a dimension and create a new one (even with the same name), the equations that contain the dimension also need to be modified, because the dependency created by the equation is lost when a dimension is deleted. You can recreate dimensions and modify equations in the custom component editor.

9 Predefined parametric profiles available in Tekla Structures

The below predefined parametric profiles are available in Tekla Structures.
 The profiles are listed in the same order as they appear in the profile catalog in the default environment.
 To change how the profiles are grouped in the profile catalog, modify the profile catalog rules.

9.1 I profiles

	<p>HIh-s-t*b (symmetric) HIh-s-t1*b1-t2*b2</p>
	<p>HIh1-h2-s-t*b HIh1-h2-s-t1*b1-t2*b2</p>

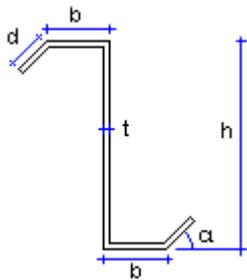
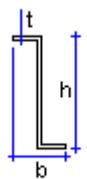
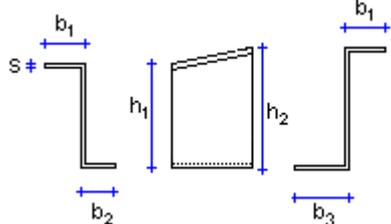
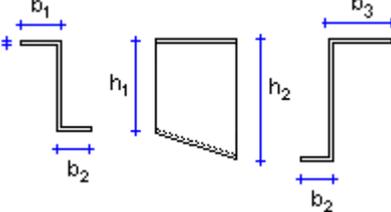
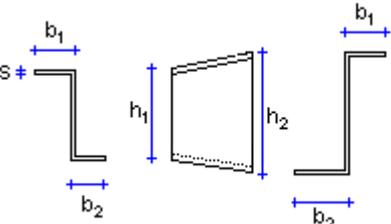
9.2 I beams (steel)

	$I_BLT_Ah-b1-s1-t1*h2-b2-s2-t2$
	$I_BLT_B h*b1*t1*s-b2*t2$
	$I_HEMh*b*c*s*t$
	$I_VAR_Ah1-ht*b1-bt*s*t$

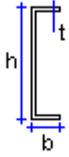
9.3 L profiles

	$Lh*b*t$
--	----------

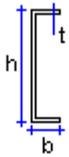
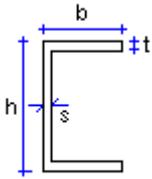
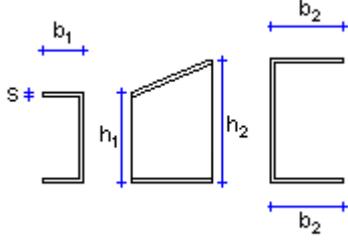
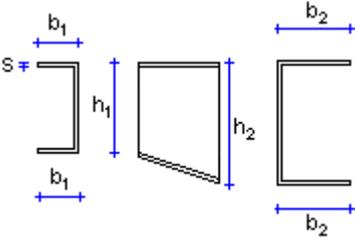
9.4 Z profiles

	<p>BENTZ $h*b*d*t[-a]$</p>
	<p>Z $h*b*t$</p>
	<p>Z_VAR_A $h1*b1*b2-s-h2*b3$</p>
	<p>Z_VAR_B $h1*b1*b2-s-h2*b3$</p>
	<p>Z_VAR_C $h1*b1*b2-s-h2*b3$</p>

9.5 U profiles

	$Uh*b*t$
---	----------

9.6 C profiles

	$Ch*b*t$
	$C_BUILTh*b*s*t$
	$C_VAR_Ah1*b1-s-h2*b2$
	$C_VAR_Bh1*b1-s-h2*b2$

	C_VAR_Ch1*b1-s-h2*b2
	C_VAR_Dh-b-d-c-s

9.7 T profiles

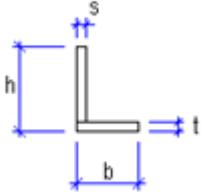
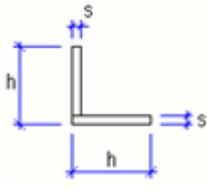
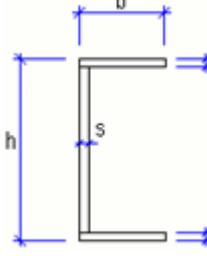
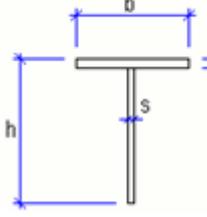
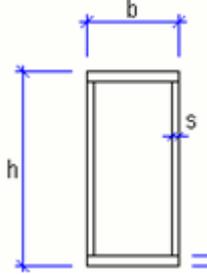
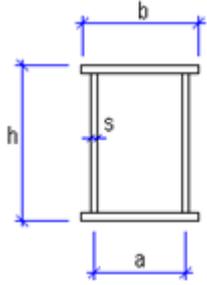
	Th-s-t-b
--	----------

9.8 Welded box profiles

	HK h-s-t*b-c HKh-s-t1*b1-t2*b2-c
--	-------------------------------------

9.9 Welded beam profiles

	B_WLD_A h*b*s*t
--	-----------------

	B_WLD_B $h*b*s*t$
	B_WLD_C $h*s$
	B_WLD_D $h*b*s*t$
	B_WLD_E $h*b*s*t$
	B_WLD_F $h*b*s*[t]$
	B_WLD_G $h*b*s*t*a$

	$B_WLD_H \ h * b_0 * b_u * s * t_0 * t_u$
	$B_WLD_I \ h * b_0 * s * t_0 * b_u * t_u * a$
	$B_WLD_J \ h_1 * h_2 * b * s * t$
	$B_WLD_K \ h_1 * h_2 * b * s * t$
	$B_WLD_L \ h * w_t * w_b * s * t_t * t_b$
	$B_WLD_M \ h_1 * p_1 * p_2 * p_3 * p_4$

<p>Technical drawing of a rectangular box profile. The overall height is P1. The overall width is P9. The inner width is P8. The inner height is P2. The thickness of the top and bottom flanges is P6. The thickness of the side walls is P4. The distance between the inner and outer walls at the top and bottom is P7. The distance between the inner and outer walls at the sides is P5. The distance between the inner and outer walls at the corners is P3.</p>	<p>B_WLD_N $p1 * p2 * p3 * p4 * p5 * p6 * p7 * p8 * p9$</p>
<p>Technical drawing of a cross-section of a box profile. The overall height is P1. The overall width is P2. The thickness of the top flange is b1. The thickness of the side wall is b4. The thickness of the bottom flange is b7. The height of the top flange is h1. The height of the side wall is h5. The height of the bottom flange is h6. The distance between the inner and outer walls at the top is b3. The distance between the inner and outer walls at the sides is b4. The distance between the inner and outer walls at the bottom is b5. The distance between the inner and outer walls at the corners is b6.</p>	<p>B_WLD_O $b1 * h1 * b4 * h5 * b7 * h6 * P1 * P2$</p>
<p>Technical drawing of a cross-section of a box profile. The overall height is H. The thickness of the top flange is TPW. The width of the top flange is W. The thickness of the side wall is WT. The thickness of the bottom flange is BPT. The width of the bottom flange is BPW. The distance between the inner and outer walls at the top is FT. The distance between the inner and outer walls at the bottom is BPT.</p>	<p>B_WLD_P $W * H * FT * WT * TPT * TPW * BPT * BPW$</p>

9.10 Box profiles

<p>Technical drawing of a rectangular box profile. The overall height is h. The overall width is b. The thickness of the top and bottom flanges is t. The thickness of the side walls is s.</p>	<p>B_BUILTh*b*s*t</p>
---	-----------------------

	$B_VAR_Ah1-h2*t$
	$B_VAR_Bh1-h2*t$
	$B_VAR_Ch1-h2*t$

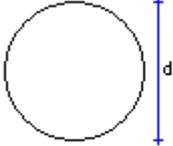
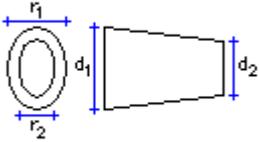
9.11 WQ profiles

	$HQh-s-t1*t2*b2$ $HQh*s-t1*b1-t2*b2-c$
--	---

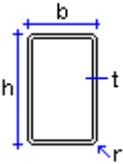
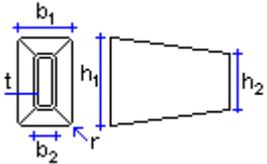
9.12 Rectangular sections

	$PLh*b$ $h=height$ $b=thickness$ <small>(smaller=b)</small>
--	--

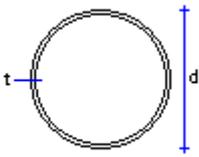
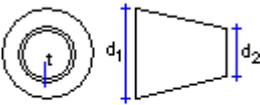
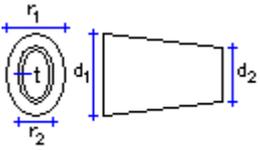
9.13 Circular sections

	Dd
	$ELDd1*r1*d2*r2$

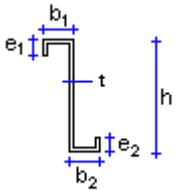
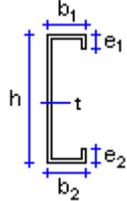
9.14 Rectangular hollow sections

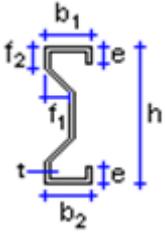
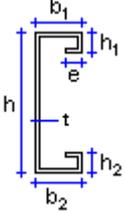
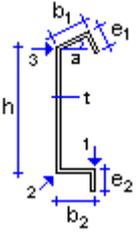
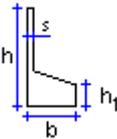
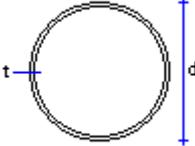
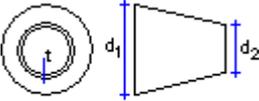
	$Ph*t$ (symmetric) $Ph*b*t$
	$Ph1*b1-h2*b2*t$

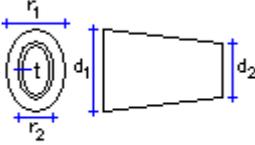
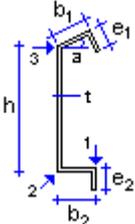
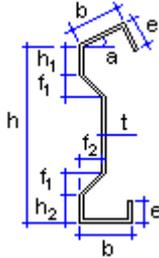
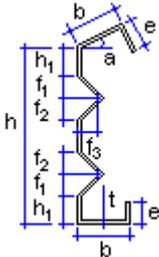
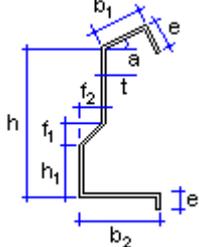
9.15 Circular hollow sections

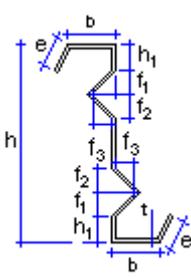
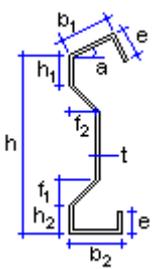
	PDd
	$PDd1*d2*t$
	$EPDd1*r1*d2*r2*t$

9.16 Cold rolled profiles

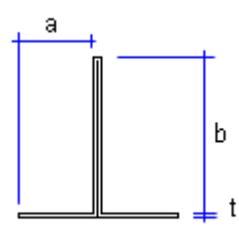
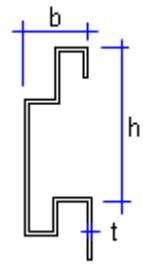
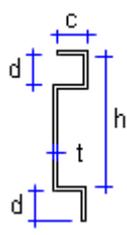
	$ZZ_h-t-e-b$ (symmetric) $ZZ_h-t-e_1-b_1-e_2-b_2$
	$CCh-t-e-b$ (symmetric) $CCh-t-e_1-b_1-e_2-b_2$

	<p>CW h-t-e-b-f-h1 (symmetric) CW h-t-e1*b1-f1-f2-e2*b2</p>
	<p>CUh-t-h1-b-e (symmetric) CUh-t-h1-b1-h2-b2-e</p>
	<p>EBh-t-e-b-a EBh-t-e1-b1-e2-b2-a Reference points: 1=right 2=left 3=top</p>
	<p>BFh-s-b-h1</p>
	<p>SPDd*t</p>
	<p>SPDd2*d2*t</p>

	ESPD d1-d2*t
	ECh-t-e-b-a ECh-t-e1-b1-e2-b2-a
	EDh-t-b-e-h1-h2-f1-f2-a
	EEh-t-e-b-f1-f3-h1-f2-a
	EFh-t-e-b1-b2-f1-f2/h1-a

	EZh-t-e-b-f1-f3-h1-f2-a
	EWh-t-e-b1-b2-f1-f2-h2-h1-a

9.17 Folded plates

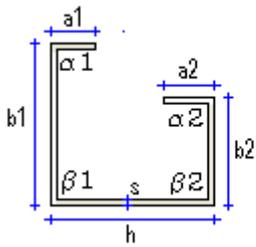
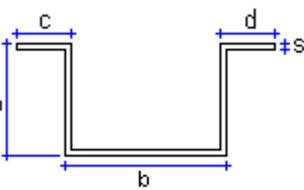
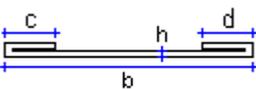
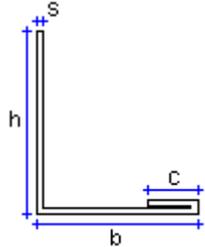
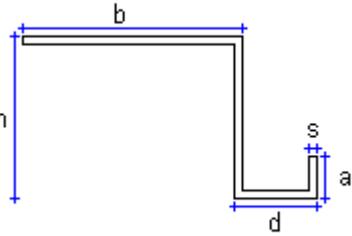
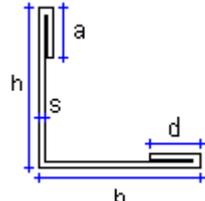
	FFLAa-b-t
	FPANBh-b-t FPANB_-b-t FPANBAh-b-t FPANBA_h-b-t
	FPANBBh-c-d-t

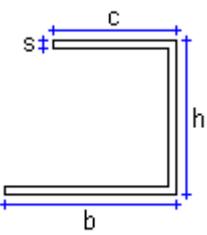
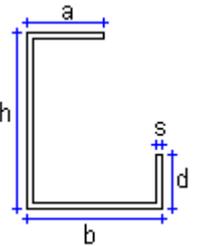
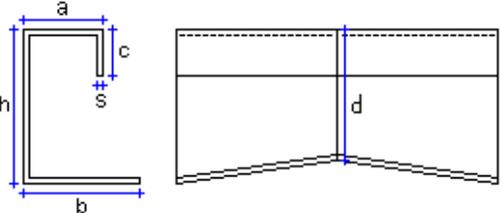
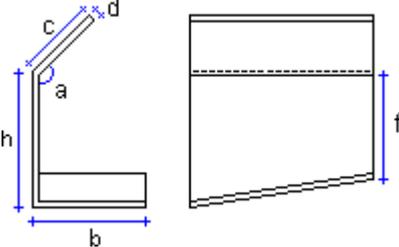
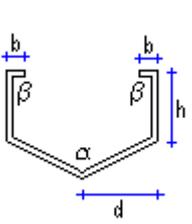
<p>A trapezoidal profile with a top width of b, a bottom width of c, a thickness of t, and a height of d.</p>	FPANCVb-c-d-t
<p>A Z-profile with a total height of h, a total width of b, a thickness of t, and a flange width of c.</p>	FPANGh-b-c-t
<p>A Z-profile with a total height of h, a total width of b, a thickness of t, and a flange width of c.</p>	FPANGAh-b-c-t
<p>A profile with a total height of c, a total width of b, a thickness of t, and a sloped flange length of a.</p>	FPANJa-b-c-t
<p>A profile with a total height of c, a total width of b, a thickness of t, and a sloped flange length of a.</p>	FPANJa-b-c-t
<p>A profile with a total height of b, a total width of c, a thickness of t, a flange width of a, and a gap of g.</p>	FPAN a-b-c-t-g
<p>A profile with a total height of b, a total width of c, a thickness of t, a flange width of a, and a gap of g.</p>	FPANVVa-b-c-t-g

	FP_Ah-b-c-d-g
	FP_AAh*b2*t*a
	FP_Bh-b-c-d-g-i
	FP_BBh-b-d
	FP_Cb-h-c
	FP_CCh-b-a-d-s

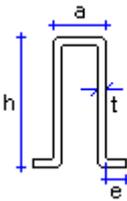
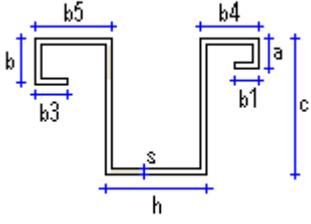
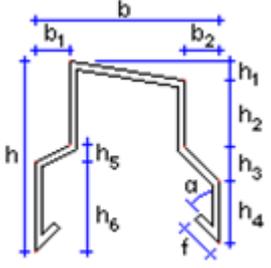
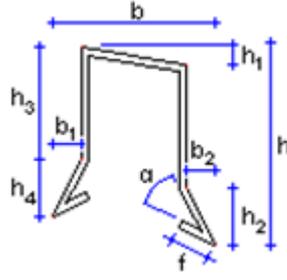
	FP_Db-h-c-d-f-g-i-j-s
	FP_Eb-h-c-d-f-g-s
	FP_Fb-h-c-d-f-g-s
	FP_Gb-h-c-d-f-g-s
	FP_Hb-h-c-d-f-s
	FP_Ib-h-c-d-f-s

	FP_Jb-h-c-d-a
	FP_Kb-h-c-d
	FP_Lb-h-c-d-f-s
	FP_Mb-h-c-d-s
	FP_Nb-h-c-d
	FP_Ob-h-c-d-s

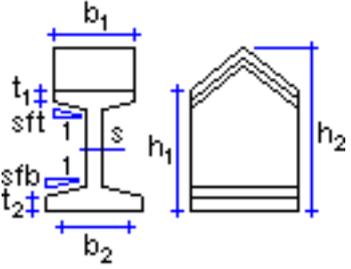
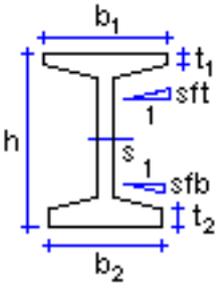
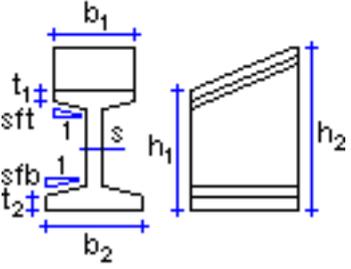
 <p> $\alpha 1 = \text{Alpha } 1$ $\alpha 2 = \text{Alpha } 2$ $\beta 1 = \text{Beta } 1$ $\beta 2 = \text{Beta } 2$ </p>	FP_Pa1*a2*h-b1*b2-Alpha1-Alpha2-Beta1-Beta2-s
	FP_Qb-h-c-d-s
	FP_Rb-h-c-d
	FP_Sb-h-c-s
	FP_Tb-h-a-d-s
	FP_Ub-h-a-d-s

	FP_Vb-h-s-c
	FP_Wb-h-a-d-s
	FP_WWh-b-a-c-s
	FP_Yh-b-c-d
 <p> $\alpha = \text{Alpha}$ $\beta = \text{Beta}$ </p>	FP_Zd-h-b-s-a-f

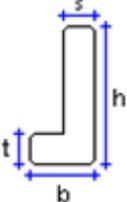
9.18 Hat profiles

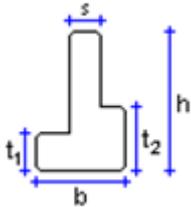
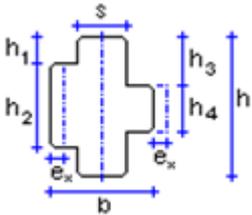
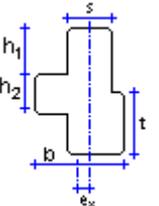
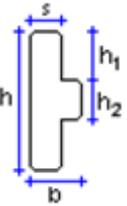
	<p>HAT $h*a*c*t$</p>
	<p>HATCa-b-c-b1-h-b3-b4-b5-s</p>
	<p>HATAb$1*h1*h2*h3*h4*h5*h6*b2*t*f$ $*a*h*b$</p>
	<p>HATBb$*b1*b2*h*h1*h2*h3*h4*t*f*a$</p>

9.19 I beams (concrete)

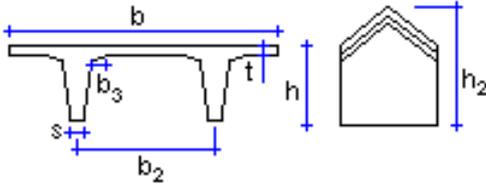
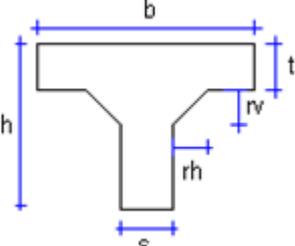
	$IIIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]$
	$IIh*b1*t1-s-b2*t2[-sft[-sfb]]$
	$SIh1*b1*t1-h2-s-b2*t2[-sft[-sfb]]$

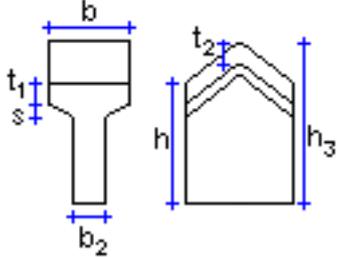
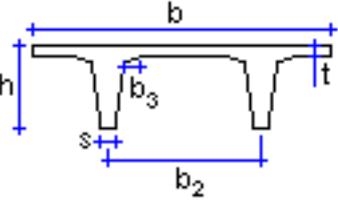
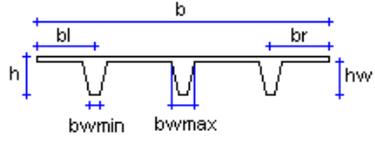
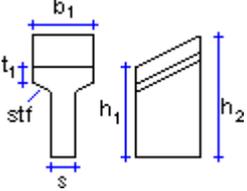
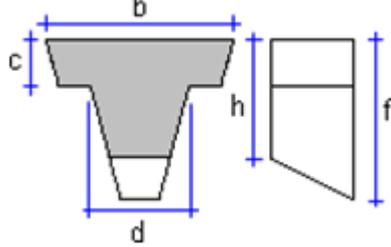
9.20 Ledger beams (concrete)

	$RCLs*h-b*t$
---	--------------

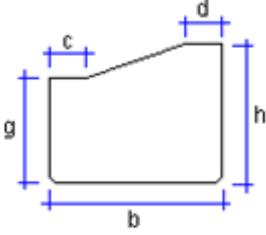
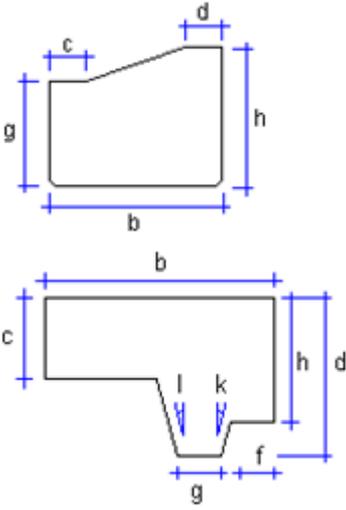
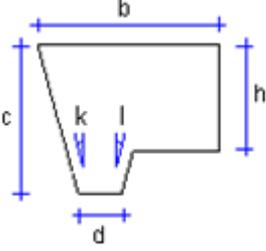
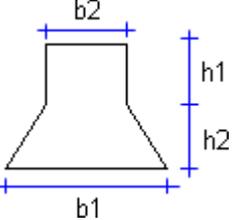
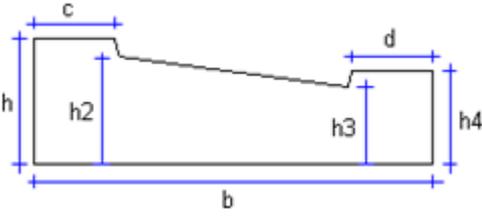
	$\text{RCDLs} \cdot h \cdot b \cdot t$ $\text{RCDLs} \cdot h \cdot b \cdot t_1 \cdot t_2$
	$\text{RCDXs} \cdot h \cdot b \cdot h_2 \cdot h_1$ $\text{RCDXs} \cdot h \cdot b \cdot h_4 \cdot h_3 \cdot h_2 \cdot h_1$ $\text{RCDXs} \cdot h \cdot b \cdot h_4 \cdot h_3 \cdot h_2 \cdot h_1 \cdot \text{ex}$
	$\text{RCXXs} \cdot h \cdot b \cdot t \cdot h_1 \cdot h_2 \cdot \text{ex}$
	$\text{RCXs} \cdot h \cdot b \cdot h_2 \cdot h_1$

9.21 T profiles (concrete)

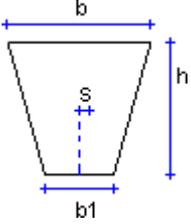
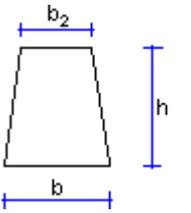
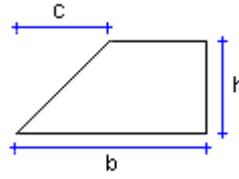
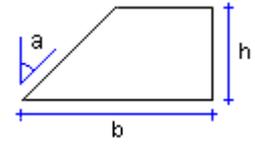
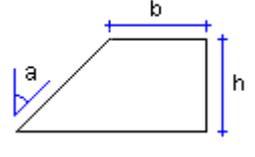
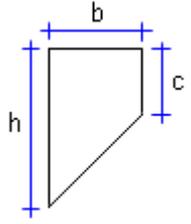
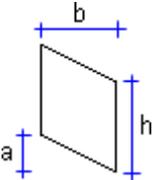
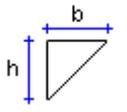
	$\text{HTTh} \cdot b \cdot s \cdot t \cdot b_2 \cdot h_2$
	$\text{TCh} \cdot b \cdot t \cdot s$

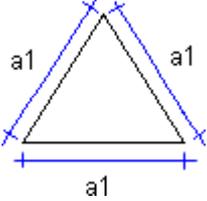
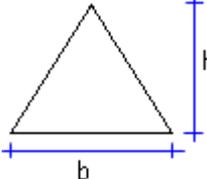
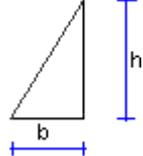
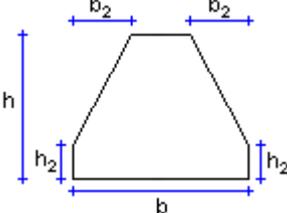
	$TRlh*b-b2*t1-h3-t2$
	$TTh*b-s-t-b2$
	$TTTh*b-bl-br-hw-bwmin-bwmax$
	$T_VAR_Ah1*h2*s*b1*t1-sft$
	$T_VAR_Bh-b-c-d$

9.22 Irregular beams (concrete)

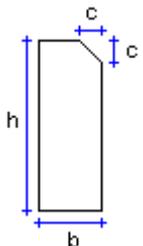
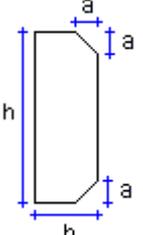
	IRR_Ab-h-g-c-d
	IRR_Bh-b-c-d-f-g
	IRR_Ch-b-c-d
	IRR_Db1*b2-h1*h2
	IRR_Eh-b-c-d-h2-h3-h4

<p>The diagram shows a cross-section of an irregular beam. It has a top edge of length a and a bottom edge of length b. The right side is a vertical line of height b. The left side is a vertical line of height a. A diagonal line connects the top-left corner to the bottom-right corner.</p>	IRR_Fa*b
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b. The bottom edge is horizontal with length b_2. The left side is a vertical line of height h. The right side is a vertical line of height h_2. A diagonal line connects the top-right corner to the bottom-left corner.</p>	IRR_Gh*b*h2*b2
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b. The bottom edge is horizontal with length b_2. The left side is a vertical line of height h_2. The right side is a vertical line of height h. A diagonal line connects the top-left corner to the bottom-right corner.</p>	IRR_Hh*b*h2*b2
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b. The bottom edge is horizontal with length b_2. The right side is a vertical line of height h. The left side is a vertical line of height h. A diagonal line connects the top-left corner to the bottom-right corner.</p>	IRR_Ih*b*b2
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b. The bottom edge is horizontal with length b_2. The right side is a vertical line of height h. The left side is a vertical line of height h. A diagonal line connects the top-right corner to the bottom-left corner.</p>	IRR_Jh*b*b2
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b_1. The bottom edge is horizontal with length B. The left side is a vertical line of height H. The right side is a vertical line of height h_1. The shape is an octagon with horizontal top and bottom edges and vertical left and right edges.</p>	OCTB*b1-H*h1
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length c. The bottom edge is horizontal with length b. The left side is a vertical line of height h. The right side is a vertical line of height h. A diagonal line connects the top-right corner to the bottom-left corner.</p>	REC_Ah-b
<p>The diagram shows a cross-section of an irregular beam. The top edge is horizontal with length b. The bottom edge is horizontal with length b_1. The right side is a vertical line of height h. The left side is a vertical line of height h. A diagonal line connects the top-left corner to the bottom-right corner.</p>	REC_Bh-b-b1

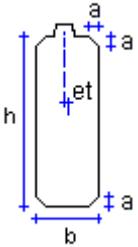
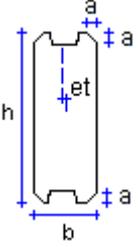
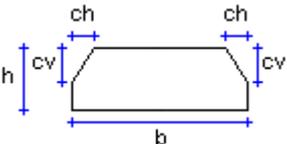
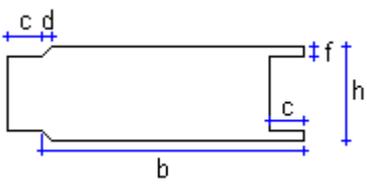
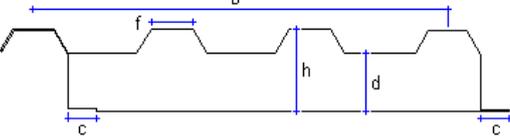
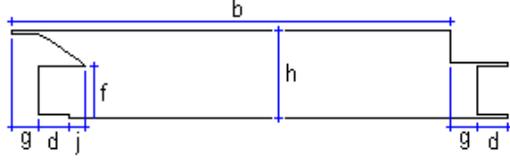
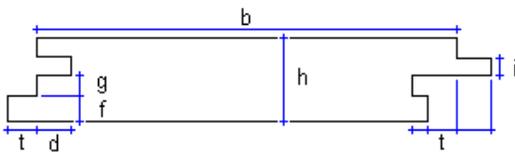
	REC_Ch-b-b1
	REC_Dh-b-b2
	REC_Eh-b
	REC_Fh-b
	REC_Gh-b
	REC_Hh-b
	REC_I a-b*h
	TRI_Ah-b

 <p>Diagram of an equilateral triangle with side length a_1.</p>	TRI_Ba1
 <p>Diagram of a triangle with base b and height h.</p>	TRI_Cb-h
 <p>Diagram of a right-angled triangle with base b and height h.</p>	TRI_Dh*b
 <p>Diagram of a trapezoid with top width b_2, bottom width b, and height h.</p>	TRI_Eb*h*h2*b2

9.23 Panels

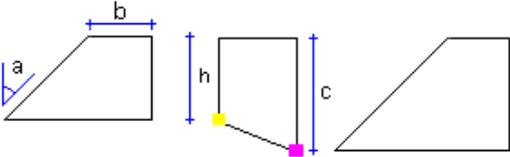
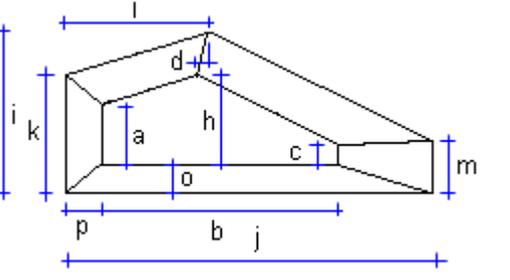
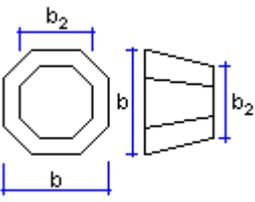
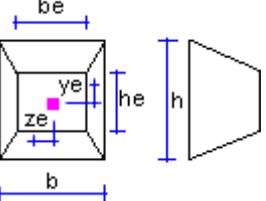
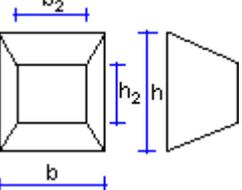
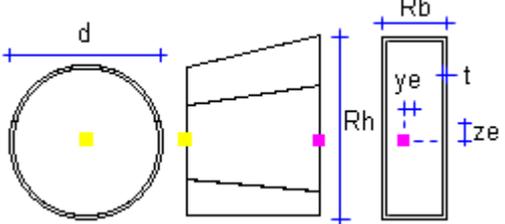
 <p>Diagram of a panel with height h, width b, and chamfered top corners with radius c.</p>	PNL_Ah*b
 <p>Diagram of a panel with height h, width b, and chamfered top and bottom corners with radius a.</p>	PNL_Bh*b

	PNL_Ch*b-a-ht*bt
	PNL_Dh*b-a-ht*bt
	PNL_Eh*b-a-ht*bt
	PNL_Fh*b-a-ht*bt
	PNL_Gh*b
	PNL_Hh*b-a-ht

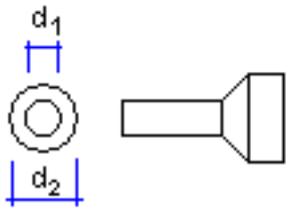
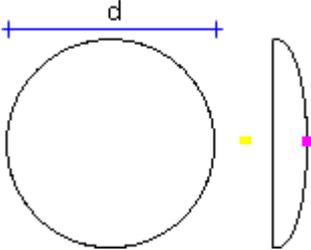
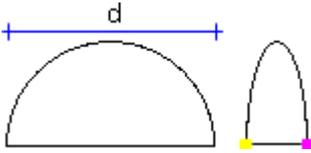
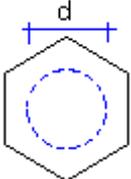
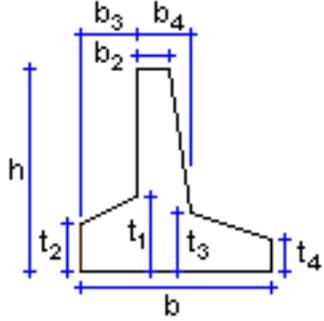
	PNL_Ih*b-a-ht*bt
	PNL_Jh*b-a-ht*bt
	PNL_Kh*b
	PNL_Lh-b-c-f
	PNL_Mh-b-c-f-d
	PNL_Nh-b-d-f-g-j
	PNL_Oh-b-d-f-g-i-t

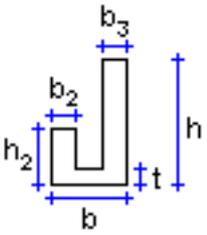
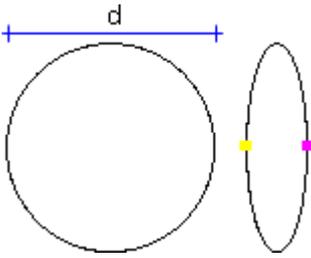
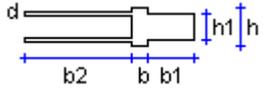
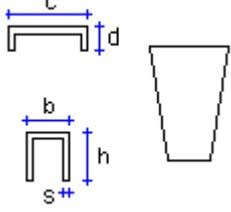
9.24 Variable cross sections

	HEXRECTh-b-br-hr
	HXGONb
	OBLINCLh1-h2-h3-h4-b
	OBLRIDh1*b1*b2-h2-h3-l2-l1
	OBLVAR_Ah1*b1*b2-h2
	OBLVAR_Bh1-h2-b
	OBLVAR_Ch-b-a-i-j-k-m-n

	OBLVAR_Dh-c-b
	OBLVAR_Eh-b-a-c-d-i-j-k-l-m-p-o
	OCTAGONb-b2
	PRMDASH*b-he*be PL_Vh*b-he*be
	PRMDh*b-h2*b2
	ROUNDRECTd-Rb*Rh-t*ye-ze

9.25 Others

 <p>Technical drawing of a flange. It shows a top view with an outer diameter dimension d_1 and an inner diameter dimension d_2. A side view shows a cylindrical neck of diameter d_2 attached to a larger flange face of diameter d_1.</p>	BLKSd1-d2
 <p>Technical drawing of a circular cap. The top view is a circle with diameter dimension d. The side view shows a curved profile with a yellow square at the base and a pink square at the top edge.</p>	CAPd
 <p>Technical drawing of a hemispherical cap. The top view is a semi-circle with diameter dimension d. The side view shows a hemispherical profile with a yellow square at the base and a pink square at the top edge.</p>	HEMISPHERd
 <p>Technical drawing of a hexagonal nut. The top view shows a regular hexagon with a dashed circle inside representing the hole. The diameter of the hole is dimensioned as d.</p>	NUT_Md
 <p>Technical drawing of a stepped profile. The total height is h and the total width at the base is b. The profile has four vertical thicknesses: t_2 on the left, t_1 in the middle, t_3 on the right, and t_4 at the bottom right. The widths at different levels are b_3 (top), b_2 (middle), b_4 (bottom), and b (base).</p>	$RCRWh*b-b2*b3-b4-t1*t2-t3*t4$

	SKh*b-h2-t-b2-b3
	SPHEREd
	STBb-h-h1-b1-b2-d
	STEPh-b*h1-b1-s

10 Modeling settings

This section provides more information about the various settings you can modify in Tekla Structures.

Click the links below to find out more:

- [View and representation settings \(page 918\)](#)
- [Part position settings \(page 923\)](#)
- [Numbering settings \(page 923\)](#)
- [Reinforcement settings \(page 927\)](#)

10.1 View and representation settings

This section provides more information about specific view and representation settings.

Click the links below to find out more:

[View properties \(page 44\)](#)

[Grid view properties \(page 45\)](#)

[Display settings \(page 918\)](#)

[Color settings for object groups \(page 921\)](#)

[Transparency settings for object groups \(page 922\)](#)

Display settings

Use the **Display** dialog box to define which object types Tekla Structures displays and how they appear in the model. Some of these settings may affect system performance.

Option	Description
Settings	
Parts	<p>Defines how parts are displayed.</p> <p>Fast uses a rapid drawing technique that displays internal hidden edges, but skips cuts. The setting does not automatically affect already modeled parts. When you switch this setting on, the fast representation mode will be applied only to newly created parts and to parts that are displayed with the Show with Exact Lines command.</p> <p>Exact displays the cuts, but hides the internal hidden lines of parts.</p> <p>Reference line shows parts as sticks (page 306). This option increases display speed significantly, when viewing the entire model, or large parts of it.</p> <p>Cast-in-place concrete structures can be displayed as Pours, or as Parts that can be Merged or Separated. For more information, see View cast-in-place concrete structures (page 404).</p>
Bolts	<p>Defines how bolts are displayed.</p> <p>Fast displays the axis and a cross to represent the bolt head. This is the recommended representation mode for bolts, because it increases display speed significantly and consumes less system memory.</p> <p>Exact shows bolts, washers, and nuts as solid objects.</p>
Holes	<p>Defines how holes are displayed.</p> <p>Fast only displays the circle in the first plane. When using this option, Tekla Structures always displays fast holes on the first part (counting from the head of the bolt). If there are slotted holes in any of the parts, a slotted hole is displayed on the first part, even if the hole in that part is</p>

Option	Description
	<p>not slotted. The new slotted hole has the same size and rotation as the first slotted hole (counting from the head of the bolt).</p> <p>Holes that are outside a part are always displayed as fast holes.</p> <p>Exact shows holes as solid objects.</p> <p>Exact slotted holes only displays slotted holes in exact mode and ordinary holes in fast mode.</p>
Welds	<p>Defines how welds are displayed.</p> <p>Fast displays a symbol for welds.</p> <p>Exact shows welds as solid objects and displays the weld symbols. When you select welds, the weld marks are displayed.</p> <p>Exact - no weld mark shows welds as solid objects but does not display the weld symbols, nor the weld marks when you select welds.</p> <p>For more information, see Set the visibility and appearance of welds (page 358).</p>
Construction planes	<p>Defines how construction planes are displayed.</p>
Reinforcing bars	<p>Defines how reinforcement objects are displayed.</p> <p>Fast displays the shape of reinforcement meshes using an outline polygon and a diagonal line. Single reinforcing bars and bar groups are displayed as solid objects.</p> <p>Exact shows reinforcing bars, bar groups, and reinforcement meshes as solid objects.</p>
Advanced	
Part label	<p>See Show part information by using part labels (page 320).</p>
Point size	<p>Defines the size and appearance of points in views. Also affects the size</p>

Option	Description
	<p>and appearance of the handles, together with XS_HANDLE_SCALE.</p> <p>In model increases the point size on the screen when you zoom in. Shows points and handles as 3D cubes:</p>  <p>In view does not increase the point size. Shows points and handles as flat 2D objects:</p> 

See also

[Set the visibility and appearance of model objects \(page 582\)](#)

[Change the rendering of parts and components \(page 584\)](#)

[Set the visibility of pour breaks \(page 417\)](#)

Color settings for object groups

Use the **Object Representation** dialog box to define the color of object groups.

Option	Description
As is	<p>The current color is used.</p> <p>If the object belongs to one of the object groups defined in the following rows, its color is defined by the settings that the object group in question has on that row.</p>
Colors	Select color from the list.
Color by class	<p>All parts are colored according to their Class property. See Change the color of a model object (page 594).</p>

Option	Description
Color by lot Color by phase	Parts belonging to different lots or phases get different colors according to the lot or phase number: 1  2  3  4  5  6  7  8  9  10  11  12  13  14 
Color by analysis type	Displays parts according to the member analysis class.
Color by analysis utility check	Displays parts according to the utilization ratio in analysis.
Color by attribute	Displays parts in different colors according to the values of a user-defined attribute.

See also

[Change the color and transparency of model objects \(page 593\)](#)

Transparency settings for object groups

Use the **Object Representation** dialog box to define the transparency of object groups.

Option	Description
As is	The current visibility. If the object belongs to any object group whose visibility and color settings have been defined, the settings will be read from that object group.
Visible	Object is shown in the views.
50% transparent	Object is transparent in the views.
70% transparent	
90% transparent	
Hidden	Object is not shown in the views.

See also

[Change the color and transparency of model objects \(page 593\)](#)

10.2 Part position settings

This section provides more information about specific part position settings. These settings can be modified in the **Position** and **End offset** sections in the part property pane, or by using the contextual toolbar.

Click the links below to find out more:

[Part position on the work plane \(page 308\)](#)

[Part rotation \(page 309\)](#)

[Part position depth \(page 310\)](#)

[Part vertical position \(page 312\)](#)

[Part horizontal position \(page 313\)](#)

[Part end offsets \(page 315\)](#)

10.3 Numbering settings

This section provides more information about specific numbering settings.

Click the links below to find out more:

- [General numbering settings \(page 923\)](#)
- [Weld numbering settings \(page 925\)](#)
- [Control number settings \(page 925\)](#)

General numbering settings

Use the **Numbering Setup** dialog box to view and modify some general numbering settings.

Setting	Description
Renumber all	All parts get a new number. All information on previous numbers is lost.
Re-use old numbers	Tekla Structures reuses the numbers of parts that have been deleted. These numbers may be used to number new or modified parts.

Setting	Description
Check for standard parts	<p>If a separate standard-part model has been set up, Tekla Structures compares the parts in the current model to those in the standard-part model.</p> <p>If the part to be numbered is identical to a part in the standard-part model, Tekla Structures uses the same part number as in the standard-part model.</p>
Compare to old	The part gets the same number as a previously numbered similar part.
Take new number	The part gets a new number even if a similar numbered part already exists.
Keep number if possible	<p>Modified parts maintain their previous numbers if possible. Even if a part or assembly becomes identical with another part or assembly, the original position number is maintained.</p> <p>For example, you might have two different assemblies, B/1 and B/2, in the model. Later on you modify B/2 so that it becomes identical with B/1. If the Keep number if possible option is used, B/2 will maintain its original position number when you renumber the model.</p>
Synchronize with master model (save-numbering-save)	Use this setting when working in multi-user mode. Tekla Structures locks the master model and performs a save, numbering, and save sequence, so that all other users can continue working during the operation.
Automatic cloning	<p>If the main part of a drawing is modified and therefore gets a new assembly position, the existing drawing is automatically assigned to another part of the position.</p> <p>If the modified part moves to an assembly position that does not have a drawing, the original drawing is automatically cloned to reflect the changes in the modified part.</p>
Holes	The location, size, and number of holes affects numbering.
Part name	The part name affects numbering.
Beam orientation	The orientation of beams affects numbering of assemblies.
Column orientation	The orientation of columns affects numbering of assemblies.
Assembly name	The assembly name affects numbering.
Assembly phase	<p>Only enabled when XS_ENABLE_PHASE_OPTION_IN_NUMBERING is set to <code>TRUE</code>.</p> <p>The assembly phase affects numbering.</p>
Reinforcing bars	Reinforcing bars affect numbering.

Setting	Description
Embedded objects	Sub-assemblies affect the numbering of cast units.
Surface treatment	Surface treatments affect the numbering of assemblies.
Welds	Welds affect the numbering of assemblies.
Tolerance	Parts get the same number if their dimensions differ less than the value entered in this box.
Assembly position sort order	See Number assemblies and cast units (page 638) .

See also

[Adjust the numbering settings \(page 636\)](#)

[Create a standard-part model \(page 662\)](#)

[Numbering examples \(page 656\)](#)

Weld numbering settings

Use the **Weld Numbering** dialog box to view and modify the weld numbering settings. The weld number is displayed in drawings and weld reports.

Option	Description
Start number	The number from which the numbering starts. Tekla Structures automatically suggests the following free number as the start number.
Apply for	Defines which objects are affected by the change. All welds changes the number of all welds in the model. Selected welds changes the number of the selected welds without affecting others.
Renumber also welds that have a number	Tekla Structures replaces existing weld numbers.
Re-use numbers of deleted welds	If some welds have been removed, Tekla Structures uses their numbers when numbering other welds.

See also

[Number welds \(page 640\)](#)

Control number settings

Use the **Create control numbers (S9)** dialog box to view and modify the control number settings.

Option	Description
Numbering	<p>Defines which parts get control numbers.</p> <p>All creates consecutive numbers for all parts.</p> <p>By numbering series creates control numbers for parts in a specific numbering series.</p>
Assembly/Cast unit numbering series	<p>Defines the prefix and start number of the numbering series for which to create control numbers.</p> <p>Needed only with the By numbering series option.</p>
Start number of control numbers	The number from which the numbering starts.
Step value	Defines the interval between two control numbers.
Renumber	<p>Defines how to treat parts that already have control numbers.</p> <p>Yes replaces the existing control numbers.</p> <p>No keeps the existing control numbers.</p>
First direction	Defines in what order to assign control numbers.
Second direction	
Third direction	
Write UDA to	<p>Defines where to save the control numbers.</p> <p>Assembly saves the control numbers to the user-defined attributes of assemblies or cast units.</p> <p>Main part saves the control numbers to the user-defined attributes of assembly or cast unit main parts.</p> <p>The control number appears on the Parameters tab.</p>

See also

[Control numbers \(page 646\)](#)

10.4 Reinforcement settings

This section provides more information about the various reinforcement settings you can modify in Tekla Structures.

Click the links below to find out more:

[Reinforcing bar and bar group properties \(page 927\)](#)

[Reinforcement mesh properties \(page 929\)](#)

[Rebar set properties \(page 933\)](#)

[Reinforcement strand properties \(page 946\)](#)

Reinforcing bar and bar group properties

Use the **Single rebar** and **Rebar group** properties to view and modify the properties of reinforcing bars and reinforcing bar groups. The file name extension of the properties file is:

- `.rbr` for [bars \(page 451\)](#)
- `.rbg` for [groups \(page 454\)](#)
- `.rci` for [circular groups \(page 463\)](#)
- `.rcu` for [curved groups \(page 461\)](#)

General, Hooks, Cover thickness, Custom properties

The following properties are available for single reinforcing bars and reinforcing bar groups:

Option	Description	
Name	User-definable name of the bar. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bar.	Size-grade-radius combinations are predefined in the reinforcing bar catalog. Click the ... button to open the Select Reinforcing Bar dialog box. The dialog box
Size	Diameter of the bar. Depending on the environment, the nominal diameter of the bar, or a mark that defines the diameter.	

Option	Description	
Bending radius	<p>Internal radius of the bends in the bar.</p> <p>You can enter a separate value for each bar bend. Separate the values with spaces.</p> <p>Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.</p>	<p>shows the available bar sizes for the chosen grade. You can also select whether the bar is a main bar or a stirrup or tie.</p> <p>The <code>rebar_database.inp</code> file contains the predefined reinforcing bar catalog entries.</p>
Class	<p>Use to group reinforcement.</p> <p>For example, you can display bars of different classes in different colors.</p>	
Numbering	<p>Mark series of the reinforcing bar.</p>	
Hook type	<p>Shape of the hook.</p>	<p>The <code>rebar_database.inp</code> file contains the predefined minimum bending radius and minimum hook length for all standard hooks.</p> <p>See Add hooks to reinforcing bars (page 508).</p>
Angle	<p>Angle of a custom hook.</p>	
Radius	<p>Internal bending radius of a standard hook or custom hook.</p>	
Length	<p>Length of the straight part of a standard or custom hook.</p>	
Cover thickness on plane	<p>Distances from the part surfaces to the bar on the same plane as the bar.</p>	<p>See Define the reinforcement cover thickness (page 510).</p>
Cover thickness from plane	<p>Distance from the part surface to the bar, or to the bar end, perpendicular to the bar plane.</p>	
Start	<p>Concrete cover thickness or leg length at the first end of the bar.</p>	
End	<p>Concrete cover thickness or leg length at the second end of the bar.</p>	

Option	Description
Custom properties	<p>You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists.</p> <p>You can use the values of user-defined attributes in reports and drawings.</p> <p>You can also change the name of the fields, and add new ones, by editing the <code>objects.inp</code> file. See Customizing user-defined attributes.</p>

Rebar group type, Distribution, Creation

The following properties are available for:

- reinforcing bar groups, including [tapered \(page 465\)](#) groups
- curved reinforcing bar groups
- circular reinforcing bar groups

Option	Description	
Rebar group type	What is the type of the group.	See Create a tapered or spiral reinforcing bar group (page 465) .
Number of cross sections		
Creation method	How the bars are spaced.	See Distribute bars in a reinforcing bar group (page 500) .
Number of reinforcing bars		
Target spacing value		
Exact spacing value		
Exact spacing values		
Exclude	Which bars are omitted from the group.	See Delete bars from a reinforcing bar group (page 502) .

Reinforcement mesh properties

Use the **Rebar mesh** properties to view and modify the properties of reinforcement meshes. The file name extension of a reinforcement mesh properties file is `.rbm`.

Option	Description
Numbering	Mark series of the mesh.
Name	User-definable name of the mesh. Tekla Structures uses mesh names in reports and drawing lists.
Class	Use to group reinforcement. For example, you can display bars of different classes in different colors.
Mesh type	Shape of the mesh. Select Polygon , Rectangle , or Bent .
Cross bar location	Define whether the crossing bars are located above or below the longitudinal bars.
Cut by father part cuts	Define whether the polygon or part cuts in the part also cut the mesh.
Mesh	To create a Standard mesh, click the ... button and select a mesh from the mesh catalog. The properties of standard meshes are defined in the <code>mesh_database.inp</code> file. To create a custom mesh (page 473) , select the Custom mesh option and define the properties (page 931) .
Grade	Steel grade of the bars in the mesh. Available for custom meshes.
Bending radius	Internal radius of the bends in the bar. Available for bent meshes.
Hooks	See Add hooks to reinforcing bars (page 508) . Available for bent meshes.
Cover thickness on plane	Distance from the part surface to the main bars on the same plane as the bars.
Cover thickness from plane	Distance from the part surface to the bar, or bar end, perpendicular to the bar plane.
Start	Thickness of concrete cover or leg length from the mesh starting point. Available for rectangular and bent meshes.

Option	Description
End	Thickness of concrete cover or leg length at the end point of the bar. Available for bent meshes.
Custom properties	You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists. You can use the values of user-defined attributes in reports and drawings. You can also change the name of the fields, and add new ones, by editing the <code>objects.inp</code> file. See Define and update user-defined attributes (UDAs).

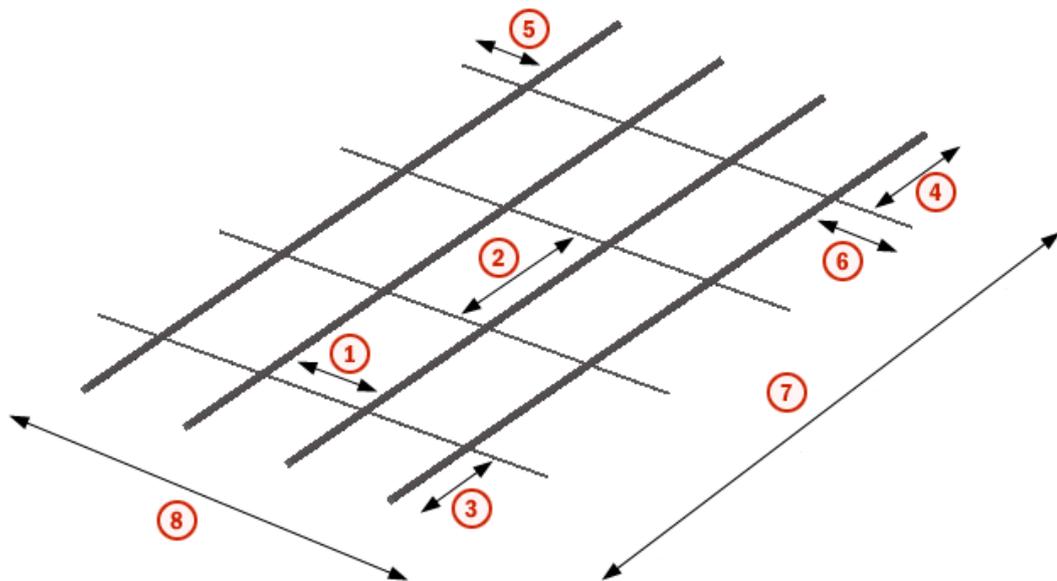
See also

[Create a reinforcement mesh \(page 467\)](#)

Custom reinforcement mesh properties

Use the **Rebar mesh** properties to view and modify the properties of customized reinforcement meshes. The file name extension of a reinforcement mesh properties file is `.rbm`.

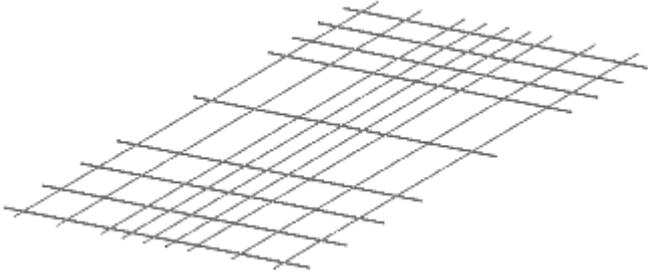
You can define the following properties for the [customized reinforcement meshes \(page 473\)](#):



1. Longitudinal distance
2. Cross distance

3. Longitudinal left overhang
4. Longitudinal right overhang
5. Cross left overhang
6. Cross right overhang
7. Length
8. Width

Option	Description
Spacing method	<p>Define how the mesh bars are distributed.</p> <ul style="list-style-type: none"> • Same distance for all: Use to create meshes with evenly-spaced bars. Tekla Structures distributes as many bars as possible for the length of Length or Width, using the Distances and Left overhang values. The Right overhang is calculated automatically, and it cannot be zero. • Multiple varying distances: Use to create meshes with unevenly-spaced bars. Tekla Structures calculates the Width and Length based on the Distances, the Left overhang and the Right overhang values. If you do not change any of the values, the spacing method changes back to Same distance for all.
Distances	<p>Spacing values of longitudinal or crossing bars.</p> <p>If you select the Multiple varying distances spacing method, enter all spacing values, separated by spaces. You can use multiplication to repeat spacing values. For example:</p> <p>2*150 200 3*400 200 2*150</p> <p>You can create meshes with unevenly-spaced bars. You can also define a different bar size or multiple different bar sizes for the longitudinal bars and the crossing bars.</p> <p>Multiple bar sizes enable pattern creation. For example, if you enter bar diameters 20 2*6 in the longitudinal direction, Tekla Structures creates a pattern with one size 20 bar and two size 6 bars. This pattern can be repeated in the mesh along the longitudinal direction.</p>

Option	Description
	
Left overhang Right overhang	Extensions of crossing bars over the outermost longitudinal bars. Extensions of longitudinal bars over the outermost crossing bars.
Diameters	Diameter or size of the longitudinal or crossing bars. You can define multiple diameters for the bars in both directions. Enter all the diameter values, separated by spaces. You can use multiplication to repeat diameter values. For example, 12 2*6 in longitudinal direction and 6 20 2*12 in crossing direction.
Width	Length of crossing bars.
Length	Length of longitudinal bars.
Grade	Steel grade of the bars in the mesh.

See also

[Create a reinforcement mesh \(page 467\)](#)

[Reinforcement mesh properties \(page 929\)](#)

Rebar set properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar sets. The file name extension of the properties file is `.rst`.

Attributes

Option	Description
Numbering	Numbering series of the bars.
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.

Option	Description	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the reinforcing bar catalog. Click the ... button in the property pane to open the Select Reinforcing Bar dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars or stirrups or ties. The <code>rebar_database.inp</code> file contains the predefined reinforcing bar catalog entries.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine. Automatic values are shown in square brackets, for example [120.00].	
Class	Used to group reinforcement. For example, you can display bars of different classes in different colors.	
Layer order number	Defines the order of the bar layers. Enter a number, or use the arrow buttons to change the number. The smaller the number, the closer to the concrete surface the bar layer is. You can use both positive and negative numbers. If you do not define the layer order numbers, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. Note that if you copy properties (page 109) from one rebar set to another, the layer order number is not copied.	

Spacing properties

The file name extension of the spacing zone properties file is `.rst.zones`.

Option	Description	
Start offset	The offsets at the start and end of a rebar set.	
End offset	<p>By default, Tekla Structures calculates the offset values according to the concrete cover settings and bar diameter. Automatic values are shown in square brackets, for example [32.00].</p> <p>You can define whether an offset value is an Exact value or a Minimum value. If you select Minimum, the actual offset value can be bigger, depending on the spacing properties. In model views, both the actual and minimum values are shown, for example 50.00 (> 32.00), where the minimum value is in parentheses.</p>	
Length	The length of each spacing zone as an absolute value in the current length units (Absolute), or as a percentage of the total length of all spacing zones (Relative).	<p>Only two of the three properties, Length, Number of spaces, and Spacing, can be set to Absolute or Exact at the same time.</p> <p>At least one of the spacing properties needs to be flexible and yield to create a practical spacing combination. In model views, the yielding value is shown in red.</p>
Number of spaces	Defines to how many spaces a spacing zone is divided. You can define a flexible number, which Tekla Structures aims at (Target), or a fixed number of spaces (Exact).	
Spacing	The spacing value of each spacing zone. You can define a flexible number, which Tekla Structures aims at (Target), or a fixed number of spaces (Exact).	

Advanced: Rounding

Option	Description
Straight bars	Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable
First and last legs	
Intermediate legs	

Option	Description
	number according to the rounding accuracy.
Rounding up at splitters	At splitter locations, define how much the bar lengths can be rounded up.

Advanced: Step tapering

Option	Description
Type	Define whether the bars are step tapered, and how the tapering steps are created. The options are None , Distance , and Number of bars . If you select the Number of bars option, enter the number of bars in one tapering step.
Straight bars	If you select the Distance option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.
First and last legs	
Intermediate legs	

Custom properties

Click the **More** button to open the user-defined attributes dialog box. The file name extension of the user-defined attributes file is `.rst.more`.

See also

- [Create a rebar set \(page 428\)](#)
- [Modify a rebar set \(page 478\)](#)
- [Secondary guideline properties \(page 936\)](#)
- [Leg face properties \(page 938\)](#)
- [Property modifier properties \(page 938\)](#)
- [End detail modifier properties \(page 941\)](#)
- [Splitter properties \(page 943\)](#)

Secondary guideline properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set secondary guidelines.

Spacing properties

If you want a secondary guideline to have the same spacing properties as the primary guideline, select **Yes** from the **Inherit from primary** list in the property pane.

If you want to define the secondary guideline spacing properties independently from the primary guideline, select **No** from the **Inherit from primary** list, and then modify the following spacing properties as needed:

Option	Description	
Start offset	The offsets at the start and end of a rebar set.	
End offset	<p>By default, Tekla Structures calculates the offset values according to the concrete cover settings and bar diameter. Automatic values are shown in square brackets, for example [32.00].</p> <p>You can define whether an offset value is an Exact value or a Minimum value. If you select Minimum, the actual offset value can be bigger, depending on the spacing properties. In model views, both the actual and minimum values are shown, for example 50.00 (> 32.00), where the minimum value is in parentheses.</p>	
Length	The length of each spacing zone as an absolute value in the current length units (Absolute), or as a percentage of the total length of all spacing zones (Relative).	<p>Only two of the three properties, Length, Number of spaces, and Spacing, can be set to Absolute or Exact at the same time.</p> <p>At least one of the spacing properties needs to be flexible and yield to create a practical spacing combination. In model views, the yielding value is shown in red.</p>
Number of spaces	Defines to how many spaces a spacing zone is divided. You can define a flexible number, which Tekla Structures aims at (Target), or a fixed number of spaces (Exact).	
Spacing	The spacing value of each spacing zone. You can define a flexible number, which Tekla Structures aims at (Target), or a fixed number of spaces (Exact).	

See also

[Modify a rebar set locally using modifiers \(page 486\)](#)

[Rebar set properties \(page 933\)](#)

Leg face properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set leg faces.

Attributes

Option	Description
Additional offset	Distance between the leg face and the bars. A negative value moves the bars outside the concrete.
Flip bar side	Shows if the bars are flipped over to the other side of the leg face (Yes) or not (No). The default value is No .
Layer order number	Defines the order of the bar layers. Enter a number, or use the arrow buttons to change the number. The smaller the number, the closer to the concrete surface the bar layer is. You can use both positive and negative values. If you do not define the layer order numbers, Tekla Structures arranges the bar layers according to their creation order. The bar layer that is created first is closest to the concrete surface. Note that if you copy properties (page 109) from one leg face to another, the layer order number is not copied.

See also

[Modify a rebar set using leg faces \(page 480\)](#)

[Rebar set properties \(page 933\)](#)

Property modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set property modifiers. The file name extension of the properties file is `.rst_pm`.

General

Option	Description
Bars affected	<p>Select how many bars can be modified in the same location:</p> <ul style="list-style-type: none"> • 1/1 = all bars are modified in the same cross section. • 1/2 = every second bar is modified in the same cross section. • 1/3 = every third bar is modified in the same cross section. • 1/4 = every fourth bar is modified in the same cross section.
First affected bar	<p>Define which is the first bar to be modified, starting from the first end of the modifier.</p> <p>Enter a positive number, or use the arrow buttons to change the number.</p>

Attributes

Option	Description	
Numbering	Numbering series of the bars.	
Name	User-definable name of the bars. Tekla Structures uses bar names in reports and drawing lists, and to identify bars of the same type.	
Grade	Steel grade of the bars.	Grade-size-radius combinations are predefined in the reinforcing bar catalog. Click the ... button in the property pane to open the Select Reinforcing Bar dialog box. The dialog box shows the available bar sizes for the chosen grade. You can also select whether the bars are main bars or stirrups or ties. The <code>rebar_database.inp</code> file contains the predefined reinforcing bar catalog entries.
Size	Diameter of the bars. Depending on the environment, the nominal diameter of the bars, or a mark that defines the diameter.	
Bending radius	Internal radius of the bends in the bars. You can enter a separate value for each bar bend. Separate the values with spaces. Bending radius complies with the design code you are using. Main bars, stirrups, ties, and hooks usually have their own minimum internal	

Option	Description
	<p>bending radii, which are proportional to the diameter of the reinforcing bar. The actual bending radius is normally chosen to suit the size of the mandrels on the bar-bending machine.</p> <p>Automatic values are shown in square brackets, for example [120.00].</p>
Class	<p>Used to group reinforcement.</p> <p>For example, you can display bars of different classes in different colors.</p>

Advanced: Rounding

Option	Description
Straight bars	<p>Define whether the lengths of straight bars, first and last legs, and intermediate legs are rounded, and whether the bar lengths are rounded up, down, or to the nearest suitable number according to the rounding accuracy.</p>
First and last legs	
Intermediate legs	
Rounding up at splitters	<p>At splitter locations, define how much the bar lengths can be rounded up.</p>

Advanced: Step tapering

Option	Description
Type	<p>Define whether the bars are step tapered, and how the tapering steps are created.</p> <p>The options are None, Distance, and Number of bars.</p> <p>If you select the Number of bars option, enter the number of bars in one tapering step.</p>
Straight bars	<p>If you select the Distance option, enter the tapering step values for straight bars, first and last legs, and intermediate legs.</p>
First and last legs	
Intermediate legs	

Custom properties

Click the **More** button to open the user-defined attributes dialog box. The file name extension of the user-defined attributes file is `.rst_pm.more`.

See also

[Modify a rebar set locally using modifiers \(page 486\)](#)

[Rebar set properties \(page 933\)](#)

End detail modifier properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set end detail modifiers. The file name extension of the property file is `.rst_edm`.

General

Option	Description
Bars affected	Select how many bars can be modified in the same location: <ul style="list-style-type: none">• 1/1 = all bars are modified in the same cross section.• 1/2 = every second bar is modified in the same cross section.• 1/3 = every third bar is modified in the same cross section.• 1/4 = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.

Hook

Option	Description	
Hook type	Shape of the hook.	The <code>rebar_database.inp</code> file contains the predefined minimum bending radius and minimum hook length for all standard hooks. See Add hooks to reinforcing bars (page 508) .
Angle	Angle of a custom hook.	
Radius	Internal bending radius of a standard hook or custom hook.	
Length	Length of the straight part of a standard or custom hook.	

Option	Description	
Hook rotation	Rotation angle of a hook out of the bar plane. Use to create 3D bars.	For example: 

Length adjustment

Option	Description
Adjustment type	<p>Select whether and how the bar length is adjusted (extended or shortened).</p> <ul style="list-style-type: none"> • No adjustment: Bar length is not adjusted. • End offset: Bar length is adjusted according to the specified end offset. Use this option to keep the leg faces on the concrete faces and adaptive to the concrete faces, but still to extend or shorten the bar ends. • Leg length: Bar length is adjusted according to the specified leg length.
Length	<p>Depending on the adjustment type, the length of the end offset or leg.</p> <p>With end offset, enter a positive value to extend the bars, or a negative value to shorten the bars.</p> <p>With leg length, enter a positive value to set the leg length.</p>

End preparations

Option	Description
Method	<p>Select the end method of the bars. The options are:</p> <ul style="list-style-type: none"> • Coupler • Coupler female • Coupler male • Threaded • Anchor
Type	<p>Select the end method type. The options are:</p> <ul style="list-style-type: none"> • Standard

Option	Description
	<ul style="list-style-type: none"> • Position • Bridging • Transition • Bolt • Weldable
Product	Product name of the end detail. Can be shown in reports.
Code	Product code of the end detail. Can be shown in reports.
Thread type	Enter the type of the threading.
Threaded length	Length of the threading from the bar end.
Extra fabrication length	Additional length needed with some threading methods. Can be shown in reports, but does not affect the total length of the bar.

Custom properties

Click the **More** button to open the user-defined attributes of the rebar set end detail modifiers. The file name extension of the user-defined attribute file is `.rst_edm.more`.

See also

[Modify a rebar set locally using modifiers \(page 486\)](#)

[Rebar set properties \(page 933\)](#)

Splitter properties

Use the property pane or the contextual toolbar to view and modify the properties of the rebar set splitters. The file name extension of the properties file is `.rst_sm`.

Some of the following settings depend on the splitter direction. An arrowhead symbol  close to the midpoint of each splitter indicates the direction and the left and right sides of the splitter. The arrow points from the start towards the end of the splitter.

General

Option	Description
Bars affected	Select how many bars can be modified in the same location: <ul style="list-style-type: none"> • 1/1 = all bars are modified in the same cross section. • 1/2 = every second bar is modified in the same cross section. • 1/3 = every third bar is modified in the same cross section. • 1/4 = every fourth bar is modified in the same cross section.
First affected bar	Define which is the first bar to be modified, starting from the first end of the modifier. Enter a positive number, or use the arrow buttons to change the number.

Option	Description
Split type	Select Lapping or Cranking .
Split offset	Defines how far from the splitter the split is created. Positive values move the split to the right side and negative values to the left side of the splitter.

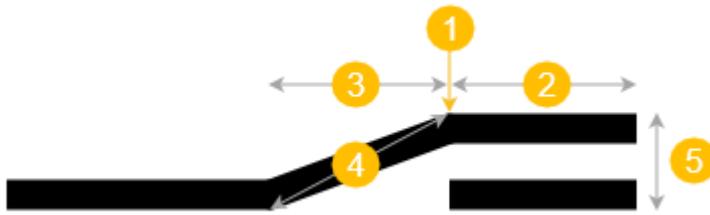
Lapping

These properties are available when **Split type** is **Lapping**.

Option	Description
Lapping type	Select Standard lapping or Custom lapping .
Lap length	With custom lapping, enter the length of the lap splice.
Lap side	Select the side of the lap splice from the splitter: <ul style="list-style-type: none"> • Lap left • Lap right • Lap middle
Lap placement	Select whether the lapping bars are parallel to each other, or on top of each other.

Cranking

These properties are available when **Split type** is **Cranking**.



(1) = Location of the splitter

Option	Description
Cranking type	Select Standard cranking or Custom cranking .
Crank straight length	With custom cranking, enter the length of the straight segment of the crank. This is (2) in the image above.
Cranked length	With custom cranking, select how the length of the cranked segment is defined, and enter the needed distance or multiplier value. The options are: <ul style="list-style-type: none"> • Diagonal ratio: Multiplier of the bar diameter in diagonal direction • Diagonal distance: Total length of the cranked segment in diagonal direction This is (4) in the image above. • Horizontal ratio: Multiplier of the bar diameter in horizontal direction • Horizontal distance: Total length of the cranked segment in horizontal direction This is (3) in the image above.
Cranked offset	With custom cranking, enter the offset distance of the straight segment of the crank. This is (5) in the image above. The default value is 2 * actual bar diameter.
Crank side	Select to which side of the splitter the crank is created, Left or Right .
Crank rotation	Defines to which angle the crank is rotated.

Staggering

Option	Description
Stagger type	Select whether and to which direction the splices are staggered. The options are: <ul style="list-style-type: none"> • No stagger

Option	Description
	<ul style="list-style-type: none"> • Stagger left • Stagger right • Stagger middle
Stagger offset	The offset of the adjacent bars if they are staggered.

See also

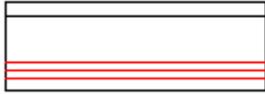
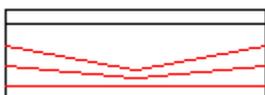
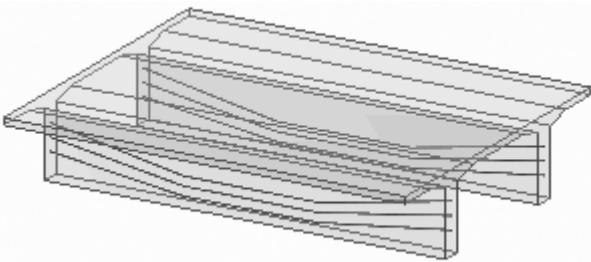
[Modify a rebar set locally using modifiers \(page 486\)](#)

[Rebar set properties \(page 933\)](#)

Reinforcement strand properties

Use the **Strand pattern** properties to view and modify the properties of strands. The file name extension of the properties file is `.rbs`.

Option	Description
General	
Numbering	Mark series of the strand.
Name	User-definable name of the strand. Tekla Structures uses strand names in reports and drawing lists, and to identify strands of the same type.
Grade	Steel grade of the strand.
Size	Diameter of the strand. Depending on the environment, the nominal diameter of the strand, or a mark that defines the diameter.
Bending radius	Internal radius of the bends in the strand. You can enter a separate value for each bend. Separate the values with spaces.
Class	Use to group reinforcement. For example, you can display strands of different classes in different colors.
Pull per strand	Pre-stress load per strand (kN).
Number of cross sections	Number of cross sections of the strand pattern. For example:

Option	Description
	<ul style="list-style-type: none"> <li data-bbox="651 271 1362 338">• Number of cross sections along strand profile = 1:  <li data-bbox="651 510 1362 577">• Number of cross sections along strand profile = 2:  <li data-bbox="651 750 1362 817">• Number of cross sections along strand profile = 3:  <li data-bbox="651 990 1362 1057">• Number of cross sections along strand profile = 4:  <p data-bbox="651 1238 1278 1305">In this double-tee beam, the number of cross sections is 4:</p> 
Debonding	
Debonded strands	Enter the strand number. The strand number is the selection order number of the strand.
From start Middle to start	Enter the length of the debonding. If you select the Symmetry check box, values from From start and Middle to start are copied to From end and Middle to end .

Option	Description
Middle to end From end	
Symmetry	Define whether the end and start lengths are symmetrical.
Custom properties	
More	<p>You can create user-defined attributes to add information about reinforcement. Attributes can consist of numbers, text, or lists.</p> <p>You can use the values of user-defined attributes in reports and drawings.</p> <p>To set values for user-defined attributes, click the More button.</p> <p>You can also change the name of the fields, and add new ones, by editing the <code>objects.inp</code> file. See Define and update user-defined attributes (UDAs).</p>

See also

[Create a reinforcement strand pattern \(page 473\)](#)

[Debond reinforcement strands \(page 475\)](#)

11 Disclaimer

© 2019 Trimble Solutions Corporation and its licensors. All rights reserved.

This Software Manual has been developed for use with the referenced Software. Use of the Software, and use of this Software Manual are governed by a License Agreement. Among other provisions, the License Agreement sets certain warranties for the Software and this Manual, disclaims other warranties, limits recoverable damages, defines permitted uses of the Software, and determines whether you are an authorized user of the Software. All information set forth in this manual is provided with the warranty set forth in the License Agreement. Please refer to the License Agreement for important obligations and applicable limitations and restrictions on your rights. Trimble does not guarantee that the text is free of technical inaccuracies or typographical errors. Trimble reserves the right to make changes and additions to this manual due to changes in the software or otherwise.

In addition, this Software Manual is protected by copyright law and by international treaties. Unauthorized reproduction, display, modification, or distribution of this Manual, or any portion of it, may result in severe civil and criminal penalties, and will be prosecuted to the full extent permitted by law.

Tekla, Tekla Structures, Tekla BIMsight, BIMsight, Tekla Civil, Tedds, Solve, Fastrak and Orion are either registered trademarks or trademarks of Trimble Solutions Corporation in the European Union, the United States, and/or other countries. More about Trimble Solutions trademarks: <http://www.tekla.com/tekla-trademarks>. Trimble is a registered trademark or trademark of Trimble Inc. in the European Union, in the United States and/or other countries. More about Trimble trademarks: <http://www.trimble.com/trademarks.aspx>. Other product and company names mentioned in this Manual are or may be trademarks of their respective owners. By referring to a third-party product or brand, Trimble does not intend to suggest an affiliation with or endorsement by such third party and disclaims any such affiliation or endorsement, except where otherwise expressly stated.

Portions of this software:

D-Cubed 2D DCM © 2010 Siemens Industry Software Limited. All rights reserved.

EPM toolkit © 1995-2006 Jotne EPM Technology a.s., Oslo, Norway. All rights reserved.

Open Cascade Express Mesh © 2015 OPEN CASCADE S.A.S. All rights reserved.

PolyBoolean C++ Library © 2001-2012 Complex A5 Co. Ltd. All rights reserved.

FLY SDK - CAD SDK © 2012 VisualIntegrity™. All rights reserved.

Teigha © 2002-2016 Open Design Alliance. All rights reserved.

CADhatch.com © 2017. All rights reserved.

FlexNet Publisher © 2014 Flexera Software LLC. All rights reserved.

This product contains proprietary and confidential technology, information and creative works owned by Flexera Software LLC and its licensors, if any. Any use, copying, publication, distribution, display, modification, or transmission of such technology in whole or in part in any form or by any means without the prior express written permission of Flexera Software LLC is strictly prohibited. Except where expressly provided by Flexera Software LLC in writing, possession of this technology shall not be construed to confer any license or rights under any Flexera Software LLC intellectual property rights, whether by estoppel, implication, or otherwise.

To see the third party open source software licenses, go to Tekla Structures, click **File menu** --> **Help** --> **About Tekla Structures** and then click the **3rd party licenses** option.

The elements of the software described in this Manual are protected by several patents and possibly pending patent applications in the United States and/or other countries. For more information go to page <http://www.tekla.com/tekla-patents>.

Index

- ***
- * character..... 186

- 3**
- 3D
 - in snapping.....78
 - views..... 43

- ?**
- ? character..... 186

- A**
- accuracy, see high accuracy.....583
- accuracy
 - in modeling..... 583
 - of parts..... 583
- adaptivity
 - default settings.....325
 - of individual model objects.....325
 - of reinforcement..... 512
- adding
 - buttons..... 198
 - grid lines.....26
- aligning
 - objects..... 92
- Analysis tab.....707
- anchoring hooks..... 508
- angles..... 605
- Applications & components..... 736
 - exporting custom components..... 862
 - importing custom components.....862
- applications in Applications and Components catalog 664
- arcs
 - measuring..... 605
- arithmetic operators.....784
- Array of objects (29).....143
- assemblies..... 386
 - adding objects.....390,391
 - changing the main assembly..... 393
 - changing the main part..... 392
 - comparing.....607
 - creating.....387
 - examples..... 394
 - exploding.....394
 - highlighting..... 393
 - joining.....392
 - nested assemblies.....127,390,392
 - numbering..... 630,638
 - removing objects.....393
 - selecting..... 127
 - showing invisible objects..... 589
 - sub-assemblies.....387
 - using bolts to create assemblies..... 388
 - using welds to create assemblies..... 389
- asterisk..... 186
- attaching parts..... 327
- attaching
 - reinforcement to part..... 514
- auto bolt
 - creating bolts..... 332
- AutoConnection..... 689
 - rules.....690,699
 - settings..... 690
 - using..... 693
- AutoDefaults..... 689,694
 - combining properties..... 702
 - connection check..... 704
 - iterating properties.....702
 - rules.....699
 - using..... 699
 - using reaction forces..... 706
 - using UDLs..... 706
- Auto
 - in snapping.....78
- automatic splicing tool..... 515

AutomaticSplicingTool..... 515

B

background color
 changing..... 64
 examples..... 64
backing up
 ribbons..... 198
base points..... 53
basepoints..... 53
beams
 cambering..... 331
 concrete beams..... 279
 concrete polybeams..... 282
 curved beams..... 244
 orthogonal beams..... 249
 spiral beams..... 251,285
 steel beams..... 239
 steel polybeams..... 242
 warping..... 329
bending..... 321
bending shapes
 in rebar shape manager..... 527,534
 of reinforcement..... 523,524
 rules..... 527
bent plates..... 256
bent plates..... 271
bent mesh..... 467
binding objects
 plane types..... 872
 to a plane..... 764
bolt points..... 570
bolted parts..... 332
bolts..... 332,388
 bolt group shape..... 332
 bolt spacing..... 605
 bolting sub-assemblies..... 388
 clash checking..... 623
 creating..... 332
 holes..... 343
 modifying..... 332
 studs..... 342
boundary planes..... 872
box profiles..... 884
brackets..... 186

C

c profiles..... 884
calculations..... 782
cambering..... 331
cambering parts..... 331
cast units..... 396
 adding objects..... 397
 cast unit type..... 396
 casting direction..... 399,400,401
 changing the main part..... 398
 creating..... 397
 exploding..... 399
 highlighting..... 398
 numbering..... 638
 removing objects..... 398
 selecting..... 127
 top-in-form face..... 401
cast-in-place..... 396
 continuous concrete structures..... 404
 pour breaks..... 415,417,418,420,421,422
 pour objects..... 408
 pours..... 402,403,407,409,623
 viewing parts..... 404
 viewing pour objects..... 404
categories
 in filtering..... 169
center planes..... 872
chamfering
 corner chamfers..... 368
 corner chamfer dimensions..... 368
 edge chamfers..... 368
 polybeams..... 368
chamfers
 in warping..... 329
changing
 welds to polygon welds..... 359
checking the model..... 599
circular hollow sections..... 884
circular reinforcing bar group..... 463
circular sections..... 884
Clash Check Manager..... 611
clash checking..... 610,611
 bolts..... 623
 changing priority..... 617
 changing status..... 617
 clash types..... 613
 comments..... 619

grouping clashes.....	618	components	
history.....	620	conceptual.....	688
list of clashes.....	616	converting.....	688
managing results.....	616	components in Applications & components	
previewing before printing.....	620	681
printing lists of clashes.....	620,621	components	
results.....	612	catalog.....	681
saving clashes.....	622	connections.....	674
searching.....	616	detailing.....	674
sessions.....	622	details.....	674
symbols.....	612	exploding.....	750
ungrouping clashes.....	618	nested components.....	750
class.....	594	properties.....	674
clip planes.....	608	selecting.....	127
closing		showing invisible objects.....	589
custom component editor.....	823	sub-components.....	750
cold rolled profiles.....	884	views.....	674
colliding objects.....	610	concrete beam properties.....	279
colors		concrete beams.....	279
changing color settings.....	64	concrete column properties.....	277
changing the background color.....	64	concrete cover	
color settings for object groups.....	921	of reinforcement.....	510
customizing.....	595	concrete item properties.....	298
finding RGB values.....	64	concrete items.....	298
for object groups.....	595	concrete panel properties.....	288
for parts.....	594	concrete panels.....	288
for pour objects.....	409	concrete parts.....	235
settings.....	597,598	beams.....	279
columns		cast units.....	396
concrete columns.....	277	casting direction.....	400
positioning.....	323	columns.....	277
steel columns.....	237	items.....	298
combining		pad footings.....	293
parts.....	327	panels.....	288
reinforcing bar groups.....	505	polybeams.....	282
command editor.....	198	showing as continuous.....	404
commands		slabs.....	290
customizing.....	198	strip footings.....	295
user-defined.....	198	walls.....	288
comments		concrete parts	
in clash checking.....	619	spiral beams.....	285
comparing parts or assemblies.....	607	concrete polybeam properties.....	282
comparing		concrete slab properties.....	290
ribbons.....	198	concrete walls.....	288
component analysis properties.....	734	conical bent plate.....	256
component catalog.....	681	conical bent plate.....	271
component planes.....	872,873	connection analysis properties.....	734
component symbol.....	736	connection design	

Excel.....	720	property references.....	780
connection weld properties.....	356	Radial array tool.....	141
connections		with Array of objects (29) component	143
custom connections.....	736,740	corner chamfers	
construction arcs.....	561	properties.....	368
construction circles.....	561	types and dimensions.....	368
construction lines.....	561,772	create	
construction objects.....	561	assemblies.....	234
construction polycurves.....	565	cast units.....	234
construction arcs.....	564	construction circles.....	234
construction circles.....	563	construction lines.....	234
construction lines.....	562	construction planes.....	234
construction planes.....	562	parts.....	234
copying with offset.....	566	points.....	234
modifying location.....	567	pours.....	234
construction planes.....	561,772	reinforcement.....	234
construction polycurves.....	561	creating bolts	
contextual toolbar		auto bolt.....	332
customizing.....	226	creating surface treatment	
modifying part position.....	306	unpainted area.....	383
continuous concrete structures.....	404	creating	
contour plate properties.....	253	assemblies.....	387
contour plates.....	253	beams close to each other.....	323
control numbers.....	646	bolts.....	332
assigning to parts.....	646	cast units.....	397
directions.....	647	clip planes.....	608
displaying in model.....	649	concrete beams.....	279
example.....	651	concrete columns.....	277
locking.....	651	concrete items.....	298
order.....	647	concrete panels.....	288
removing.....	650	concrete polybeams.....	282
settings.....	925	concrete slabs.....	290
unlocking.....	651	concrete walls.....	288
control points.....	53	construction arcs.....	564
converting		construction circles.....	563
welds to polygon welds.....	359	construction lines.....	562
coordinate system.....	48	construction planes.....	562
coordinates.....	21	construction polycurves.....	565
copying		contour plates.....	253
construction objects with offset.....	566	curved beams.....	244
contextual toolbar.....	109	custom components.....	747
filters.....	193	fittings.....	361
object groups.....	591	grids.....	21
objects.....	131,133	holes.....	343
parts.....	235	model templates.....	231
pour breaks.....	420	nested assemblies.....	392
properties.....	109	object groups.....	591
property pane.....	109	objects.....	96

orthogonal beams.....	249
pad footings.....	293
parts.....	96,235
pour breaks.....	418
spiral beams.....	251,285
standard-part models.....	662
steel beams.....	239
steel columns.....	237
steel items.....	298
steel polybeams.....	242
strip footings.....	295
studs.....	342
sub-assemblies.....	387
tile patterns.....	379
twin profiles.....	246
view filters	
selection filters.....	158
views.....	31
welds.....	346
curved parts.....	244,321
curved reinforcing bar group.....	461
Custom component browser.....	761,780
Custom component dialog editor.....	824
changing the language.....	857
setting options.....	857
custom component editor.....	761
closing.....	823
saving.....	823
custom component view.....	761
Custom component wizard.....	747
properties.....	864
custom components	
about custom components.....	736
adding to model.....	859
arithmetic operators.....	784
binding objects.....	764,772
construction planes and lines.....	772
copying property references.....	780
creating.....	747
default properties.....	867
dialog box files.....	827
exporting.....	862
importing.....	862
inp files.....	827
locking.....	830
modifying.....	761
modifying the dialog box.....	824
nested components.....	750

plane types.....	872
preventing modifications.....	830
protecting with passwords.....	762
seams.....	744
settings.....	863
sub-components.....	750
thumbnail images.....	753
toggle groups.....	831
types.....	736
variables.....	763
custom connections.....	740
custom details.....	742
custom hooks.....	508
custom inquiry	
modifying default attributes.....	602
custom parts.....	738
adding to model.....	860
custom seams.....	744
customized mesh.....	467
customizing	
commands.....	198
contextual toolbar.....	194
contextual toolbar.....	226
Custom Inquiry tool attributes.....	602
keyboard shortcuts.....	194
keyboard shortcuts.....	195
property pane.....	194
property pane.....	213
ribbon.....	194
ribbons.....	198
tabs.....	198
toolbars.....	194,225
cut planes.....	872
cuts	
hints and tips.....	362
in rebar sets.....	493
line cuts.....	362
part cuts.....	362
polygon cuts.....	362
cyclic dependencies	796
cylindrical bent plate.....	256
cylindrical bent plate.....	271

D

default adaptivity.....	325
default connection properties.....	708
default connection settings.....	708

defining unpainted area	
surface treatment.....	383
defining	
weld cross sections.....	360
deleting	
clip planes.....	608
objects.....	96
parts.....	96
views.....	41
dependencies	
in variable formulas.....	796
design check.....	731
design group numbering.....	653
Design tab.....	707,731
detaching parts.....	327
detail analysis properties.....	734
detailing parts	
detaching.....	327
exploding.....	327
details	
custom details.....	736,742
diagnosing model.....	624
dialog boxes	
loading properties.....	111
modifying.....	824
saving properties.....	111
dialog editor.....	824
direct modification	
modifying.....	103
display detailing.....	590
display settings.....	918
displaying	
control numbers.....	649
distance variables.....	763,764
distance	
reference distance.....	774
distances	
measuring.....	605
drawing objects	
moving.....	144
rotating.....	150

E

edge chamfers	
properties.....	368
editing	
custom components.....	761

editors	
Custom component dialog editor.....	824
end detail modifiers.....	486
end offsets.....	315
exact lines.....	583
examples	
adding option to create object in custom component.....	799
assemblies.....	394
background colors.....	64
component planes.....	873
creating custom connection.....	754
creating parametric variable.....	798
determining bolt group distance from beam flange.....	807
determining bolt size and bolt standard.....	806
determining number of bolt rows in custom component.....	809
drawing filters.....	186
exploding components.....	754
modifying custom component dialog box.....	835
modifying dialog box of stiffener detail....	835
nested components.....	757
numbering.....	651,656,658,659
rebar set modifiers in custom components.....	816
replacing sub-components in custom component.....	800
snapping in model.....	87
sub-components.....	757
tile pattern definition.....	379
using construction planes in custom component.....	803
using Excel spreadsheets with custom components.....	815
using properties file in custom component.....	802
using template attributes in custom components.....	813
using user-defined attributes in custom components.....	811
view and selection filters.....	186
excel design.....	707
Excel	
connection design.....	720

- using with custom components..... 815
- exploding
 - assemblies..... 394
 - cast units..... 399
 - components..... 750
 - parts..... 327
- exporting
 - custom components..... 862
 - keyboard shortcuts..... 195
- extensions in Applications and components catalog..... 664
- extensions
 - importing..... 670

F

- family numbers..... 634,635
 - example..... 656
 - modifying..... 636
- filtering
 - And/Or..... 166
 - categories..... 169
 - conditions..... 166
 - copying filters..... 193
 - object properties..... 169
 - objects..... 155
 - parentheses..... 166
 - removing filters..... 193
 - template attributes..... 166,185
 - use for..... 155
 - view filters
 - selection filters..... 158
 - wildcards..... 186
- filters
 - examples..... 186
 - selection..... 122
- finding distant objects..... 626
- finding
 - clashes..... 611
- fit part end..... 361
- fittings..... 361
- fixed profiles..... 316
- flying through model..... 609
- folded plates..... 884
- footings..... 293,295
- formulas
 - arithmetic operators..... 784
 - in custom components..... 782

- in rebar shape manager..... 533
- functions..... 782

G

- General tab..... 707,730
- geometry
 - of reinforcement..... 514
- global coordinate system..... 48
- global point of origin..... 48
- grid lines
 - adding..... 26
 - deleting..... 29
 - modifying..... 26
 - properties..... 26
- grid planes..... 872
- grid views
 - properties..... 45
- grids
 - coordinates..... 18,21
 - creating..... 18,21
 - deleting..... 18,21
 - labels..... 18,21
 - line extensions..... 18
 - modifying..... 18,21
 - origin..... 18
 - properties..... 18,21
 - radial..... 18,21
 - rectangular..... 18,21
- grid
 - work plane grid..... 50
- grouping
 - clashes..... 618
 - reinforcement..... 504

H

- handles..... 303
 - in pour breaks..... 421
 - of reinforcement..... 506
- hat profiles..... 884
- hatching
 - DX..... 584
- hiding
 - cut lines..... 362
 - dimensions..... 116
 - part labels..... 320

rebar set modifiers.....	486
reference lines.....	303
selected parts.....	587
top-in-form face.....	401
unselected parts.....	588
work area.....	46
high accuracy.....	583
highlighting	
assemblies.....	393
cast units.....	398
objects.....	129
history	
of clashes.....	620
holes.....	343
horizontal position.....	313

I	
i beams (concrete).....	884
i beams (steel).....	884
i profiles.....	884
identical	
areas.....	324
parts.....	632
reinforcement.....	632
if you cannot select objects.....	129
import items.....	298
importing	
custom components.....	862
keyboard shortcuts.....	195
points.....	570
inp files	
in custom components.....	824,827
inquiring	
object properties.....	599
interrupting	
object selection.....	129
irregular beams (concrete).....	884
items.....	235,298
changing the shape.....	298
limitations.....	298

J	
joining assemblies.....	392
joints. def.....	708
joints.def.....	707

K	
keyboard shortcuts.....	195

L	
l profiles.....	884
label color	
changing.....	64
labels	
part labels.....	320
ledger beams (concrete).....	884
leg faces.....	480
levels.....	323
line cuts.....	362
linear array tool.....	138
lines	
exact.....	583
linking	
parametric variables to object properties	
.....	777
loading	
saved properties.....	111
local coordinate system.....	48
locking and unlocking	
custom components.....	830
log files	
pour errors.....	623
solid errors.....	623

M	
macros	
adding.....	666
editing.....	666
global.....	664,666
local.....	664,666
macro folder.....	664
recording.....	666
running.....	666
magnetic construction planes and lines..	772
measuring objects.....	605
angles.....	605
arcs.....	605
bolt spacing.....	605
distances.....	605
mesh	

bent.....	467
customized.....	467
handles.....	506
modifying.....	496
polygonal.....	467
rectangular.....	467
ungrouping.....	503
model templates	
creating.....	231
modifying	
options.....	231
model objects.....	95
showing and hiding.....	581
model views.....	29,31
rendering options.....	584
representation options.....	584
modeling direction.....	322
Modeling of floor bay (66).....	329
modeling	
identical areas.....	324
more accurately.....	583
models	
checking for errors.....	599
flying through model.....	609
moving.....	75
numbering.....	627
rotating.....	75
zooming.....	75
modifiers	
in custom components.....	816
showing or hiding.....	486
visibility.....	486
modifying	
parts.....	235
modifying	
concrete beams.....	279
concrete columns.....	277
concrete items.....	298
concrete panels.....	288
concrete polybeams.....	282
concrete slabs.....	290
concrete walls.....	288
construction objects.....	567
contour plates.....	253
curved beams.....	244
custom components.....	747,761
items.....	298
model templates.....	231

objects.....	103
orthogonal beams.....	249
pad footings.....	293
parts.....	324
pour breaks.....	421
pour object properties.....	410
rebar sets.....	478
reinforcement.....	478,496
steel beams.....	239
steel columns.....	237
steel polybeams.....	242
strip footings.....	295
twin profiles.....	246
moving	
buttons.....	198
drawing objects.....	144
models in a view.....	75
objects.....	103,131,133,144
pour breaks.....	420
multi-story structures.....	323

N

nested assemblies.....	127,390,392
nested	
components.....	750,757
no paint area.....	383
numbering.....	627,637
about numbering.....	627
assemblies.....	638
cast units.....	638
checking and repairing.....	645
clearing.....	641
control numbers.....	646,647,649,650,651
examples.....	656,658,659
family numbers.....	634,635,636,656
history.....	644
identical parts.....	632
manually.....	640
modifying.....	640
numbering series.....	630
parts.....	637,645
preliminary numbers.....	640
reinforcement.....	632,639
renumbering.....	645
series.....	628,629,631
settings.....	636,661,923,925
standard-part model.....	662

user-defined attributes.....	634
welds.....	640
what affects.....	633
numbering	
design group numbering.....	653
numbering settings.....	923
numeric snapping.....	87

O

object groups.....	591
changing the color.....	595
color settings.....	921
copying to another model.....	591
creating.....	591
deleting.....	591
transparency settings.....	922
object properties	
in filtering.....	169
object property report templates.....	601
object representation settings....	593,597,598
objects	
clash checking.....	610
copying.....	131,133
creating.....	96
defining visibility.....	582
deleting.....	96
filtering.....	155
finding distant objects.....	626
inquiring properties.....	599
measuring.....	605
moving.....	131,133,144
numbering.....	637
placing objects in model.....	561
rotating.....	150
selecting.....	116
showing and hiding.....	48
offsets.....	315
operators.....	782
origin.....	48
orthogonal beam properties.....	249
orthogonal beams	
positioning.....	323
orthogonal	
snapping.....	82
others.....	884
outline planes.....	872
overlapping	

numbering series.....	631
oversized holes.....	343

P

pad footing properties.....	293
pad footings.....	293
positioning.....	323
properties.....	293
page margins.....	621
page orientation.....	621
paintbrush icon.....	109
panels.....	288,884
panning.....	75
paper size.....	621
parametric profiles.....	316
available in Tekla Structures.....	884
predefined.....	884
parametric variables.....	763
creating.....	777
linking.....	777
part add, see attaching parts.....	327
part cuts.....	362
part labels	
showing and hiding.....	320
part position.....	303
on the work plane.....	308
part properties.....	96
part reference lines.....	303
part view angle	591
part	
handles.....	303
position.....	303
reference lines.....	303
parts	
adding to assembly.....	391
assemblies.....	386
bending.....	321
cambering.....	331
changing the color.....	594
changing the material.....	316
changing the profile.....	316
combining.....	327
comparing.....	607
concrete parts.....	235
copying.....	235
creating.....	235
curved parts.....	321

custom parts.....	736,738	at the intersection of two part axes....	570
cutting with another part.....	362	importing.....	570
handles.....	303	on line.....	570
hiding.....	587	on plane.....	570
horizontal parts.....	322	parallel to two points.....	570
identical parts.....	632	projected points on line.....	570
items.....	298	properties.....	570
labels.....	320	tangent to circle.....	570
location.....	306	polybeams.....	242,282
modifying.....	235,324	chamfers.....	368
modifying part shape.....	103	polygon cuts.....	362
numbering.....	630,637,645	polygon mesh.....	467
position.....	303,306	polygon welds.....	346
position settings.....	923	converting.....	359
properties.....	235	splitting double-sided to single-sided	360
showing and hiding.....	581	position	
showing only selected parts.....	588	depth.....	310
showing with exact lines.....	583	end offsets.....	315
showing with high accuracy.....	583	horizontal.....	313
splitting.....	326	part position settings.....	923
steel parts.....	235	rotation.....	309
user-defined attributes.....	319	vertical.....	312
passwords		pour breaks.....	415
protecting custom components.....	762	adaptivity.....	417
performance		copying.....	420
modeling tips.....	229	creating.....	418
phases		handles.....	421
in numbering.....	659	modifying.....	421
pour phase.....	407	moving.....	420
plane of curvature.....	321	removing.....	422
plane types.....	872	selecting.....	420
plane views.....	43	visibility.....	417
Plane		pour objects.....	408
in snapping.....	78	changing color and transparency.....	409
plates		modifying properties.....	410
bent plates.....	256	reinforcing.....	459
bent plates.....	271	user-defined attributes.....	410
points.....	561,570	viewing.....	404
along arc using center and arc points	570	pour units.....	410
along extension of two points.....	570	adding objects automatically.....	414
at any position.....	570	pours	
at the intersection of a circle and a line....	570	changing color and transparency.....	409
at the intersection of a part and a line....	570	errors.....	422,623
at the intersection of a plane and a line	570	example workflow.....	425
.....	570	introduction.....	402
at the intersection of two lines.....	570	pour breaks.....	415
		pour objects.....	408
		pour phase.....	407

pour representation.....	404
reinforcing.....	459
switching on/off.....	402,403
viewing.....	404
working with.....	425
preliminary numbers.....	640
preparing parts for welding.....	357
prestressed strands	
debonding.....	475
previewing	
lists of clashes.....	620
printing	
lists of clashes.....	620
priority	
in clash checking.....	617
profiles	
fixed.....	316
parametric.....	316,884
predefined.....	884
profile names.....	186
standardized values.....	316
twin profiles.....	246
properties	
copying.....	109
Custom Component Wizard.....	864
custom components.....	867
grid lines.....	26
grids.....	21
twin profile.....	246
variable properties.....	875
property files.....	111
property modifiers.....	486
property pane.....	96
customizing.....	213
loading properties.....	111
saving properties.....	111
publishing groups in Applications & components catalog.....	671

Q

question mark.....	186
quick access toolbar	
undo history.....	114

R

radial array tool.....	141
radial grid.....	21
radius.....	321
rebar classifier.....	517
rebar groups.....	452
circular.....	463
curved.....	461
spiral.....	465
tapered.....	465
rebar sequence numbering.....	517
rebar set modifiers.....	486
rebar sets.....	428
cutting.....	493
in curved structures.....	445
leg faces.....	480
modifiers.....	486
modifiers in custom components.....	816
modifying.....	478,486
rebar shape placing tool.....	438
showing or hiding modifiers.....	486
spacing.....	494
rebar shape manager.....	523
bending shapes.....	524,534
formulas.....	533
rules.....	527,534
rebar shape placing tool.....	438
RebarClassifier.....	517
rebars	
adaptivity.....	512
bar leg length.....	521
bending types.....	535
combining.....	505
concrete cover.....	510
creating.....	428,451
geometry.....	514
grouping.....	504
handles.....	506
hooks.....	508
length.....	518
modifying.....	478,496
shape catalog.....	454,456,457
splitting.....	506
ungrouping.....	503
RebarSeqNumbering.....	517
recording	
macros.....	666

rectangular grid.....	21	splitting.....	506
rectangular hollow sections.....	884	strands.....	473
rectangular mesh.....	467	tapered bar group.....	465
rectangular sections.....	884	ungrouping.....	503
redrawing views.....	43	reinforcing bar groups.....	452
reference distance variables.....	763,774	adaptivity.....	512
reference lines.....	303	bar leg length.....	521
reference models		bar length.....	518
clash checking.....	610	circular.....	463
refreshing views.....	43	combining.....	505
reinforcement		curved.....	461
adaptivity.....	512	excluding bars.....	502
attaching to part.....	514	geometry.....	514
bar groups.....	452	grouping.....	504
bar leg length.....	521	handles.....	506
bar length.....	518	modifying.....	496
bending shapes.....	524	omitting bars.....	502
bending types.....	535	pours.....	459
circular bar groups.....	463	shape catalog.....	454,456,457
classifying.....	517	spiral.....	465
combining.....	505	splitting.....	506
concrete cover.....	510	tapered.....	465
curved bar groups.....	461	ungrouping.....	503
customized mesh.....	467	reinforcing bars.....	451
debonding strands.....	475	bending types.....	535
for pour objects.....	459	combining.....	505
geometry.....	514	geometry.....	514
grouping.....	504	handles.....	506
handles.....	506	hooks.....	508
hooks.....	508	leg length.....	521
identical.....	632	length.....	518
in templates.....	559	modifying.....	496
layer information.....	517	pours.....	459
mesh.....	467	shape catalog.....	454,456,457
modifying.....	477,478,496	spacing.....	500
numbering.....	632,639	reinforcing pour objects.....	459
omitting bars.....	502	removing	
prestressed strands.....	473	filters.....	193
rebar sets.....	428	pour breaks.....	422
running numbers.....	517	weld cross sections.....	360
sequential numbers.....	517	rendering	
shape catalog.....	454,456,457	DirectX.....	67
shape codes.....	524	OpenGL.....	67
shape recognition.....	523	rendering	
single bars.....	451	of parts and components.....	584
spiral bar group.....	465	renumbering.....	645
splice.....	476	repairing model.....	624
splicing.....	515	repairing	

numbering errors.....	645
report templates	
for inquiring object properties.....	601
reports	
on reinforcement.....	533,559
representation	
of cast-in-place concrete structures...	404
of parts and components.....	584
reshaping	
objects.....	103
resizing	
buttons.....	198
RGB values.....	64
ribbon editor.....	198,213
ribbons	
backing up.....	198
customizing.....	198
restoring.....	198
right-click	
selecting.....	129
right-hand rule.....	50
rollover highlight.....	129
rotating	
drawing objects.....	150
objects.....	150
parts.....	309
settings.....	150
rotation angle knob.....	306
rotation settings.....	150
round	
holes.....	343
plates.....	253
slabs.....	290
rules	
in bending shapes.....	527
in rebar shape manager.....	527,534
running	
macros.....	666

S

saving	
clashes.....	622
custom component.....	823
dialog box properties.....	111
property pane properties.....	111
views.....	41
seams	

custom seams.....	736,744
searching	
clashes.....	616
secondary guidelines.....	486
select work plane.....	63
selecting toolbar.....	122,225
selecting	
all objects.....	116
assemblies.....	127
by id.....	116
cast units.....	127
dates from model.....	194
handles.....	116
interrupting object selection.....	129
multiple objects.....	116
nested assemblies.....	127
objects.....	116,122,129
objects in components.....	127
on right-click.....	129
pour breaks.....	420
previous objects.....	116
unable to select objects.....	129
values from model.....	194
selection dial.....	306
selection filters.....	122,158
selection switches.....	122
setting up	
grid.....	17
views.....	17
work area.....	17
work plane.....	17
workspace.....	17
settings	
adaptivity.....	325
bolt properties.....	332
color settings.....	597,598,921
concrete beam properties.....	279
concrete column properties.....	277
concrete item properties.....	298
concrete panel properties.....	288
concrete slab properties.....	290
contour plate properties.....	253
corner chamfer properties.....	368
custom components.....	863
dialog editor settings.....	857
display settings.....	918
edge chamfer properties.....	368
grid line properties.....	26

grid properties.....	21	views.....	41
grid view properties.....	45	welds.....	358
model templates.....	231	work area.....	46
numbering.....	661	single bolts.....	332
numbering settings.....	636,923,925	single-part welds.....	346
orthogonal beam properties.....	249	slabs.....	290
pad footing properties.....	293	warping.....	329
part position settings.....	923	slotted holes.....	343
point properties.....	570	snap grid.....	92
reinforcement properties.....	927	snap override toolbar.....	225
rotation settings.....	150	snap settings.....	94
snapping.....	94	snap switches.....	77,79
steel beam properties.....	242	snap symbols.....	79
steel beam properties.....	239,244	snapping.....	77,82,84
steel column properties.....	237	example.....	87
steel item properties.....	298	extension line.....	84
strip footing properties.....	295	line.....	84
transparency settings.....	597,598,922	lock coordinates.....	92
twin profile properties.....	246	orthogonal directions.....	82
units and decimals.....	18	settings.....	94
view properties.....	44	snap depth.....	78
view settings.....	918	snap zone.....	78
weld properties.....	346	snap priority.....	78
shape codes		snap switches.....	79
of reinforcement.....	523,524,527	snap symbols.....	79
shape		temporary reference lines.....	92
modifying part shape.....	103	to middle points.....	303
shapes		to numeric coordinates.....	87
of items.....	298	using coordinates.....	87
shifting the work plane.....	51	snapping mode	
shortcuts, see keyboard shortcuts.....	195	absolute.....	87
showing and hiding		global.....	87
model objects.....	581	relative.....	87
parts.....	581	snapping toolbar.....	77,78,225
rebar set modifiers.....	486	spacing in rebar sets.....	494
showing part reference lines.....	303	spacing reinforcing bars.....	500
showing		special characters.....	186
cast-in-place concrete structures.....	404	spiral beams.....	251,285
cut lines.....	362	splice.....	476
dimensions.....	116	splicing	
invisible assembly objects.....	589	reinforcement.....	515
invisible component objects.....	589	splitters.....	486
part labels.....	320	splitting	
parts with exact lines.....	583	parts.....	326
parts with high accuracy.....	583	reinforcing bar group.....	506
rebar set modifiers.....	486	stand-alone bent plate.....	256
reference lines.....	303	stand-alone bent plate.....	271
top-in-form face.....	401	standard-part models.....	662

standardized values for parametric profiles	316
status	
in clash checking	617
steel beam properties	239
steel column properties	237
steel connection properties	356,730,731
steel curved beam properties	244
steel detail properties	730
steel item properties	298
steel items	298
steel parts	235
assemblies	386
beams	239
bent plates	256
bent plates	271
columns	237
contour plates	253
curved beams	244
items	298
orthogonal beams	249
polybeams	242
spiral beams	251
twin profiles	246
steel polybeam properties	242
stories	
creating multi-story structures	323
strand pattern	
of reinforcement	473
strip footing properties	295
strip footings	295
studs	342
sub-assemblies	387
bolting to existing assembly	388
welding to existing assembly	389
sub-components	750
examples	757
surface treatment	
add new subtype	378
adding	373
modifying	373
on all faces of part	373
on chamfered parts	373
on cut faces	373
on part face	373
on parts with openings and recesses	373
on selected areas	373
tiled surface treatment	373

surfaces	385
Swap Handles	303
switching between	
views	43
switching on/off	
pour management	402,403
system components	674

T

t profiles	884
t profiles (concrete)	884
tabs	198
tail information	346
template attributes	
in filtering	185
templates	
model templates	231
thumbnail image	
of custom component	753
tiled surface treatment	
creating new tile patterns	379
pattern definition example	379
pattern definitions	379
pattern elements	379
tips	
creating beams close to each other	323
creating curved parts	321
creating custom components	881
creating horizontal parts	322
custom components in new Tekla Structures version	882
finding RGB values for colors	64
hiding cut lines	362
modeling identical areas	324
modeling large models	229
numbering settings	661
positioning columns, pad footings and orthogonal beams	323
right-hand rule	50
sharing custom components	882
toggle_group	831
toolbar	
selection switches	122
toolbars	
customizing	225
snapping toolbar	78
top-in-form face	401

tracking	
along line.....	87
transparency settings.....	922
transparency	
of pour objects.....	409
settings.....	597,598
twin profiles.....	246

U

u profiles.....	884
UDL.....	731
undo history.....	114
ungrouping	
clashes.....	618
reinforcement.....	503
uniform distributed load.....	731
units and decimals.....	18
up direction.....	730
updating views.....	43
user-defined attributes	
for parts.....	319
in numbering.....	634
user-defined weld cross sections.....	360
user-defined	
commands.....	198

V

value types.....	875
values	
selecting from model.....	194
variable cross sections.....	884
variable formulas.....	782
variables	
creating dependencies.....	777
dependencies.....	796
distance variables.....	764
in custom components.....	763
parametric variables.....	777
value types.....	875
variable properties.....	875
vertical position.....	312
view depth.....	48
view filters.....	158
view plane.....	50
view planes	

moving.....	31
view settings.....	918
viewing	
cast-in-place concrete parts.....	404
models.....	608
numbering history.....	644
pour objects.....	404
views.....	29
creating.....	31
deleting.....	41
modifying.....	41
naming.....	41
opening.....	41
properties.....	44
refreshing.....	43
saving.....	41
switching between views.....	43
visibility of objects.....	48,582
visibility	
of parts.....	581
of pour breaks.....	417

W

walls.....	288
warping.....	329
beams.....	329
concrete slabs.....	329
weld cross sections	
defining.....	360
removing.....	360
weld preparation.....	357
welded beam profiles.....	884
welded box profiles.....	884
Welds in components.....	707
welds.....	356
between parts.....	346
creating.....	346
defining cross sections.....	360
numbering.....	640,925
polygon welds.....	346
properties.....	346
removing cross sections.....	360
showing.....	358
single-part welds.....	346
user-defined cross sections.....	360
visibility in model.....	358
weld preparation.....	357

weld types.....	346
welding sub-assemblies.....	389
wildcards.....	186
work area.....	46,48
hiding.....	46
work plane handler toolbar	
base points.....	63
project base points.....	63
work plane.....	63
work plane	
shifting.....	51
show or hide.....	50
toolbar.....	63
wq profiles.....	884

Z

z profiles.....	884
zooming in or out.....	75