Design Module

User's Guide

6.4

COMSOL

Design Module User’s Guide

© 2005–2025 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement. Portions of this software are owned by Siemens Industry Software, Inc. © 1986–2025. All Rights Reserved. Portions of this software are owned by Spatial Corp. © 1989–2025. All Rights Reserved.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. ACIS and SAT are registered trademarks of Spatial Corporation. CATIA is a registered trademark of Dassault Systèmes or its subsidiaries in the US and/or other countries. Parasolid is a trademark or registered trademark of Siemens Industry Software, Inc. or its subsidiaries in the United States and in other countries. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those or the above non-COMSOL trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 6.4

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located on the COMSOL Access page at www.comsol.com/support/case. Useful links:

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: www.comsol.com/support/learning-center

Part No. CM024001

Contents

Chapter 1: Introduction

Chapter 2: Geometry Tools and Features

1

Introduction

Welcome to the Design Module User's Guide. This guide details the functionality of this optional package that extends the COMSOL Multiphysics® modeling environment with additional tools and features to create and modify geometry, and to import and export geometry using the most common 3D CAD file formats.

This introductory chapter contains an overview of the capabilities of the module, including a summary of the included geometry features, an overview of this guide, and a description of where to find documentation and model examples.

About the Design Module

Overview of the Included Geometry Tools and Features

The Design Module extends the geometry modeling capabilities of COMSOL Multiphysics with constraint and dimension features in 2D, a dedicated geometric kernel, the *CAD kernel*, features for creating and modifying geometry, import, and export of several 3D CAD formats, and functionality to repair and defeature imported geometry objects. Included geometry features are, for example, the fillet and chamfer features in 3D, and the loft feature that can generate 3D surfaces based on cross-sectional profiles, which could come from an MRI scan, or could be the faces of existing geometry objects. Further functionality such as the midsurface and thicken allows for converting a thin solid object into a surface, or the other way around.

The import capabilities cover the most common 3D CAD file formats: *ACIS*®, *AutoCAD*®, *IGES*, *Inventor*®, *NX*®, *Parasolid*®, *PTC Creo Parametric*™, *PTC Pro/ENGINEER*®, *SOLIDWORKS*®, and *STEP*. In addition, support for *CATIA*® *V5* is available as a separate add-on. To exchange data with CAD packages, you can export your geometry to the *ACIS*®, *IGES*, *Parasolid*®, and *STEP* file formats.

Finally, the product provides a wide range of tools for you to prepare an imported 3D design for meshing and analysis. You can interactively search for and remove geometric features, for example, fillets, holes, slivers, small faces, and short edges. You can also modify objects by detaching a portion to form an additional computational domain, or by creating a fluid domain for computation, in case the CAD design only includes the solid parts.

GEOMETRY FEATURE	ICON	DESCRIPTION
2D Geometry Features		
Angle		Constrain the angle between two edges
Coincident		Constrain two geometric entities to coincide with each other
Concentric		Constrain circular edges and vertices to have the same center
Directed Distance		Constrain the distance between two geometric entities in a given direction

GEOMETRY FEATURE	ICON	DESCRIPTION
Distance		Constrain the distance between two geometric entities
Equal Distance		Constrain the distances between two pairs of geometric entities to be equal
Equal Radius		Constrain two circular edges to have the same radius
Horizontal		Constrain a straight edge to be parallel to the x-axis
Parallel		Constrain straight edges to be parallel
Perpendicular		Constrain two straight edges to be perpendicular
Position		Constrain the x- and y-coordinates of a point
Projection		Project 3D objects and entities to a work plane
Radius		Constrain the radius of a circular edge
Tangent Constraint		Constrain two edges to be tangent
Total Edge Length		Constrain the total length for a set of edges
Vertical		Constrain a straight edge to be parallel to the y-axis
x-Distance		Constrain the distance in the x direction between entities
y-Distance		Constrain the distance in the y direction between entities
3D Geometry Features		
Cap Faces		Generate faces from edges to fill gaps and create solid objects, or to partition solids
Check		Check CAD objects for faults, for example tolerance issues and invalid entities
Chamfer		Create a bevel on selected edges
Convert to COMSOL		Convert to the COMSOL kernel representation
Replace Faces		Delete and replace faces
Delete Fillets		Search for and delete fillets
Delete Holes		Search for and delete holes
Delete Short Edges		Search for and delete short edges
Delete Sliver Faces		Search for and delete sliver faces
Delete Small Faces		Search for and collapse small faces

GEOMETRY FEATURE	ICON	DESCRIPTION
Delete Spikes		Search for and delete spikes from faces
Detach Faces		Detach faces to form a new object from them
Detect Interferences		Search for interferences, such as intersections, gaps, touches, and containments, between objects
Export		Export geometry objects to 3D CAD file formats
Fillet		Create rounds on selected edges
Import		Import geometry objects from 3D CAD file formats
Knit to Solid		Knit surface objects to form solid or surface object
Loft		Create a lofted surface from a set of profile curves
Midsurface		Generate midsurfaces for selected solid objects
Offset Faces		Offset faces of 3D objects in the normal direction
Project to Faces		Project edges to faces in 3D
Repair		Repair defects and remove small details from 3D objects
Replace Faces		Replace faces by growing surrounding faces or creating new faces
Thicken		Create a solid by offsetting selected surfaces
Transform Faces		Displace, rotate, and scale faces of 3D objects

Overview of the User's Guide

This documentation covers the Design Module and the add-on for file import of CATIA® V5 files. Instructions on how to use the geometry modeling tools in COMSOL Multiphysics® in general are included with the *COMSOL Multiphysics Reference Manual*. To help you get started with modeling this module is also accompanied by the quick-start guide *Introduction to Design Module*.

Where Do I Access the Documentation and Application Libraries?

A number of internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

> If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to other modules (as long as you have a license), application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes all core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to many of the features on the GUI. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context*) *help*.

Win

To open the **Help** window:

- In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1.
- On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1.
- From the **File** menu, click **Help** (?).
- In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac Linux	To open the **Help** window: • In the **Model Builder** or **Physics Builder** click a node or window and then press F1. • In the main toolbar, click the **Help** ( ) button. • From the main menu, select **Help>Help**.

Opening the Documentation Window

Win	To open the **Documentation** window: • Press Ctrl+F1. • From the **File** menu select **Help>Documentation** ( ).

Mac Linux	To open the **Documentation** window: • Press Ctrl+F1. • In the main toolbar, click the **Documentation** ( ) button. • From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each application includes documentation with the theoretical background and step-by-step instructions to create a model application. The applications are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual applications as a template for your own modeling and applications. In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the application and its properties, including options to open it or a PDF document.

	The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window

To open the **Application Libraries** window ( ):

Win	• In the **Home** toolbar, click **Windows** and select **Application Libraries**. When the toolbar is compressed, you sometimes find it under **Layout>Windows**. • From the **File** menu select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu select ( ) **Update COMSOL Application Libraries**.

Mac Linux	From the **File** or **Windows** menu select **Application Libraries**. To include the latest versions of model examples, from the **Help** menu select ( ) **Update COMSOL Application Libraries**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL WEBSITES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Video Gallery	www.comsol.com/videos
Support Knowledge Base	www.comsol.com/support/knowledgebase

2

Geometry Tools and Features

This chapter describes the tools and features available for creating, importing, and modifying geometry with the Design Module.

In this chapter:

Constraint and Dimension Features

In this section:

Working with Constraints and Dimensions

With the Design Module you can apply constraints and dimensions to geometry in 2D geometry sequences, including in geometric parts and on work planes in 3D. By using drawing tools you can quickly draw geometry that resembles a desired shape, then, by adding constraints and dimensions, you can obtain the final geometry.

A dimension (or constraining dimension) is a requirement on geometric entities that has a value, for example the distance between vertex 3 and edge 7 should be `5[m]`. You may also define a dimension using an expression that may include global parameters, for example `A*5[m]` that depends on the parameter `A`. A constraint is a requirement on geometric entities that does not have a value; for example, a requirement that edges 9 and 11 should be perpendicular.

When you apply a constraint or dimension feature, the software immediately adjusts the drawing to satisfy the applied feature. It does this by adjusting the values of the input fields of the geometric features that created the objects. The constraints and dimensions are visualized with symbols and arrows in the **Graphics** window.

ENABLING THE CONSTRAINTS AND DIMENSIONS FUNCTIONALITY

The constraint and dimension functionality is automatically enabled when you add a constraint or dimension feature to a 2D geometry sequence using the buttons in the **Constraints** and **Dimensions** sections in the **Sketch** toolbar, and from the context menus in the **Graphics** and **Model Builder** windows. To manually enable the use of constraints and dimensions in a 2D geometry or part, or work plane, go to the Settings window for the **Geometry** or **Plane Geometry** node and set **Use constraints and dimensions** to **On**. In addition to enabling the use of constraint and dimension features, this checkbox also enables the editing of objects that result from previous geometric operations by dragging of vertices and edges in the Graphics window. Note that turning this functionality on for a geometry that consists of a very large number of edges and vertices may slow down the sketch visualization, and it is therefore not recommended.

To enable constraints and dimensions by default in new models make sure that the **Use in new geometries** checkbox is selected on the **Geometry** > **2D Constraints and Dimensions** page in the **Preferences** window.

It is not possible to add Measurement Parameters to a 2D geometry sequence when the use of constraints and dimensions is enabled. To create parameters that measure geometric entities, you can instead change constraining dimension features to Measuring Dimensions and create Dimension Parameters.

When you open an existing model, the constraint and dimension features in the model are normally active. To change this behavior clear the **Use when opening a model that uses constraints and dimensions** checkbox on the **Geometry** > **2D Constraints and Dimensions** page in the **Preferences** window. After this, when you open a model that uses constraints and dimensions you will get a question whether you want to disable the use of constraints and dimensions. If you answer yes, the constraint and dimension features will be loaded, but they will not have any effect.

If you want to avoid that dimension and constraint features modify the input fields of a geometry feature click the **Constrain** ( ) button (visible only when the use of constraints and dimensions is enabled) to the right of the text field in the Settings window of the feature. This locks the text field (the **Constrain** button icon changes into to accept only values or expressions that are entered directly, so that the text field now becomes a *built-in dimension*. In some cases this is also indicated in the Graphics window by the built-in dimensions being visualized with arrows.

Constraints and dimensions can be applied to just about any geometry object, for example to the output of features you have added from the Model Builder, and the output of features such as the Union operation. The following features create objects that cannot be modified using constraints or dimensions:

- Polygon where the data source is not Table
- Interpolation Curve where the data source is not Table or Relative tolerance is not 0
- Rectangle, Square, Circle, Ellipse using layers
- Parametric Curve
- Cross Section
- Edit Object
- Import
- Offset
- Part Instance
- Partition Objects, Partition Domains, and Partition Edges
- Projection
- Tangent
- Thicken

Note that even when an object cannot be modified by a constraint or dimension feature, its entities can be used as input for such a feature together with entities that can be modified. When you use constraints and dimensions together with the listed features, or with the Box Selection or Disk Selection features, you may need to build the geometry several times to satisfy all the constraints and dimensions.

When applying a constraint or dimension to a straight or circular edge, the entire line or full circle is usually considered. For example, a dimension that constrains the distance between a vertex and a straight edge really constrains the distance between the vertex and the straight line underlying the edge.

CREATING CONSTRAINT AND DIMENSION FEATURES

When you are creating a drawing in sketch mode — that is, the **Sketch** button is active; see also The Sketch Visualization — activate the **Snap to Geometry** ( ) and **Constrain** ( ) buttons in the **Draw Settings** group of the **Sketch** toolbar to automatically generate constraints as you draw curve segments using the geometry primitives. For example, a Coincident constraint is automatically applied when a vertex is snapped to an existing vertex, and Perpendicular and Tangent constraints may be applied where appropriate between curve segments as you draw them.

When applying constraint and dimension to a geometry it is recommended to add the constraints before the dimensions. The applied constraints and dimensions appear as feature nodes in the geometry sequence and the corresponding symbols are displayed in the Graphics window. You can create constraint and dimension features by using the toolbar buttons from the **Sketch** toolbar or the contextual menu in the Graphics window, as described below in the sections Creating Constraints and Creating Dimensions, or by using the contextual menu of a **Geometry** or **Plane Geometry** node in the Model Builder. By the latter method you first add a constraint or dimension feature to the geometry sequence, then you select the input entities in its Settings window. When assigning the selections in the Settings window, note that the numbering of the vertices and edges in the sketch visualization (see The Sketch Visualization) sometimes differs from the numbering used in the nonsketch visualization. After building the feature the symbol for the constraint will appear the Graphics window.

CREATING CONSTRAINTS

You can add a constraint using the buttons from the **Sketch** toolbar by one of the following methods:

- Click the **Constraint** ( ) button in the **Sketch** toolbar, also available from the right-click context menu in the Graphics window, to enter the *smart constraint* mode. In this mode, you can start with selecting edges and vertices in the Graphics window. When you have selected a sufficient number of entities, a constraint symbol will appear next to the mouse pointer. This symbol indicates a suggested constraint for the selected entities. If you are satisfied with this suggestion, move the symbol to the desired position on the canvas and left-click to place it there. This will also add the constraint feature node to the geometry sequence, and the geometry will be rebuilt so that you see the effect of the constraint. You can drag the symbol afterward to adjust its position. To select a different type of constraint from the one suggested, click the button of another constraint, from the **Sketch** toolbar or from the context menu displayed after right-clicking in the Graphics window, before placing the symbol. If a sufficient number of entities are selected for this new constraint, the symbol for the constraint will appear immediately at a default position, and the constraint feature node will be added to the geometry sequence. The smart constraint mode remains active after each applied constraint until you click the **Constraint** button again, or press **Esc**.

- Click a button (other than **Constraint**) from the **Constraint** group of the **Sketch** toolbar, to select a specific constraint. If you have already selected some entities you can also select an applicable constraint from the context menu displayed when you right-click in the Graphics window. If the preselected entities are sufficient for applying the constraint, the symbol for the constraint will appear immediately at a default position, and the constraint feature node will be added to the geometry sequence. Otherwise, when you have selected a sufficient number of entities, the constraint symbol will appear under the pointer. Move the symbol to the desired position on the canvas and click to place it there. The feature node for the constraint will be added to the geometry sequence, and the geometry will be rebuilt.

CREATING DIMENSIONS

You can add a dimension using the buttons from the **Sketch** toolbar by one of the following methods:

- Click the **Dimension** ( ) button in the **Sketch** toolbar to enter the *smart dimension* mode. In this mode, you can start with selecting edges and vertices in the Graphics window. When you have selected a sufficient number of entities, a dimension symbol will appear next to the mouse pointer. This symbol indicates a suggested dimension for the selected entities (also considering the mouse pointer position in relation to the selected entities). If you are satisfied with this suggestion, move the symbol to the desired position on the canvas and click to place it there. This will also add the dimension feature node to the geometry sequence. Change the dimension's value in the Settings window for the feature; then click **Build Selected** to see its effect. To select a different type of dimension from the one suggested, click the button of another dimension, from the **Sketch** toolbar or from the context menu displayed after right-clicking in the Graphics window, before placing the symbol. If a sufficient number of entities are selected for this new dimension, the symbol for the dimension will appear immediately at a default position, and the dimension feature node will be added to the geometry sequence. You can drag the symbol afterward to adjust its position. The smart dimension mode remains active after each applied dimension until you click the **Dimension** button again, or press **Esc**.

- Click a button (other than **Dimension**) from the **Dimensions** group of the **Sketch** toolbar, to select a specific dimension. If you have already selected some entities you can also select an applicable dimension from the context menu displayed when you right-click in the Graphics window. If the preselected entities are sufficient for applying the dimension, the symbol for the dimension will appear immediately at a default position, and the dimension feature node will be added to the geometry sequence. Otherwise, when you have selected a sufficient number of entities, the dimension symbol will appear under the pointer. Move the symbol to the desired position on the canvas and click to place it there. This will also add the dimension feature node to the geometry sequence. Change the dimension's value in the Settings window for the feature, then click **Build Selected** to see its effect.

CONSTRAINT AND DIMENSION SYMBOLS

In the Graphics window the constraint and dimension features are visualized with symbols, arrows, and dimension values. These are displayed when the sketch visualization (see The Sketch Visualization) is turned on. To turn on sketch visualization click the **Sketch** ( ) button on the **Geometry**, **Sketch**, or **Work Plane** toolbars. You can use the constraint and dimension symbols in the Graphics window in the following ways:

- When you hover over a symbol, it will be highlighted in red, together with the geometric entities it acts on. The name of the constraint or dimension feature appears in the top-left corner of the Graphics window. To turn off the display of the name when hovering over constraint and dimension symbols, clear the **Show information on hover** checkbox on the **Graphics** > **Interaction** > **Hovering** page in the **Preferences** window.
- Use the scroll wheel to cycle through overlapping symbols.
- Drag the symbol to change its position.
- Double-click the symbol to select the corresponding feature node in the geometry sequence, so that its settings are shown in the Settings window. For a dimension, this also gives focus to the dimension value text field.
- Click to select the symbol. Hold down the Ctrl key to select several symbols. Selected symbols are blue in the Graphics window.
- Click the **Delete** ( ) toolbar button or press the Delete key to delete the selected constraints and dimensions.

STATUS OF THE CONSTRAINTS AND DIMENSIONS

Before any constraints or dimensions are applied to a geometry it is possible to grab any of the edges or points of the geometry and move it around in all directions. In this state the geometry is *underdefined*. All geometric entities are possible to move, and they have a gray color in the Graphics window.

As you add constraints or dimensions to the geometry, the software automatically modifies the geometry and computes the remaining degrees of freedom — that is, the directions in which it is still possible to move the geometric entities. The edges and vertices that are not possible to move become black in the Graphics window to signify that they are uniquely defined. In this state the geometry is still underdefined.

When a sufficient number constraints and dimensions are added, all degrees of freedom become locked, so that none of the vertices or edges can be moved. In this state all geometric entities are black and the geometry is now *well defined*.

Adding further dimensions to the geometry may cause it to be *overdefined*. In such a state, the geometric entities are colored magenta in the Graphics window. Note that entities might also be colored magenta due to other problems (for example, in case a constraint cannot be applied to the selected entities).

The overall status of the applied constraints and dimensions is also indicated in the Settings window for the **Geometry** or the **Plane Geometry** node, at the bottom of the **Constraints and Dimensions** section. The following status messages can appear there:

TABLE 2-1: CONSTRAINT AND DIMENSION STATUS MESSAGES.

STATUS	DESCRIPTION
Underdefined	There are too few constraints and dimensions to define the geometry uniquely. Add more constraints and dimensions.
Well defined	The constraints and dimensions define the geometry uniquely, and they do not contradict each other. Note that redundant constraints are accepted in a well-defined geometry, but not redundant dimensions.
Overdefined	There are too many dimensions. Remove dimensions to get a well defined geometry.
Overdefined and underdefined	There are some conflicting or redundant constraints/ dimensions but the geometry is not uniquely defined.
Inconsistent	The software was unable to satisfy the constraints and dimensions, but a solution might exist.

TABLE 2-1: CONSTRAINT AND DIMENSION STATUS MESSAGES.

STATUS	DESCRIPTION
Need X more constraints/ dimensions	Add X constraints/dimensions to get a well defined geometry.
Y rigid body degrees of freedom	Of the X remaining degrees of freedom, Y degrees of freedom are rigid body motions (translation and rotation of the whole geometry).
X too many constraints/ dimensions	Remove X constraints/dimensions to get a well defined geometry.

Note that the status concerns the state of the geometry when it was last built, that is, the geometry you see in the Graphics window.

Each constraint and dimension node in the Model Builder can have a warning subnode that tells if the node is overdefined or inconsistent. When this happens, the software tries to satisfy the other constraints and dimensions, ignoring the problematic ones.

BUILDING A SUBSET OF THE CONSTRAINT AND DIMENSION FEATURES

To troubleshoot an overdefined or inconsistent state for constraints and dimensions and to find conflicting constraints or dimensions you can disable the corresponding features in the geometry sequence. Select one or several nodes in the Model Builder, then right-click the selected nodes and select **Disable**. Rebuild the geometry sequence to see the effect.

Another option is to change how the constraint and dimension features are built in the geometry sequence. To do this, in the Settings window for the **Geometry** or the **Plane Geometry** node, in the **Constraints and Dimensions** section, the **Constraint and dimension features to build** list provides the following options:

- **All**: This option means that all constraint and dimension features in the sequence are applied, also the features that come after the feature which you are building up to. This is the default setting.
- **None**: Use this option to turn off the build of constraint and dimension features. The corresponding feature nodes are grayed out in the geometry sequence. Note that built-in dimensions defined by the geometric primitive features and other features may still apply.
- **Up to build target**: With this option selected only the constraint and dimension features up to the feature you are building up to are applied to the geometry.

MEASURING DIMENSIONS

Dimensions are created by default as *constraining dimensions* that are treated as a requirement on the drawing. In the settings for a dimension you can change it to become a *measuring dimension* by toggling the **Constrain** button from the locked padlock ( ) to the open padlock ( ). A measuring dimension is not imposing a requirement on the entities it applies to, but rather it is updated according to the other features in the geometry sequence. In the Graphics window, measuring dimensions are displayed in a blue color that is lighter than that of the constraining dimensions. Measuring dimensions are useful when you want to know the value of dimension in the drawing but adding a constraining dimension would overdefine the geometry.

DIMENSION PARAMETERS

To use the value of a dimension as a parameter in other geometry and nongeometry features select the **Create measuring parameter** checkbox in the settings for the dimension. This setting is available for both constraining and measuring dimensions. The following conditions apply when using dimension parameters:

- You can use a dimension parameter in the expression for another dimension's value, as well as expressions in other geometry features, within the same 2D geometry sequence. The geometry feature where you use the dimension parameter may come above the dimension feature that defines the parameter. However, the following features and properties do not support dimension parameters:

- Copy
- Edit Object
- Else If
- If
- Move
- Offset
- Parametric Curve
- Part Instance
- Partition Edges
- Selections
- Tangent
- Thicken
- Array: Size
- Cubic Bezier and Quadratic Bezier: Weights
- Directed Distance: Direction
- Interpolation Curve: End Conditions
- Scale: Scale Factor
- Primitives: Layers
- In any feature: Relative tolerance.

• When used in a geometry feature in the same 2D geometry sequence, the expression containing the dimension parameters must be linear in the dimension parameters. For example, if `A` is a parameter defined in a **Global Definitions** > **Parameters** node and `geom1.dist3` is a dimension parameter, you can enter the expression `2+A^2*geom1.dist3`, but not `2+A^2*geom1.dist3^2`.

• The **Condition** expression in **If** and **Else If** features does not support dimension parameters.

• You can use a dimension parameter that is defined by a feature in a work plane's plane geometry sequence in expressions in 3D geometry features that come below the work plane feature.

- You can use dimension parameters created in a component in expressions in nongeometry features in that component as well as in other components, for example in physics, mesh, and materials settings.
- You can use dimension parameters created in a component in features under a **Study**, **Results**, and **Global Definitions** > **Materials** nodes.
- When entering expressions in a text field press Ctrl+Space to display a menu where you find available dimension parameters under **Geometric dimensions**.
- For dimensions that define a parameter, the name of the dimension parameter is displayed in the Graphics window, in the dimension's label.

HELP POINTS

For some constraint and dimension features the Settings window contains a section **Help Points** where you can specify coordinates for the help points on curved edges. The help points are used as initial guesses when solving the constraint or dimension. More specifically, when there are several solutions to a constraint or dimension, the software chooses the solution that is closest to the help points. When you add a constraint or dimension using a toolbar button, which is the recommended way, the help points are based on the position of the mouse click for the edge selection. When you add a constraint or dimension using the context menu in the Model Builder window the coordinates for the help points are instead set to (0,0), which may sometimes result in a solution that is not desired. The software updates the help point coordinates when the geometry is built, so that the help points correspond to the found solution.

Angle

Use the Angle dimension to specify the angle between two edges. The specified angle is applied between the tangent rays at the point of intersection for the two curves underlying the edges.

Add an Angle dimension as follows:

1. Click the **Angle** ( ) toolbar button.
2. Select two edges, or a vertex that has exactly two adjacent edges.
3. Move the mouse pointer to select one of the four angular sectors to measure, and to adjust the radial position of the circular arrow symbol.
4. Click to place the symbol.
5. In the Settings window for **Angle** change the value in the **Angle** text field.
6. Click **Build Selected** to rebuild the geometry with the new value.

The Settings window for **Angle** contains the following sections:

- **First Ray**: Change the selection for the first edge. The **Reverse direction** checkbox determines the direction of the edge's tangent ray at the intersection point.
- **Second Ray**: Change the selection for the second edge. The **Reverse direction** checkbox determines the direction of the edge's tangent ray at the intersection point.
- **Dimension Value**:
  - Adjust the value of the angle. The angle is measured from the first tangent ray to the second tangent ray in the counterclockwise direction.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.
- **Help Points**: Change the help point coordinates for the two edges. These are used as initial guesses when computing a point of intersection (see Help Points).

Coincident

Use the Coincident constraint to constrain two geometric entities to coincide with each other. Depending on the type of entities selected for the constraint the following conditions may apply:

- A vertex coincides with a straight edge if the vertex lies on the line.
- A vertex coincides with a circular edge if the vertex lies on the circle.
- A vertex coincides with a spline edge if the vertex lies on the spline parameterization (but possibly outside the edge's parameter interval).
- Two straight edges coincide if they lie on the same line.
- Two circular edges coincide if they lie on the same circle.
- Two spline edges coincide if they have the same spline parameterization (but possible different parameter intervals).

Add a Coincident constraint as follows:

1 Click the **Coincident** (≡) toolbar button.
2 Select two vertices or edges, or one vertex and one edge.
3 Move the mouse pointer to position the coincident symbol.
4 Click to place the symbol.

The Settings window for **Coincident** contains the following sections:

- **Geometric Entity Selection**: Change the selected entities and their type.
- **Help Points**: Change the help point coordinates for the selected entities. These are used as initial guesses when making a vertex coincident with a curved edge (see Help Points).

Concentric

Use the Concentric constraint to constrain circular edges to have the same center, or to constrain the center point of circular edges and vertices to coincide.

Add a Concentric constraint as follows:

1 Click the **Concentric** (◎) toolbar button.
2 Select circular edges and/or vertices.
3 Move the mouse pointer to position the concentric symbol.
4 Click to place the symbol.

The Settings window for **Coincident** contains the following sections:

- **Geometric Entity Selection**: Change the selected circular edges and vertices.

Directed Distance

Use the Directed Distance dimension to set the distance in a specified direction between two geometric entities.

The directed distance from, or to, an edge is defined using a stationary point for the directed point-to-point distance along the edge. For example, there are four ways to define the directed distance between two circles.

Add a Directed Distance dimension as follows:

1 Click the **Directed Distance** (⟶) toolbar button.

2 Select two vertices or curved edges, or a vertex and a curved edge.

3 Optionally, select a straight edge that specifies the direction in which the distance between the entities is measured.

4 Move the mouse pointer to position the arrow symbol.

5 Click to place the symbol.

6 If you have not chosen a straight edge in Step 3 above, in the Settings window for **Directed Distance** specify the components of the direction vector in the **x** and **y** text fields.

7 If you have chosen a straight edge in Step 3, you can optionally change the **Direction** for the dimension from **Parallel with edge** (default) to **Perpendicular to edge**.

8 Edit the value in the **Distance** text field. Note that the distance can also be negative or zero.

9 Click **Build Selected** to see the effect of the new distance value.

The Settings window for **Directed Distance** contains the following sections:

- **Geometric Entity Selection**: Select the entities for the dimension.
- **Direction**: Set the direction in which the distance is measured by selecting one of the options from the **Direction** list box:
  - **Vector**: Specify the direction by the **x** and **y** components of a direction vector. This is the default option.
  - **Parallel with edge**: The direction is parallel with the edge in the **Straight edge** selection.
  - **Perpendicular to edge**: The direction is perpendicular to the edge in the **Straight edge** selection.
- **Dimension Value**:
  - Enter the distance expression in the **Distance** text field. The distance may be positive, negative, or zero. The sign does not matter when **Direction** is set to **Perpendicular to edge**.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in

sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.

- **Help Points**: Change the help point coordinates for the selected entities. The help points are used as the initial guess to determine which directed distance to measure (see Help Points).

Distance

Use the Distance dimension to set the distance between two geometric entities. The distance from, or to, an edge is defined using a stationary point for the point-to-point distance along the edge. For example, there are four ways to define the distance between two circles. When you apply it to two straight edges, the distance dimension constrains the edges to be parallel. For a single edge, the distance is applied between the endpoints for the edge.

Add a Distance dimension as follows:

1. Click the **Distance** ( ) toolbar button.
2. Select two vertices or edges, a vertex and an edge, or an edge.
3. Move the mouse pointer to position the arrow symbol.
4. Click to place the symbol.
5. In the Settings window for **Distance** edit the value in the **Distance** text field.
6. Click **Build Selected** to see the effect of the new distance value.

The Settings window for **Distance** contains the following sections:

- **Geometric Entity Selection**: Change the selected entities and their type.
- **Dimension Value**:
  - Enter the distance expression in the **Distance** text field.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's

label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.

- **Help Points**: Change the help point coordinates for the selected entities. The help points are used as the initial guess to determine which distance to measure (see Help Points).

Equal Distance

Use the Equal Distance constraint to constrain the distances between two pairs of geometric entities to be equal. The distance from and to an edge is defined using a stationary point for the point-to-point distance along the edge. For example, there are four ways to define the distance between two circles.

The help points (the points where you clicked when selecting the edges) are used to determine which distance to measure.

Add an Equal Distance constraint as follows:

1. Click the **Equal Distance** () toolbar button.
2. Select two vertices or edges, or a vertex and an edge.
3. Move the pointer to position the arrow symbol, then click to place the symbol for the first pair of entities.
4. Again, select two vertices or edges, or a vertex and an edge.
5. Click to place the second arrow symbol.

   The same label will be displayed with the arrow symbols to indicate the constraint.

The Settings window for **Equal Distance** contains the following sections:

- **First Distance**: Change the selected entities and their type for the first pair.
- **Second Distance**: Change the selected entities and their type for the second pair.
- **Graphics**: Edit the label displayed with the arrow symbol for the constraint.
- **Help Points**: Change the help point coordinates for the selected entities. The help points are used as the initial guess to determine which distance to measure (see Help Points).

Equal Radius

Use the Equal Radius constraint to constrains two circular edges to have the same radius.

Add an Equal Radius constraint as follows:

1. Click the **Equal Radius** ( ) toolbar button.
2. Select a circular edge.
3. Move the pointer to position the arrow symbol, then click to place the symbol for the first edge.
4. Select another circular edge.
5. Click to position the second arrow symbol.

   The same label will be displayed with the arrow symbols to indicate the constraint.

The **Settings** window contains the following sections:

EDGE SELECTION

Change the selected entities.

GRAPHICS

Edit the label displayed with the arrow symbol for the constraint.

Horizontal

Use the Horizontal constraint to constrain straight edges to be parallel to the x-axis.

Add a Horizontal constraint as follows:

1. Click the **Horizontal** ( ) toolbar button.
2. Select straight edges.
3. Move the pointer to position the parallel symbol, then click to place it.

The Settings window for **Horizontal** contains the following section:

EDGE SELECTION

Change the selected entities.

Parallel

Use the Parallel constraint to constrain straight edges to be parallel.

Add a Parallel constraint as follows:

1. Click the **Parallel** ( ) toolbar button.
2. Select straight edges.

3 Move the pointer to position the parallel symbol, then click to place it.

The Settings window for **Parallel** contains the following section:

EDGE SELECTION

Change the selected entities.

Perpendicular

Use the Perpendicular constraint to constrain two straight edges to be orthogonal.

Add a Perpendicular constraint as follows:

1. Click the **Perpendicular** (⊥) toolbar button.
2. Select two straight edges, or a vertex that has exactly two adjacent edges.
3. Move the pointer to position the perpendicular symbol, then click to place it.

The settings window for **Perpendicular** contains the following section:

- **Edge Selection**: Change the selected entities.

Position

Use the Position dimension to specify the coordinates for a vertex.

Add a Position dimension as follows:

1. Click the **Position** ( ) toolbar button.
2. Select a vertex.
3. Click somewhere on the canvas to place the position symbol.
4. In the Settings window for **Position** edit the coordinates in the **x** and **y** text fields.
5. Click **Build Selected** to rebuild the geometry with the new coordinates for the vertex.

The Settings window for **Position** contains the following sections:

- **Vertex Selection**: Change the selected vertex.
- **Coordinates**:
  - Edit the coordinate expressions.
  - Toggle the **Constrain** buttons to switch between using the dimensions as constraining dimensions (, default) or measuring dimensions ( ). The value of a measuring dimension is not imposed as a requirement to the drawing.

Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.

- Select the **Create measuring parameters** checkbox to generate a parameter for the dimensions. Edit the names displayed in the **x parameter name** and **y parameter name** text fields. When in sketch visualization mode, the parameter names are displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.

Radius

Use the Radius dimension to set the radius for a circular edge.

Add a Radius dimension as follows:

1 Click the **Radius** ( ) toolbar button.

2 Select a circular edge.

3 Click somewhere on the canvas to place the arrow symbol.

4 In the Settings window for **Radius** edit the value in the **Radius** text field.

5 Click **Build Selected** to rebuild the geometry with the new radius.

The Settings window for **Radius** contains the following sections:

- **Edge Selection**: Change the selected circular edge.
- **Dimension Value**:
  - Edit the radius expression.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.

Tangent Constraint

Use the Tangent Constraint to constrain two edges to have a point of tangency. For each edge, you can optionally specify an adjacent vertex as the point of tangency.

Add a Tangent Constraint as follows:

1. Click the **Tangent Constraint** ( ) toolbar button.
2. Select two edges (and optionally a vertex adjacent to each edge), or a vertex having exactly two adjacent edges.
3. Click somewhere on the canvas to place the tangent symbol.

The Settings window for **Tangent Constraint** contains the following sections:

- **First Edge**: Change the selected edge. To set the point of tangency to a vertex, from the **Point of tangency** list, choose **Vertex**, then activate the **Vertex** selection, and select a vertex adjacent to the edge. With the default setting for **Point of tangency**, **Anywhere**, the tangency can be applied anywhere on the edge.
- **Second Edge**: Change the selected edge and point of tangency similarly as in the **First Edge** section.
- **Help Points**: Change the help point coordinates for the selected edges. The help points are used as the initial guess when computing the intersection of the edges if a vertex has not been selected (see Help Points).

Total Edge Length

Use the Total Edge Length dimension to specify the total length for a set of edges. The edges must form a chain and all lie on the same line, circle, or spline.

Add a Total Edge Length dimension as follows:

1. Click the **Total Edge Length** ( ) toolbar button.
2. Select edges that form a chain on the same line, circle, or spline.
3. Click somewhere on the canvas to place the arrow symbol.
4. In the Settings window for **Total Edge Length** edit the value in the **Length** text field.
5. Click **Build Selected** to rebuild the geometry with the new length.

The Settings window for **Total Edge Length** contains the following section:

- **Edge Selection**: Change the selected edges.

- **Dimension Value**:
  - Edit the length expression.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.

Vertical

Use the Vertical constraint to constrain straight edges to be parallel to the y-axis.

Add a Horizontal constraint as follows:

1. Click the **Vertical** ( ) toolbar button.
2. Select straight edges.
3. Move the pointer to position the parallel symbol, then click to place it.

The Settings window for **Vertical** contains the following section:

EDGE SELECTION

Change the selected entities.

x-Distance

Use the **x-Distance** dimension to set the distance in the x direction between two geometric entities. The x-distance from or to an edge is defined using a stationary point for the point-to-point x-distance along the edge. For example, there are four ways to define the x-distance between two circles.

Add an x-distance dimension as follows:

1. Click the **x-Distance** ( ) toolbar button.
2. Select two vertices, two curved edges, or a vertex and a curved edge.
3. Click somewhere on the canvas to place the arrow symbol.

4 In the Settings window for **x-Distance** edit the value in the **Distance** text field. Note that the distance can also be negative or zero.

5 Click **Build Selected** to rebuild the geometry with the new distance value.

The Settings window for **x-Distance** contains the following sections:

- **Geometric Entity Selection**: Change the selected entities and their type.
- **Dimension Value**:
  - Enter the distance expression in the **Distance** text field. The distance may be positive, negative, or zero.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.
- **Help Points**: Change the help point coordinates for the selected entities. The help points are used as the initial guess to determine which x-distance to measure (see Help Points).

y-Distance

Use the **y-Distance** dimension to set the distance in the y direction between two geometric entities. The y-distance from or to an edge is defined using a stationary point for the point-to-point y-distance along the edge. For example, there are four ways to define the y-distance between two circles.

Add a y-distance dimension as follows:

1 Click the **y-Distance** ( ) toolbar button.

2 Select two vertices, two curved edges, or a vertex and a curved edge.

3 Click somewhere on the canvas to place the arrow symbol.

4 In the Settings window for **y-Distance** edit the value in the **Distance** text field. Note that the distance can also be negative or zero.

5 Click **Build Selected** to rebuild the geometry with the new distance value.

The Settings window for **y-Distance** contains the following sections:

- **Geometric Entity Selection**: Change the selected entities and their type.
- **Dimension Value**:
  - Enter the distance expression in the **Distance** text field. The distance may be positive, negative, or zero.
  - Toggle the **Constrain** button to switch between using the dimension as a constraining dimension (, default) or measuring dimension ( ). The value of a measuring dimension is not imposed as a requirement to the drawing. Instead it is updated according to other dimensions and constraints in the geometry sequence. For more information see Measuring Dimensions.
  - Select the **Create measuring parameter** checkbox to generate a parameter for the dimension. Edit the name displayed in the **Parameter name** text field. When in sketch visualization mode, the parameter name is displayed in the dimension's label in the Graphics window. Use dimension parameters in expressions in geometry and other features as described in Dimension Parameters.
- **Help Points**: Change the help point coordinates for the selected entities. The help points are used as the initial guess to determine which y-distance to measure (see Help Points).

Geometry Representation

Working with the CAD Kernel

The component of the COMSOL Multiphysics® software that is used to represent, build, and manage the interactions between geometric objects is the geometric kernel or geometric modeler. There are two kernels used by the software, the *COMSOL kernel*, and the *CAD kernel* (the Parasolid® kernel) that is included with the CAD Import Module, the Design Module, and LiveLink™ products interfacing CAD packages.

With a license for the Design Module the software defaults to the CAD kernel for representing the geometry. You need to use the CAD kernel to apply the geometry features included with this module, for example the defeaturing and repair tools, as well as to import 3D geometries using various 3D CAD file formats. Exceptions are the constraint and dimension features that do not require the CAD kernel.

The 3D operations and primitives listed in Table 2-2 do not support the CAD kernel — they always use the COMSOL kernel. However, an automatic conversion is performed for these objects before they are used as input to geometry features that require the CAD kernel, see Converting Objects to CAD Kernel Representation.

TABLE 2-2: 3D GEOMETRY FEATURES THAT DO NOT SUPPORT THE PARASOLID GEOMETRY KERNEL.

FEATURE NAME	FEATURE NAME
Bezier Polygon	Point
Eccentric Cone	Polygon
Extrude	Pyramid
Helix	Revolve
Hexahedron	Sweep
Interpolation Curve	Tetrahedron
Parametric Curve	Torus
Parametric Surface	Work Plane

CHANGING THE GEOMETRIC KERNEL

To switch between geometric kernels, you can click the **Geometry** node, then in its Settings window, from the **Geometry representation** list choose either the **CAD kernel** or **COMSOL kernel**.

When you change the **Geometry representation** setting, all nodes that support the CAD kernel are marked as edited with an asterisk (*) in the upper-right corner of the node's icon. To rebuild the geometry using the new kernel, click the **Build All** button ( ). To avoid re-solving an already solved model, you can click the **Update Solution** button ( ) in the **Study** toolbar to map the solutions from the geometry represented by the CAD kernel to the new geometry represented by the COMSOL kernel.

> If you solve a model using the CAD kernel, it is not possible to view and postprocess the solution if you open it in a COMSOL Multiphysics session where a license for the CAD Import Module, Design Module, or one of the LiveLink for CAD products is not available, unless, before saving the model, you change the geometry representation to COMSOL kernel and update the solution. This is possible to do only for 3D geometry sequences that do not contain geometry features that require the CAD kernel.

When you create a new model, its default geometry representation is controlled by the preference setting **Geometry** > **Geometry representation** > **In new geometries**.

When you open an existing model, you normally use the geometry representation used in the model. To always get the possibility to convert the geometry to the COMSOL kernel, change the preference setting **Geometry** > **Geometry representation** > **When opening an existing model** to **Convert to COMSOL kernel**.

USING THE DESIGN MODULE BOOLEAN OPERATIONS

The Design Module includes 3D Boolean geometry operations that may in some cases be more successful in computing the resulting geometry, for example when building the union of objects that intersect or have gaps. By specifying an absolute tolerance that corresponds to the width of the overlap or gap, the intersecting region is collapsed or the gap is closed when building the union operation. To find appropriate tolerance values use the *Detect Interferences* tool.

If your license includes the Design Module and the geometry representation is set to use the CAD kernel, you can select the **Design Module Boolean operations** checkbox in a 3D **Geometry** node's **Settings** window to use the Boolean operations available with the Design Module. When you open an existing model, you normally use the Boolean operations used in the model. To always use the Design Module Boolean operations when adding a new component or part, select the preference setting **Geometry** > **3D Design Module Boolean operations** > **Use in new geometries**.

Converting Objects to COMSOL Kernel Representation

To convert CAD objects (geometric objects represented by the CAD kernel) to objects represented by the COMSOL kernel, from the **Geometry** toolbar, **Conversions** menu, select **Convert to COMSOL** ( ).

The COMSOL geometry file format (.mphbin, or .mphtxt) can contain geometric objects saved in both the CAD kernel and COMSOL kernel representations. To import geometry from such a file to a geometry sequence that uses the COMSOL kernel, you need to convert geometry objects to the COMSOL representation before exporting to the file.

CONVERT TO COMSOL

Select the objects that you want to convert in the **Graphics** window. The selected objects are displayed in the **Input objects** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects.

For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Converting Objects to CAD Kernel Representation

If the current geometry representation for the geometry sequence is **CAD kernel**, an automatic conversion of COMSOL objects to CAD objects takes place before using the objects in Boolean operations and before using the objects in the **Convert to Solid**, **Convert to Surface**, **Convert to Curve**, and **Convert to Point** operations. This ensures that the CAD kernel is used in the abovementioned operations. This conversion is also performed when COMSOL objects are used as input to features that require the CAD kernel, for example the **Knit to Solid** feature

An automatic conversion to CAD objects is also performed before exporting geometry in the ACIS®, Parasolid®, STEP, and IGES file formats.

If the automatic conversion cannot be performed, the geometry operation is performed by the COMSOL kernel. For example, geometry objects created from a mesh cannot be converted to CAD kernel representation. Other examples of geometry objects that cannot be converted to CAD representation include objects that have an edge adjacent to three or more isolated faces, or objects that have a face bounded by an edge loop that intersects itself.

The automatic conversion to CAD kernel representation is not performed if one of the input objects to the Boolean or conversion operation is the result of a previous **Convert to COMSOL** operation.

Importing and Exporting CAD Files

Importing 3D CAD Files

To import geometry objects from a 3D CAD file, from the **Home** or the **Geometry** toolbar, click **Import** ( ).

SOURCE

In the **Source** section of the Settings window, select **3D CAD file** from the **Source** list. You can also skip this step as the type of the selected file is automatically recognized by the code. Click **Browse** to locate the file to import, or enter the path to the file. Before clicking the **Import** button consider to review and configure the import settings. If you have changed some settings after importing a file, the file is automatically reimported when you click a build button.

When importing STEP files that contain multibody parts it can be useful to generate object names that include both the body and the part names retrieved from the file. Choose how to name the objects imported from STEP files from the **Import body names** list:

- Choose **Automatic** to include the body names in the object name only for the multibody parts.
- Choose **On** to include the body names in the objects names for all imported parts.
- Choose **Off** to not include the body names in the object names.

The imported geometry objects are represented by the CAD kernel, see Working with the CAD Kernel, which is the geometric kernel used by the CAD Import Module, Design Module, and LiveLink™ products interfacing CAD packages.

Some 3D CAD formats use periodic parameterization for edges and faces. For example, a full-revolution cylindrical edge or face appears seamless in the CAD program. During import edges or faces that have a periodic parameterization are cut in two halves by inserting new vertices and edges. This is done because the mesh algorithms do not support periodic entities. You can ignore such inserted edges using an **Ignore Edges** feature from **Virtual Operations**.

Supported Formats

The CAD import supports the following 3D CAD formats:

TABLE 2-3: SUPPORTED 3D CAD FILE FORMATS.

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
ACIS®	1, 2	`.sat`, `.sab`	Up to 2025 1.0
AutoCAD®	1, 3	`.dwg`, `.dxf`	2.5-2026
CATIA® V5	3, 4	`.CATPart`, `.CATProduct`	R8 to R2025
IGES	1, 2	`.igs`, `.iges`	Up to 5.3
Inventor® assembly	1, 3	`.iam`	11-2026
Inventor® part	1, 3,	`.ipt`	6-2026
NX™	1, 3	`.prt`	Up to 2506
Parasolid®	1	`.x_t`, `.x_b`	Up to V38.0
PTC Creo Parametric™	1, 2, 5	`.prt`, `.asm`	1-12
PTC Pro/ENGINEER®	1, 2	`.prt`, `.asm`	16 to Wildfire 5
SOLIDWORKS®	1, 3, 6	`.sldprt`, `.sldasm`	98-2025
STEP	1, 2	`.step`, `.stp`	AP203E1, AP214, AP242

Note 1: This format requires a license for one of the CAD Import Module, Design Module, or LiveLink product for a CAD package.

Note 2: This format is available only on supported Windows® and Linux operating systems with Intel® 64-bit processors and on macOS.

Note 3: This format is available only on supported Windows® and Linux operating systems with Intel® 64-bit processors.

Note 4: This format requires, in addition to the CAD Import Module, or Design Module, or a LiveLink product for a CAD package, a license for the File Import for CATIA V5 module.

Note 5: The import of files saved with PTC Creo Parametric 12 software is available only on supported Windows® and Linux operating systems with Intel® 64-bit processors.

Note 6: Files saved with an educational version of the SOLIDWORKS® software are not supported.

Preview of CAD Assemblies

When importing CAD assemblies saved in the file formats of the CATIA V5, Inventor, NX, PTC Creo Parametric, PTC Pro/ENGINEER, and SOLIDWORKS software you can display a graphical preview of the assembly components by clicking the **Preview** ( ) button in the **Settings** window toolbar. The components are displayed in the **Graphics** window, where you can also select the components you want to import. See also Assembly Components to Import.

Note: Displaying a preview of assemblies with embedded parts saved in the file format of the SOLIDWORKS software are not supported. To preview such an assembly, first convert the embedded parts to external parts.

Associativity

When possible the import maintains associativity for the imported geometry objects, so that when the CAD file is reimported the settings applied to the geometric entities, for example physics or material settings, are retained. To maintain associativity the import relies on information in the CAD file that uniquely identifies the geometry objects and their entities, such as faces, edges, and points. This information is usually included in the CAD file if the geometry is saved in the format of the CAD software where it was created, but not when the geometry is exported to another CAD format. When reimporting a CAD file the import automatically tries to identify and match all geometry objects and their entities to the previous version. This may fail if the topology (structure) of the geometry has changed since the last import.

Note: To ensure that associativity is maintained when reimporting a CAD file work with CAD files saved in the originating CAD software's format, and avoid changes to the topology (structure) of the geometry. When an associative import is not possible use coordinate-based selections, such as the Ball, Box, and Cylinder selections in 3D (see Creating Selections From Geometric Primitives and Operations in the *COMSOL Multiphysics Reference Manual*).

LENGTH UNIT

In the **Length unit** list, select **From CAD document** to change the geometry's length unit to the unit in the file (if the file has a length unit). Select **From COMSOL** to keep the geometry's length unit and scale the objects in the file to the geometry's unit.

ASSEMBLY COMPONENTS TO IMPORT

When importing CAD assemblies saved in the file formats of the CATIA V5, Inventor, NX, PTC Creo Parametric, PTC Pro/ENGINEER, and SOLIDWORKS software you can display the assembly tree where you can choose the components to import from the assembly. After you have selected an assembly file under the **Source** section, click **Load Assembly Tree** to display the components of the selected assembly. You can also click the **Preview** ( ) button in the **Settings** window toolbar to load the assembly tree and to show a preview of the components in the **Graphics** window.

When you click **Import** ( ) under the **Source** section, only the components that are selected in the displayed assembly tree will be imported. Thus, to prevent importing a component you can clear its checkbox in the tree. To quickly select or clear all similar components, you can right-click a component, and from the menu choose to either **Select All Instances** or **Clear All Instances** of the component. Components can also be selected from the **Graphics** window when **Preview** ( ) is active.

For large assemblies it may help to filter the assembly tree to display only the components that you have selected to import by selecting the **Only list components to import** checkbox. You can also enter any text in the **Filter** text box to only display the component nodes that match the filter.

The **Select All** ( ) and **Clear All** ( ) buttons below the assembly tree provide further shortcuts for efficiently choosing components. Use the **Expand All** ( ) and **Collapse All** ( ) buttons to quickly expand and collapse the displayed assembly tree. When the assembly has changed on disk, or if you select a different version of the assembly, click the **Reload Assembly Tree** ( ) button to update the tree displayed in the Settings window. When reloading the assembly tree the selection of components is retained, and any new components are selected for import by default.

OBJECT TYPES TO IMPORT

Select the types of objects to import using the **Solids**, **Surfaces**, and **Curves and points** checkboxes.

If the **Surfaces** checkbox is selected, you can choose how COMSOL imports the surfaces using the list under **For surface objects**:

- Choose **Form solids** (the default) to knit together surface objects to form solids. The input surface objects must have manifold topology, and the operation can only form solids with manifold topology. An example of a solid object with nonmanifold topology is a solid that has an interior surface that separates two domains. A surface object that contains an edge that is adjacent to more than one boundary is an example of a surface object with nonmanifold topology.

- Choose **Knit** to form surface objects by knitting.
- Choose **Do not knit** to not form any surface or solid objects from the imported surfaces.

For the **Form Solids** and **Knit** options, the knitting merges edges that have a distance smaller than the **Absolute import tolerance** and deletes gaps and spikes smaller than the **Absolute import tolerance**. Additionally, select the **Fill holes** checkbox to generate new faces that can cover larger holes due to missing faces in the imported CAD file.

If a solid cannot be created while importing, you can also try the Knit to Solid operation after the import, and use a larger tolerance for the knitting. Geometry operations, see for example the Cap Faces and Loft operations, can also help with creating new faces to cover larger holes on imported geometry.

To import wireframe geometry you need to select the **Curves and points** checkbox. With this option, the **Unite curve objects** checkbox is selected by default to unite the imported curve objects, which speeds up the rendering of the geometry.

SIMPLIFY AND REPAIR

The **Absolute import tolerance** is a length measured in the geometry's unit after the import. When importing 3D CAD files, this tolerance is used by some of the simplification and repair operations as detailed in this section. During the import, the repair operations are performed in the order of the corresponding settings (starting from the top) in the user interface.

If the **Remove redundant edges and vertices** checkbox is selected, edges and vertices that are considered redundant, such as the edges of an imprint on a face, are removed from the imported geometry.

Imported geometric objects can sometimes contain geometric and topological errors, including missing edges and vertices, and entities with invalid sense and invalid tolerance. Errors such as these can be fixed during the import by choosing one of the available options from the **Fix errors** list:

- Choose **Automatic** (the default) to first check the geometry for topological errors, and if issues are detected attempt to repair.

- Choose **On** to always run the repair operations during import. An initial check of the geometry is not performed in this case.
- Choose **Off** to turn off the repair of topological errors during the import. This can speed up the import if the geometry is known to be correct.

The option **Simplify curves and surfaces** is selected by default to simplify, within the **Absolute import tolerance**, the underlying curve and surface manifolds of the imported geometric entities. Importing objects with this option may improve both the performance and reliability of geometric operations on some imported geometry, for example it may help in some cases when Boolean operations on the imported objects fail. Simplification means that the manifolds are converted where possible to analytical form: linear, circular, and elliptical curves; and planar, spherical, cylindrical, conical, and toroidal surfaces. Manifolds that are converted are B-spline curves and surfaces, or certain surfaces generated by operations such as sweeping, revolving, and filleting.

The **Delete small details** checkbox is selected by default, to remove geometric details smaller than the **Absolute import tolerance**. Details that can be deleted include short edges, sliver faces, small faces, and spikes.

With the **Heal edges** checkbox selected the import tries to replace edges with a large tolerance with accurate edges calculated by intersecting the adjacent surfaces. This also improves the adjacent vertices, which become exact.

If the **Minimize tolerances** checkbox is selected tolerant edges and vertices on the geometry are detected, and their tolerances are reduced when possible.

Select the **Check resulting objects for errors** checkbox (default) to check the validity of the imported objects as the last stage of the import. Warning nodes appear with details about the detected problems, if any. Use the **Zoom to Selection** button next to the **Entities** list in a warning node to locate the problematic edges or faces. For information on geometry problems that may occur see the Check feature.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Select the **Individual objects selections** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence for each individual object in the geometry file and for each relevant entity level. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, if available, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS GENERATED BASED ON INFORMATION IN THE CAD FILE

The following types of data from the CAD file are used to generate selection on the imported geometry:

- Material assignments can generate objects selections that are named according to the material names in the CAD file.
- Layer assignments of objects and entities, when supported by the CAD format, can generate object, boundary, edge, and point selections that are named according to the layer names in the CAD file.
- Color assignments to objects, faces, or edges can generate object, boundary, and edge selections, respectively.

After the import the generated selections are displayed in the Settings window for the Import node in sections named according to the entity level of the selections:

- **Object Selections**
- **Boundary Selections**

- **Edge Selections**
- **Point Selections**

Depending on which selections are generated, a subset of the above sections is displayed. The selections are listed in tables with the following columns:

- **Name**: Here you can edit the selection name that is generated by the import. For colors the generated names are of the type *Color 1*, *Color 2*, and so on, for materials and layers the names from the CAD file are used.
- **Name in file**: This column contains the original name of the selection. To display this column select the **Show names from file** checkbox above the table.
- **Keep**: Select the checkbox in this column to make the selection available in selection lists for subsequent nodes in the geometry sequence.
- **Physics**: Select the checkbox in this column to make the selection available in all applicable selection lists (in physics and materials settings, for example).
- **Contribute to**: If you want to make the objects or entities in the selection contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New Cumulative Selection** button under the table to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Click a row in a table to highlight the corresponding selection on the geometry in the **Graphics** window. To help with identifying the color selections, these are highlighted with the colors defined in the imported CAD file. To always highlight on the geometry the color selections that you keep select from the **Graphics** toolbar **Colors** > **Show Selection Colors**.

The selections listed in the **Object Selections** section that are made available for the geometry sequence or physics setup are always available in all input selection lists, including all applicable entity selection lists. For example, the object selection of a solid object, generated for a material from the CAD file, automatically results in domain, boundary, edge, and point selections with the same name, so that you can use it to apply a boundary material, or a boundary condition. In contrast, a color assigned to a face of a solid object in the CAD file results in a boundary selection that is displayed in the **Boundary Selections** section, and it is available in all applicable boundary selection lists, but not, for example, in any edge selection lists.

ASSIGNED ATTRIBUTES

Select the **Construction geometry** checkbox to make the resulting objects available only in the feature's geometry sequence. For more information see Construction Geometry.

Exporting Objects to 3D CAD Formats

With a license for the Design Module you can export 3D geometry objects to the ACIS® (version 2016 1.0), IGES (version 5.3), Parasolid® (version 37.0), and STEP (version AP203) formats. To do this:

- right-click the **Geometry** node and select **Export** ( ), or
- in the **Geometry** toolbar click **Export** ( ).

Then, in the **Export** window, the **File type** list, select **Parasolid binary file**, **Parasolid text file**, **ACIS binary file**, **ACIS text file**, **IGES file**, or **STEP file**. Use the **Browse** button to choose the filename, or enter a filename including the path in the **Filename** field.

Next, select **Export entire finalized geometry** to export the resulting geometry of a Form Union or Form Assembly operation. Alternatively, select **Export selected objects**, then from the **Selection** list, choose **Manual** (default) to select the objects you want to export in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Selection** list.

The following limitations apply when exporting geometry objects to the 3D CAD formats mentioned in this section:

- Geometry objects that are created from mesh cannot be exported.
- Geometry objects that are the result of virtual geometry operations that come after a Form Union or Form Assembly node in the geometry sequence cannot be exported. The finalized geometry resulting from the Form Union or Form Assembly node is exported instead.
- Geometry objects created with a license for the ECAD Import Module from ECAD files imported with the options **Ignore vertices with continuous tangent** and **Eliminate short edges** cannot be exported. In this case the exported geometry objects contain the vertices and edges removed by the import.

COMSOL objects are automatically converted to CAD objects before saving the file.

For details on which objects can be converted to CAD objects see Converting Objects to CAD Kernel Representation.

To export the geometry to the specified file, click the **Export** ( ) button. A confirmation message appears in the Messages window.

ADVANCED

When exporting to an ACIS file format choose the **ACIS file format version**. Available versions are **4.0**, **7.0**, **2016 1.0** (default).

For the Parasolid, IGES, and STEP file formats select a **Length Unit**. A unit conversion is carried out when the selected unit is different from the length unit of the geometry. A unit conversion is not done for the default **From geometry** option.

The Parasolid binary and text formats do not allow coordinate values larger than 500. Therefore you might have to change the export unit in the **Length unit** list box to be able to export the geometry.

For the Parasolid file formats the option **Split in manifold objects** is selected by default to make sure that the exported geometry objects are manifold objects. A nonmanifold object is, for example, a solid with an interior boundary that separates two domains. When exported using this option the solid is split along the interior boundary into two separate objects. When exporting to the ACIS, IGES, and STEP formats nonmanifold objects are always split.

Repairing and Defeaturing

When importing 3D CAD files, the default import settings ensure that the validity of the imported objects is checked, and that defects are repaired when possible. In addition to the checks and repair performed during import, the Design Module provides operations for checking, repairing, and defeaturing 3D geometry objects, and locating overlaps and gaps in imported CAD assemblies.

OPERATIONS FOR CHECKING AND REPAIRING 3D OBJECTS

Use the operations listed in the table below to check and repair geometry objects and to detect interferences between objects:

TABLE 2-4: OPERATIONS FOR DETECTING DEFECTS IN AND REPAIRING 3D OBJECTS AND DETECTING INTERFERENCES BETWEEN 3D GEOMETRY OBJECTS.

ICON	NAME	DESCRIPTION
	Check	Check CAD objects for faults, for example tolerance issues and invalid entities
	Detect Interferences	Search for interferences, such as intersections, gaps, touches, and containments, between objects
	Repair	Repair defects and remove small details from 3D objects

OPERATIONS FOR DEFEATURING

With the defeaturing tools listed in the table below you can search for and delete both small details, such as short edges, small faces, sliver faces, and spikes, and larger details, for example, fillets, chamfers, and cylindrical holes. You can also replace and detach a selection of faces to form 3D objects.

TABLE 2-5: OPERATIONS FOR DEFEATURING 3D GEOMETRY OBJECTS.

ICON	NAME	DESCRIPTION
	Delete Fillets	Search for and delete fillets
	Delete Holes	Search for and delete holes
	Delete Short Edges	Search for and collapse short edges
	Delete Sliver Faces	Search for and delete slivers faces
	Delete Small Faces	Search for and collapse small faces

TABLE 2-5: OPERATIONS FOR DEFEATURING 3D GEOMETRY OBJECTS.

ICON	NAME	DESCRIPTION
	Delete Spikes	Search for and delete spikes from faces
	Detach Faces	Detach faces to form a new object from them
	Replace Faces	Replace faces by growing surrounding faces or creating new faces

Check

To check the validity of CAD objects, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Check** ( ).

From the **Input objects** list, choose **Manual** (default) to select the objects that you want to check in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Check** node, these will also be available in the **Input objects** list.

If any problems are detected in the selected objects when building this feature, warning nodes appear with details about the issues. In the warning nodes use the **Zoom to Selection** button next to the **Entities** list to locate the faulty edges or faces.

Warnings on geometric entities are usually associated with objects imported from CAD files, but could also be introduced by geometric operations, for example when repairing an object with a tolerance that is too large. The presence of warnings does not generally mean that the geometry is invalid and cannot be used for setting up a simulation. However, the faulty entities may in some cases cause the failure of geometric operations that involve these entities, and meshing of entities with certain types of faults may fail. In the following you can read about two commonly occurring class of faulty geometric entities, and how to repair these:

- Tolerance issues: The warning messages *vertex not on edge*, *vertex not on face*, *edge not on face* belong to this category. These type of faults indicate that the topology (structure) of the object is not correct locally, for example that an edge is not located on the boundary where it is expected to be. You can often repair tolerance issues in an object by applying the Repair operation with a tolerance that is larger than the tolerance used for the import. Repair adjusts the tolerance of the entities where required, and, if this fixes the issues, warning nodes will not be displayed after the Repair node. In some cases it may also help to use the option **Simplify curves and surfaces** for the Repair operation. Note that increasing the repair tolerance too much may lead to removing important details from the geometry. As an alternative to the Repair operation you can also reimport the CAD file using a larger tolerance. For more details see Repair, and Importing 3D CAD Files.

- Invalid entities: The warning messages *invalid curve or surface*, *self-intersecting face*, *face-to-face inconsistency*, *self-intersecting curve or surface* signify invalid entities. If you encounter any of these faults after importing a geometry from a CAD file you may need to repair the geometry to avoid problems with meshing. For example:
  - For face-to-face inconsistency faults on solid objects, try to repair the object using the **Repair face-to-face inconsistencies in solids** option for the Repair feature.
  - Try to replace the faulty face. Use the Delete operation to delete the face, then use the Cap Faces operation to generate a new face in its place. Another solution may be to use Delete Faces to delete and patch in one operation several connected faces that have a fault.

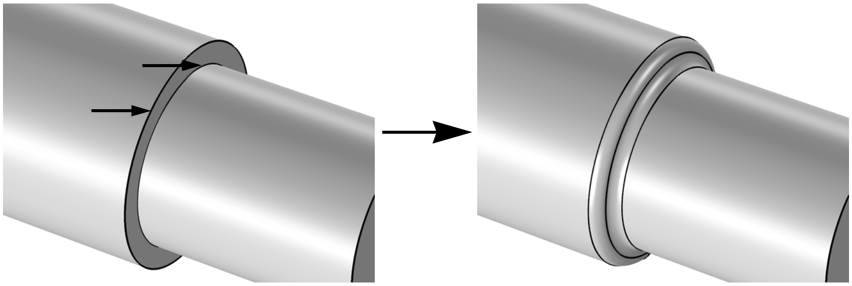
Delete Fillets

Use the Delete Fillets tool to search for fillets of a specified radius and delete these from an object. The faces forming the fillet are removed and the adjacent faces are extended to cover the resulting wound.

Note that this defeaturing tool may not find all fillets on nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To remove the fillets make sure to defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

The Delete Fillets tool cannot delete fillets for which the adjacent faces cannot be extended to cover the wound. The figure below shows an example of such fillets.

Applying the fillets on the highlighted edges deletes the annular face from the geometry, which cannot be recreated if the fillets are to be deleted.



To open the Tools window for **Delete Fillets**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Fillets** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Fillets**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE FILLETS

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

The Delete Fillets tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the fields **Minimum fillet radius** and **Maximum fillet radius**, enter the size of the fillets you want to search for. When you click the **Find Fillets** button, a list of fillets with radii between the given values is shown in the **Fillet selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Fillets contains the additional settings described below.

To delete all fillets returned by the search, set the **Deletion type** to **All fillets**. You can delete a subset of the found fillets by selecting them in the **Fillet selection** list, and choosing **Selected fillets** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Delete Holes

Use this defeaturing tool to search for and delete cylindrical holes from an object. The tool can find and delete both through or blind holes on solid as well as surface objects. The faces forming the hole are deleted and the resulting wound is covered by extending the adjacent faces.

Note that holes found on nonmanifold objects are not possible to delete. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To remove the holes make sure to defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

To open the **Tools** window for **Delete Holes**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Holes** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Holes**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE HOLES

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

> The Delete Holes tool can only be applied to objects that are represented by the CAD kernel; see Converting Objects to CAD Kernel Representation.

In the fields **Minimum hole radius** and **Maximum hole radius**, enter the size of the holes you want to search for. When you click the **Find Holes** button, a list of holes with radii between the given values is shown in the **Hole selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Holes contains the additional settings described below.

To delete all holes returned by the search, set the **Deletion type** to **All holes**. You can delete a subset of the found holes by selecting them in the **Hole selection** list, and choosing **Selected holes** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Delete Short Edges

Use the Delete Short Edges tool to find edges shorter than a specified length, and then to delete these by collapsing the edges.

Note that this defeaturing tool cannot find short edges on nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To avoid this situation defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

To open the **Tools** window for **Delete Short Edges**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Short Edges** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Short Edges**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE SHORT EDGES

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

The Delete Short Edges tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the field **Maximum edge length**, enter the maximum length of the edges you want to delete. When you click the **Find Short Edges** button, a list of edges with length smaller that the given value is shown in the **Short edge selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Short Edges contains the additional settings described below.

To delete all edges returned by the search, set the **Deletion type** to **All short edges**. You can delete a subset of the found edges by selecting them in the **Short edge selection** list, and choosing **Selected short edges** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Delete Sliver Faces

Use the Delete Sliver Faces tool to search for high aspect ratio faces of a specified maximum width. The tool deletes the selected sliver faces by collapsing these.

Note that this defeaturing tool cannot find sliver faces on nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To avoid this situation defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

To open the **Tools** window for **Delete Sliver Faces**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Sliver Faces** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Sliver Faces**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE SLIVER FACES

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

The Delete Sliver Faces tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the field **Maximum face width**, enter the maximum width of the faces you want to delete. When you click the **Find Sliver Faces** button, a list of faces with width smaller that the given value are shown in the **Sliver faces selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Sliver Faces contains the additional settings described below.

To delete all faces returned by the search, set the **Deletion type** to **All sliver faces**. You can delete a subset of the found faces by selecting them in the **Sliver face selection** list, and choosing **Selected sliver faces** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Delete Small Faces

Use the Delete Small Faces tool to find and delete faces of a specified maximum size. The tool deletes the selected small faces by collapsing these.

Note that this defeaturing tool cannot find small faces on nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To avoid this situation defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

To open the **Tools** window for **Delete Small Faces**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Small Faces** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Small Faces**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE SMALL FACES

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

The Delete Small Faces tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the field **Maximum face size**, enter the maximum diameter of the faces you want to delete. When you click the **Find Small Faces** button, a list of faces with diameter smaller that the given value appears in the **Small faces selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Small Faces contains the additional settings described below.

To delete all faces returned by the search, set the **Deletion type** to **All small faces**. You can delete a subset of the found faces by selecting them in the **Small face selection** list, and choosing **Selected small faces** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Delete Spikes

A spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges. Using the Delete Spikes tool you can search for and delete spikes from an object, by collapsing the narrow face region defined by the spike.

To open the **Tools** window for **Delete Spikes**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Delete Spikes** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Delete Spikes**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

DELETE SPIKES

From the **Input objects** list, choose **Manual** (default) to select the objects you want to examine in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

Note that this defeaturing tool cannot find spikes on faces that belong to nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can for example result from a Union or a Partition operation. To avoid this situation defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

The Delete Spikes tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the field **Maximum spike width**, enter the maximum width of the spikes you want to delete. When you click the **Find Spikes** button, a list of spikes with width smaller that the given value are shown in the **Spike selection** list.

To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.

If you want to modify the performed deletion operation, you can select the added node in the geometry branch. Then, edit the node's form that appears in the **Settings** window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Delete Spikes contains the additional settings described below.

To delete all spikes returned by the search set the **Deletion type** to **All spikes**. You can delete a subset of the found spikes by selecting them in the **Spike selection** list, and choosing **Selected spikes** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Detach Faces

By detaching, faces are removed from an object (the parent) and are used to form a new object (the child). The wound that results from detaching the faces is healed by either creating new faces based on the surrounding edges or by growing or shrinking adjacent faces.

To open the **Tools** window for **Detach Faces**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Detach Faces** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Detach Faces**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

Select the faces you want to detach in the **Graphics** window. They appear in the **Faces to detach** list. If the geometry sequence includes user-defined selections above the currently built node, choose **Manual** to select faces, or choose one of the selection nodes from the list next to **Faces to detach**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Faces to detach** selections.

The Detach Faces tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

The **Parent heal method** list determines how to replace the detached faces in the parent object: **Create capping faces** means that a new faces are constructed based on the edges adjacent to each wound, and **Extend adjacent faces** (default) means that the wound is covered by growing and shrinking the adjacent faces.

The **Child heal method** list controls how to construct the child solid from the detached faces: **Create capping faces** means that a new face is formed based on the surrounding edges of each wound, **Extend adjacent faces from child** means that the detached faces grow or shrink to form a solid, and **Extend adjacent faces from parent** (default) means that the parent faces surrounding the detached faces grow or shrink to form a solid together with the detached faces.

When you click the **Detach Selected** button, the program detaches the selected faces and adds a node corresponding to this operation to the geometry branch of the model tree. The Tools window for Detach Faces remains open so that you can continue defeaturing using this or another defeaturing tool.

If you want to modify the performed detach operation, select the added node in the geometry branch. Then edit the node's form that appears in the Settings window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Detach Faces contains the additional settings described below.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Detect Interferences

To analyze the interference of geometry objects, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Detect Interferences** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Detect Interferences**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

From the **Input objects** list, choose **Manual** (default) to select the objects you want to analyze in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the currently built node, these will also be available in the **Input objects** list.

In the **Tolerance** text field, enter the absolute tolerance, which has the default value of 0.1 mm, to be used for the search for intersections. In the **Gap tolerance** text field enter the absolute tolerance for detecting gaps, the default value is 1 mm. When you click the **Find Interferences** button, a list of interferences is displayed in the **Interfering faces** list.

The interference detection applies to the exterior faces of the selected objects, thus ignoring interior faces, isolated edges and vertices. Each detected interference involves two objects. The following types of interferences appear in the list when detected by the tool:

- *Touch*. Two interfering objects are classified as touching when they intersect, and the interfering faces are located within a distance less than the specified tolerance from each other. Thus, an intersection (as defined below) may become a touch after you increase the tolerance such that it becomes larger than the distance between the interfering faces.
- *Intersection*. An intersection between the two objects is detected, and the interfering faces are located at a distance that is larger than the tolerance from each other.
- *Gap*. No intersection is detected between the two objects, but they have faces with portions that are located within a distance less than the specified gap tolerance from each other. The detected size of the gap appears in the list.
- *Containment*. One object is a contained in another object, which is a solid.

By default the **Interfering faces** list displays the detected interferences in a tree with the interferences as the top level nodes sorted by the type of interference. To sort the list by the objects select the **Group by object** checkbox. In this case the objects involved in an interference are listed as the top level nodes in the tree. To filter the list by the interference type select or clear the checkboxes **Show intersections**, **Show touches**, **Show gaps**, **Show containments**.

In the **Interfering faces** list, expand a top level node to see the list of objects that the object on the top level is interfering with. Expand a subnode to see the list of detected interferences for the objects. You can expand the nodes for the detected interferences to reveal the two interfering objects, and then expand the nodes for the objects to look at the interfering faces displayed in subnodes according to the following:

- For two intersecting solid objects the nodes for the objects have two subnodes each, *Inside* and *Outside*. Click the Inside node to highlight the parts of the interfering faces that are inside the other object. Click the Outside node to highlight the parts of the interfering faces that are outside the other object.
- For two intersecting surface objects the nodes for the objects have two subnodes each, *Small side* and *Large side*. The faces resulting from the intersection are grouped according to size since surface objects do not delimit a volume in space, thus it is not possible to determine what is inside or outside the other object. Click the Small side and Large side nodes to highlight the parts of the interfering faces that result from the intersection with the faces of the other object.
- For an intersection of a solid and a surface object, the node for the solid object has the subnodes *Small side* and *Large side*, while the node for the surface object has subnodes *Inside* and *Outside*.
- For two touching objects, the nodes for the objects may have two subnodes, *Touching* and *Not touching*. Click these nodes to highlight the corresponding portions of the interfering faces.
- For a gap between two objects, the nodes for the objects may sometimes have two subnodes *Touching* and *Not touching*.
- For a Containment node, the first subnode is the containing object, and the second subnode is the contained object.

VISUALIZATION OF DETECTED INTERFERENCES

For a better visualization of the detected interferences, the Detect Interferences tool partitions the faces involved in the intersections and touches so that the interfering face regions can be shown and highlighted separately. In the **Interfering faces** list, when you select the nodes for the interferences, or the topmost or first sublevel object nodes when the **Group by object** checkbox is selected, the interferences are visualized only by highlighting those faces or face regions that enclose an intersection or are directly in touch. In the list, these are the faces belonging to the first subnode of each object node under the selected interference node. Thus, when you click a node for an interference in the list, the faces or face regions belonging to nodes Outside, Large side, and Not touching are not highlighted. Expand the node for the interference, and select the object subnodes, to see highlighted the involved faces in their entirety, including the face regions that are not directly interfering.

Note that the face partitioning that is the result of the intersections is visible only while working with the Detect Interferences tool, and that the geometry is not modified by this tool.

To change which objects are shown in the **Graphics** window while selecting nodes in the **Interfering faces** list choose one of the options from the **Show in graphics** list:

- Choose **Interfering faces only** to show only the interfering faces involved in the selected node. For example, if you select an intersection node from the list, only the face regions from the two objects that are involved in the intersection are shown.
- Choose **Selected object** (default) to show only the objects involved in the selected node.
- Choose **Other object** to show the object that is interfering with the currently selected object subnode to the interference node.
- Choose **Both objects** to show both objects involved in an interference when you select one of the subnodes to the interference node.
- Choose **All objects** to show all objects regardless of which nodes are selected.

The **Zoom to Selection** button next to the **Interfering faces** list may also help to find the detected interferences on the geometry. For a better view of the interferences between objects you can also click the **Wireframe Rendering** or **Transparency** buttons in the Graphics toolbar.

Repair

To repair objects, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Repair** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

REPAIR

From the **Input objects** list, choose **Manual** (default) to select the objects you want to repair in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Repair** node, these will also be available in the **Input objects** list.

Enter the **Absolute repair tolerance** that is used by some of the repair options as detailed below.

With the **Fix errors** checkbox selected (default) the software tries to repair the following geometric and topological errors:

- Entities with invalid sense
- Invalid edge and vertex tolerances
- Invalid manifolds
- Self-intersecting manifolds
- Non-G1 manifolds
- Missing edge or vertex manifolds
- Missing vertex
- Vertices not on curve of edge
- Edges and vertices not on surface of a face
- Surface self-intersections that lie outside the face
- Edge intersections that have no vertex
- Removal of discontinuities by either splitting or smoothing.

Select the **Repair face-to-face inconsistencies in solids** checkbox to try to repair solid objects with this fault reported by Import, Check, or Repair features.

Select the option **Simplify curves and surfaces** to also simplify within the **Absolute repair tolerance** the underlying curve and surface manifolds of the geometric entities. Repairing objects with this option may improve both the performance and reliability of geometric operations on some imported geometry, for example it may help in some cases when Boolean operations on the imported objects fail. Simplification means that the manifolds are converted where possible to analytical form: linear, circular, and elliptical curves; and planar, spherical, cylindrical, conical, and toroidal surfaces. Manifolds that are converted are B-spline curves and surfaces, or certain surfaces generated by operations such as sweeping, revolving, and filleting.

The **Delete small details** checkbox is selected by default to remove geometric details smaller than the **Absolute repair tolerance**. Details that can be deleted include short edges, sliver faces, small faces, and spikes.

With the **Heal edges** checkbox selected the operation tries to replace edges with a large tolerance with accurate edges calculated by intersecting the adjacent surfaces. This also improves the adjacent vertices, which become exact.

If the **Minimize tolerances** checkbox is selected any tolerant edges and vertices on the geometry are detected, and if possible their tolerances are reduced.

When the option **Check resulting objects for errors** is selected (default) the repaired objects are checked for remaining problems. Warning nodes appear with details about the detected problems, if any. Use the **Zoom to Selection** button next to the **Entities** list in a warning node to locate the problematic edges or faces. For information on geometry problems see the Check feature.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual.*

Replace Faces

By replacing faces from an object you can delete the geometric features formed by the faces. The deleted faces are replaced either by new faces created based on the edges surrounding the wound or by growing or shrinking of adjacent faces.

To open the **Tools** window for **Replace Faces**, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Replace Faces** ( ). You can also right-click the **Geometry** node and select the same option from the context menu.

Note: When you are in the **Tools** window for **Replace Faces**, you can at any time switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the window.

Select the faces you want to replace in the **Graphics** window. They appear in the **Faces to replace** list. If the geometry sequence includes user-defined selections above the currently built node, choose **Manual** to select faces, or choose one of the selection nodes from the list next to **Faces to replace**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Faces to replace** selections.

The Replace Faces tool can only be applied to objects that are represented by the Parasolid® geometry kernel, also called CAD objects.

In the **Heal method** list, select the method to use for covering the wounds after deleting the faces to be replaced: **Create capping faces** means that new faces are generated based on the edges surrounding each wound, while **Extend adjacent faces** means that the adjacent faces are grown or shrunk to heal the wounds. Select the **Heal as through hole** checkbox if you have selected faces that make up a hole that you want to delete and replace.

When you click the **Replace Selected** button, the program deletes and replaces the selected faces and adds a node corresponding to this operation to the geometry branch of the model tree. The Tools window for Replace Faces remains open so that you can continue defeaturing using this or another defeaturing tool.

If you want to modify the performed replace operation, select the added node in the geometry branch. Then edit the node's form that appears in the Settings window. Click the **Build Selected** button ( ) to see the result of your edits. The Settings window for Replace Faces contains the additional settings described below.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Creating and Modifying Geometries

This section describes the operations for creating and modifying geometries listed in the table below.

TABLE 2-6: OPERATIONS FOR CREATING AND MODIFYING.

ICON	NAME	DESCRIPTION
	Cap Faces	Generate faces from edges to fill gaps and create solid objects, or to partition solids
	Chamfer	Create a bevel on selected edges
	Fillet	Create rounds on selected edges
	Knit to Solid	Knit surface objects to form solid or surface object
	Loft	Create a lofted surface from a set of profile curves
	Midsurface	Generate midsurfaces for selected solid objects
	Offset Faces	Offset faces of 3D objects in the normal direction
	Projection	Project 3D objects and entities to a work plane
	Project to Faces	Project edges to faces in 3D
	Thicken	Create a solid by offsetting selected surfaces
	Transform Faces	Displace, rotate, and scale faces of 3D objects

Cap Faces

You can add cap faces to fill holes in a geometry (for example, to make a domain for the void inside a cylinder geometry for simulating fluid flow inside the cylinder, or to create a domain inside a surface object with a hole) or to partition the geometry. To add cap faces to objects, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Cap Faces** ( ).

CAP FACES

Select edges that form loops around the faces you want to create. The edges then appear in the **Bounding edges** list. If the geometry sequence includes user-defined selections above the **Cap Faces** node, choose **Manual** to select edges, or choose one of the selection nodes from the list next to **Bounding edges**.

Click the **Activate Selection** button to toggle between turning ON  and OFF  the **Bounding edges** selections.

To automatically extend the selection to all adjacent edges that form a loop or chain, select the **Group adjacent edges** checkbox.

A cap face is created for each loop of edges in the input selection. The cap faces are joined with the original objects. If new closed volumes are created by the cap faces, these are converted to solid domains. The selected edges can contain more than one edge loop, but no two loops can have edges or vertices in common. The selected edges can contain edges from more than one object. In this case, each object is processed individually. This means that two edges or vertices can overlap as long as they are not in the same object. It also means that if new closed volumes are created, but bounded by faces from more than one object, these volumes are not converted to solid domains. If you want to perform a **Cap Faces** operation involving more than one object, first unite the objects using a **Union** operation.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

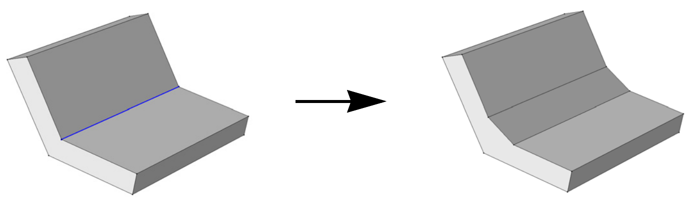
Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Chamfer

To chamfer corners in 3D geometry objects, from the **Geometry** toolbar, **Editing** ( ) menu, select **Chamfer** ( ). You can also right-click the **Geometry** node to add this node from the context menu.



EDGES

Select the edges that you want to chamfer in the **Graphics** window. They then appear in the **Edges to chamfer** list. If the geometry sequence includes user-defined selections above the **Chamfer** node, choose **Manual** to select edges, or choose one of the selection nodes from the list next to **Edges to chamfer**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Edges to chamfer** selections.

To automatically extend the selection to tangent edges select the **Group by continuous tangent** checkbox (cleared by default). Modify the **Angular tolerance** to control which edges are added to the selection. Values between 0 and 180 degrees are supported (default: 5 degrees).

RADIUS

Enter the **Radius** of the chamfer. The size of the chamfer is determined by rolling a ball of the given radius so that it is tangent to the faces that are adjacent to the edge. The chamfer surface is generated by the line segment that connects the points of tangency.

OPTIONS

Select or clear the following checkboxes as needed.

- If the **Propagate to tangent edges** checkbox is selected, the chamfer is propagated to edges that have continuous tangent to the edges selected in **Edges to chamfer**.
- If the **Preserve overlapped entities** checkbox is selected, geometric features such as holes and bosses on faces that are overlapped by the chamfer surface are preserved.
- Select the **Y-shaped chamfer** checkbox to get a *y*-shaped chamfer at a vertex where three or more edges meet and there are two chamfer surfaces of different convexity. In some cases, using this option is necessary for the operation to succeed.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see in the *COMSOL Multiphysics Reference Manual.*

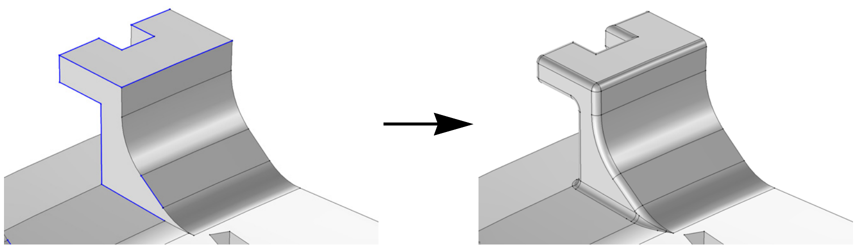
Fillet

To fillet corners in 3D geometry objects, from the **Geometry** toolbar, **Editing** ( ) menu, select **Fillet** ( ). You can also right-click the **Geometry** node and add this node from the context menu.

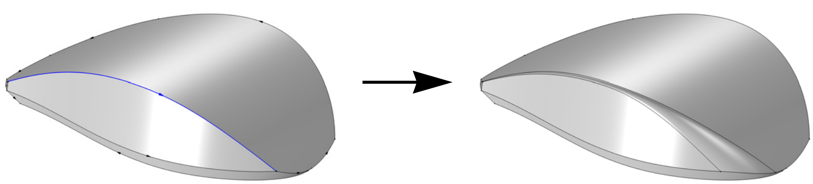
TYPE OF FILLET

Select a **Type** for the fillet — **Constant radius** (the default), **Constant width**, or **Variable radius**.

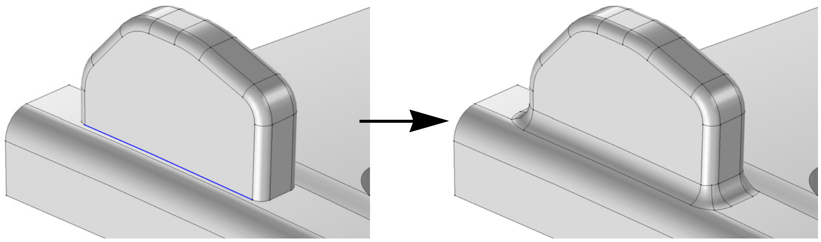
- For **Constant radius**, the fillet surface is generated by rolling a ball of the given radius so that it is tangent to the faces adjacent to the edge. When you build the feature, the faces adjacent to the selected edges are shrunk and a fillet face is inserted in between.



- For **Constant width**, the fillet face is generated in a way that the distance between the two edges that separate the fillet face from the adjacent faces is constant.
- For **Variable radius**, you can specify the radius at chosen locations along the selected edges. The fillet radius is then interpolated between the given radius values.



For all fillet types, when more than two selected edges meet at a vertex, one or several additional patch faces are inserted at the vertex to get a smooth result. If the radius is large, it can happen that the fillet face overflows the original faces. In this case the fillet face meets other, more distant, faces in the object.



EDGES

Select the edges that you want to fillet in the **Graphics** window. They then appear in the **Edges to fillet** list. If the geometry sequence includes user-defined selections above the **Fillet** node, choose **Manual** to select edges, or choose one of the selection nodes from the list next to **Edges to fillet**.

Click the **Activate Selection** button to toggle between turning ON  and OFF  the **Edges to fillet** selections.

To automatically extend the selection to tangent edges select the **Group by continuous tangent** checkbox (cleared by default). Modify the **Angular tolerance** to control which edges are added to the selection. Values between 0 and 180 degrees are supported (default: 5 degrees).

The objects containing the selected edges must have manifold topology in the neighborhood of the selected edges. In particular, each edge must be adjacent to exactly two faces. For **Variable radius**, you can only select edges from a single geometry object.

RADIUS

This section is displayed when **Type** is set to **Constant radius**. Enter the **Radius** of the fillet (SI unit: m). The fillet surface is generated by rolling a ball of the given radius so that it is tangent to the faces adjacent to the edge.

WIDTH

This section is displayed when **Type** is set to **Constant width**. Enter the **Width** of the fillet (SI unit: m). The width of the fillet is the distance between the two edges that separate the fillet face from the adjacent faces.

RADII

This section is displayed when **Type** is set to **Variable radius**. Use the table to specify the radii at locations given by parameter values along the selected edges.

- The **Edge** column displays the edge numbers for the selected edges. When you select a row in the table the corresponding edge is highlighted in the **Graphics** window.
- In the **Parameter** column enter the relative arc length parameter where the radius is specified along an edge. The parameter can have values between 0 (at the start vertex of the edge) and 1 (at the end vertex of the edge). See the direction arrows displayed in the **Graphics** window for determining the edge orientations.
- In the **Radius** column enter the value for the radius to be applied at the location along the edge specified by the parameter. The radius value can be a positive number or 0. An empty cell (the default) means that the radius is not defined, and that the row for the corresponding location is ignored when creating the fillet.
- Select the checkbox (cleared by default) in the **Clamp** column to constrain to zero the derivative of the fillet radius with respect to the relative arc length parameter at this point along the edge.

Below the table, click the **Insert Above** ( ) or **Insert Below** ( ) button to add rows to the table for specifying radius values at additional locations along an edge. Select a row and click **Delete** ( ) to remove it from the table. Only rows that do not correspond to the first and last points of an open ended chain are possible to delete. You can also leave a radius field empty to ignore a row when creating the fillet.

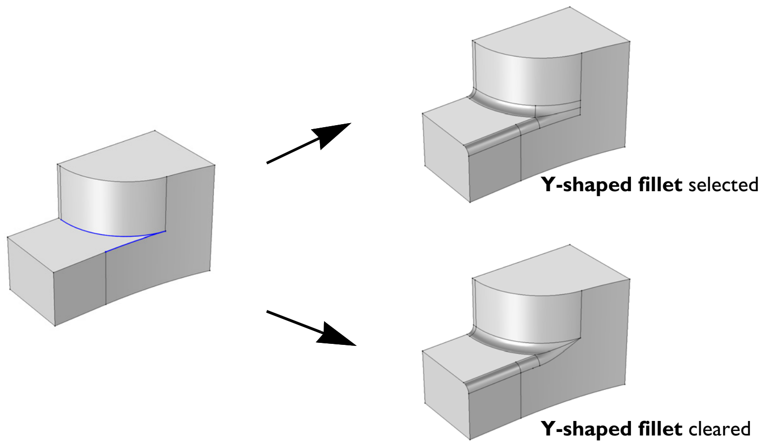
Click **List All Fillet Edges** ( ) to insert one row for each fillet edge that is missing from the table, and to sort the table. The **Delete Rows Without Radius** ( ) button is useful to compact the table in case there are many rows with empty radius values. Rows that correspond to the first and last points of an open edge chain are not deleted since you need to specify a radius for these. For a better overview of the specified radius values click the **Sort Edge Chains** ( ) button. The table is then sorted primarily on the edge chains, secondly on the natural order of the edges, and thirdly on the parameters in the chain's direction.

OPTIONS

Select or clear the following checkboxes as needed.

- If the **Propagate to tangent edges** checkbox is selected, the fillet is propagated to edges that have continuous tangent to the edges selected in **Edges to fillet**. This checkbox is available only when **Type** is **Constant radius** or **Constant width**.

- If the **Preserve overlapped entities** checkbox is selected, geometric features such as holes and bosses on faces that are overlapped by the fillet surface are preserved.
- Select the **Y-shaped fillet** checkbox to get a *y*-shaped fillet at a vertex where three or more edges meet and there are two fillet surfaces of different convexity. This option is not available when the fillet **Type** is **Constant width**. In some cases, using this option is necessary for the operation to succeed.



- Select the **Fillet sharp edges at vertices** checkbox to get a smooth fillet surface at vertices where two filleted edges intersect at an angle. This option is not available when the fillet **Type** is **Constant width**.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Knit to Solid

To knit surface objects to form solid objects, from the **Geometry** toolbar, **Defeaturing and Repair** ( ) menu, select **Knit to Solid** ( ).

KNIT TO SOLID

Select the objects to knit together in the **Graphics** window. They appear in the **Input objects** list.

From the **Input objects** list, choose **Manual** (default) to select the objects you want to knit together in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Knit to Solid** node, these will also be available in the **Input objects** list.

The knitting merges edges that have a distance smaller than the **Absolute repair tolerance** and deletes gaps and spikes smaller than the **Absolute repair tolerance**. If the **Fill holes** checkbox is selected the operation attempts to generate new faces to replace missing geometry.

The input surface objects must have manifold topology, and the operation can only form solids with manifold topology. An example of a solid object with nonmanifold topology is a solid that has an interior surface that separates two domains. A surface object that contains an edge that is adjacent to more than one boundary is an example of a surface object with nonmanifold topology.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

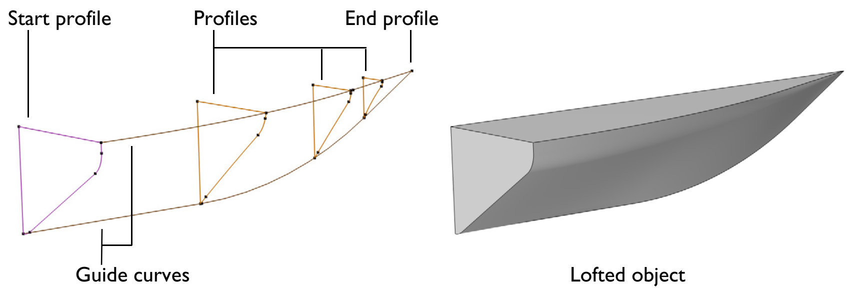
Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Loft

To create a lofted object from a set of profiles in 3D, in the **Geometry** toolbar, click **Loft** (). You can also right-click the **Geometry** node and add this node from the context menu. Enter the properties of the loft operation according to the following sections.



Each profile is a chain or loop of edges, also called a profile curve. The profiles must be all open or all closed, and they must have the same number of edges. The output is a loft surface, which consists of one or several faces that interpolate the profiles. In the closed curve case, a profile can optionally contain a set of faces (with manifold topology) that is bounded by the profile curve. These faces can be added to the loft surface to give the resulting object. The start and end profiles can degenerate to a point. Also, in the closed profile case, the start and end profiles can degenerate to an open curve. The loft can be periodic, which means that the end profile should not be specified because it equals the start profile (in this case the degenerate profiles are not allowed).

There can also be curves in the lofting direction that the loft surface should interpolate; these are called guide curves. If there are no guide curves, there must be at least two profiles.

GENERAL

Select or clear the following checkboxes as needed.

- Select the **Periodic loft** checkbox to create a periodic loft, for which the start and end profiles coincide.
- If the **Unite with input objects** checkbox is selected, the resulting object is the union of the loft surface with the objects containing the start and end profiles and the objects containing the start and end guide curves. The faces that might exist in the start and end profiles are always included in the resulting object.

- Select **Remove redundant profile vertices** to remove vertices that separate edges with identical curves before generating the loft. This option is useful when the outputs of Interpolation Curve features is used as profiles for the loft.
- If the **Keep intermediate profile faces** checkbox is selected, faces in the intermediate profile objects are added to the resulting object. Any faces belonging to the start and end profiles are always kept.
- Select an **Object type** — **Solid** (the default) or **Surface**. This determines whether domains should be created in the resulting object.
- Select a **Face partitioning**:
  - If **Minimal** (the default) is selected, the loft surface is divided along the loft direction only at vertices where the profile curve has a tangent discontinuity.
  - If **Columns** is selected, the loft surface is divided along the loft direction at each vertex of the profile curves.
  - If **Grid** is selected, in addition to the Columns partitioning, the loft surface is divided by the profile curves. The loft surface is always partitioned by the profile faces when **Keep intermediate profile faces** is selected.

PROFILES

This section specifies the profiles that are not specified in the **Start Profile** or **End Profile** sections.

From the **Profile objects** list, choose **Manual** (default) to select the objects you want to use as profiles in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Profile objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Loft** node, these will also be available in the **Profile objects** list.

Note: You can select a set of connected surface objects, curve objects, or point objects. Surface objects must have manifold topology and be bounded by a single edge loop. Curve objects must be a single edge loop or chain. Point objects are only allowed for use as start or end profiles and must have a single vertex.

START PROFILE

Use this section to specify the start profile in the following cases:

- If you want to explicitly specify which profile should be the start profile.
- If the start profile is part of a larger object.
- If you want to prescribe the direction of the loft surface on the start profile.

In other cases, you can specify the start profile in the Profiles section, and leave the selection in the Start Profile section empty.

Select a **Geometric entity level** for the profile — **Object**, **Point**, **Edge**, or **Boundary**. Click to select the entities in the **Graphics** window. An object selection must fulfill the requirements detailed in the **Profiles** section. A point selection must consist of a single point. An edge selection must form a single edge loop or chain. A boundary selection must have manifold topology and be bounded by a single edge loop. The selected entities appear in the **Start profile** list. For **Object**, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Loft** node, choose **Manual** to select objects or entities, or choose one of the selection nodes from the list next to **Start profile**.

Click the **Activate Selection** button to toggle between turning ON  and OFF  the **Start profile** selection.

Select a **Loft direction** — **Not prescribed** (the default), **Parallel**, **Perpendicular**, or **At angle**. For **Parallel** the loft direction is prescribed along the profile curve, while for **Perpendicular** or **At angle** it is only prescribed at the vertices on the profile curve.

Select **Relative to** — **Adjacent faces** (the default), **Profile faces**, or **Profile edges' plane**. When **At angle** is selected, also enter an **Angle** (SI unit: deg).

- Adjacent faces are the faces that are adjacent to the profile edges and that are not contained in the **Start profile** selection.
- Profile faces are the faces contained in the **Start profile** selection.
- Profile edges' plane means that the loft direction is prescribed in relation to the plane tangent to the profile's edges at each vertex on the profile curve.

END PROFILE

The settings for this section are analogous to the Start Profile section. This section should not be used if the loft is periodic.

GUIDE CURVES

Use this section if you want to specify guide curves for the lofted object.

From the **Guide objects** list, choose **Manual** (default) to select the objects you want to use as guides in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Guide objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Loft** node, these will also be available in the **Guide objects** list.

Note: You can select a set of curve objects. In the nonperiodic case, each guide object must be a single edge chain. In the periodic case, each guide object must be a single edge loop. Each guide object must have continuous tangents and intersect each profile exactly once.

START GUIDE CURVES

Use this section to specify the start guide curve in the following cases:

- If the start guide curve is part of a larger object.
- If you want to prescribe the direction of the loft surface on the start guide curve.

In other cases, you can specify the start guide curve in the Guide Curves section, and leave the selection in the Start Guide Curve section empty.

The selected set of edges must form a single edge loop or chain, and fulfill the other requirements on a guide curve. Click the **Activate Selection** button to toggle between turning ON and OFF the **Edges** selection.

Select a **Loft surface direction** — **Not prescribed** (the default) or **Parallel to adjacent faces**, which means that the loft surface is prescribed to be tangent to the adjacent faces along the guide curve.

END GUIDE CURVES

The settings for this section are analogous to the Start Guide Curve. This section should not be used in the closed profile case.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS ON INPUT OBJECTS

If you have Named Selections that include entities on the input objects, select the **Propagate selections to resulting objects** (selected by default) checkbox to update the selections to corresponding entities on the output objects, when possible. Clear the checkbox to not propagate the selection to the resulting objects. Selecting this option can be useful in combination with clearing the **Unite with input objects** checkbox so that the selections refer only to the input objects.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Midsurface

The Midsurface feature is the inverse of the Thicken feature (with symmetric offset). It removes the thickness of a solid object (having constant thickness), resulting in a surface object, which can be useful if you can use a Shell interface, for example, and model the physics on surfaces only.

In the **Geometry** toolbar, **Conversions** menu, click **Midsurface** ( ). You can also right-click the **Geometry** node and add this node from the **Conversions** submenu.

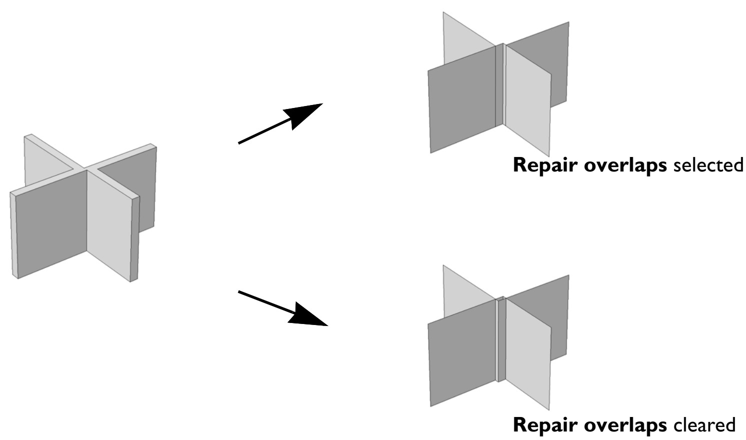
INPUT

From the **Input objects** list, choose **Manual** (default) to select the objects you want to use as input in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Midsurface** node, these will also be available in the **Input objects** list. A midsurface object is generated for each input object independently.

Select the **Keep input objects** checkbox to use the selected geometry objects for further geometry operations.

OPTIONS

Select the **Repair overlaps** checkbox to repair areas where two or more generated midsurfaces overlap.



Click to select the **Split in smooth components** checkbox as needed. If this is selected, each output object is split into components, where each component is of manifold type and has smooth normal vector.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS ON INPUT OBJECTS

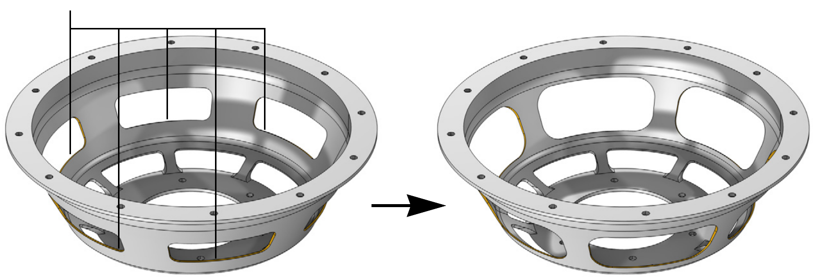
If you have Named Selections that include entities on the input objects, select the **Propagate selections to resulting objects** (selected by default) checkbox to update the selections to corresponding entities on the output objects, when possible. Clear the checkbox to not propagate the selection to the resulting objects. Selecting this option can be useful in combination with selecting the **Keep input objects** checkbox so that the selections refer only to the input objects.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Offset Faces

To offset the faces of 3D geometry objects in the normal direction, from the **Geometry** toolbar, **Editing** ( ) menu, select **Offset Faces**( ). You can also right-click the **Geometry** node and add this node from the context menu.



Enter the properties of the Offset Faces operation using the following sections:

FACES

Select the faces that you want to offset in the **Graphics** window. They then appear in the **Faces to offset** list. If the geometry sequence includes user-defined selections above the **Offset Faces** node, choose **Manual** to select faces, or choose one of the selection nodes from the list next to **Faces to offset**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Faces to offset** selections.

Select the **Keep input objects** checkbox if you want to use the selected geometry objects for further geometry operations.

When the **Subtract input objects** checkbox is selected, after the faces are offset, the corresponding input objects are subtracted from the offset objects.

OFFSET

Enter a **Distance** for the offset. Switching between a positive and negative number reverses the offset direction.

Select the **Reverse side** checkbox (cleared by default) to reverse the offset direction. This is equivalent to changing the sign of the offset distance.

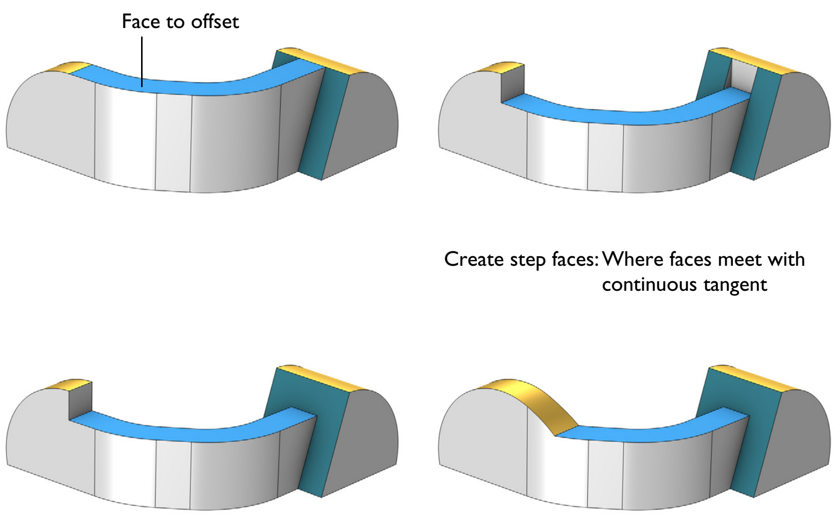
OPTIONS

Select the checkbox **Fillet convex edges**, cleared by default, to fillet convex edges between two offset faces. By default, such offset faces are instead extended until they meet.

The checkbox **Perpendicular step edges for surface objects**, cleared by default, applies to offsetting faces of a surface object, when the operation sometimes needs to split a vertex in two and introduce a step edge in between them. By default, the step edge is not perpendicular to an adjacent edge. With the checkbox selected, the created step edge is perpendicular to an adjacent edge.

To decide how step faces that connect the bounding edges of the offset faces with the adjacent faces are created, select one of the available options for **Create step faces**:

- Select **Where faces meet with continuous tangent** (default) to create step faces at all bounding edges where the selected faces meet the adjacent faces with tangent continuity.
- Select **No** to not create step faces.



Create step faces: Where faces meet with continuous tangent

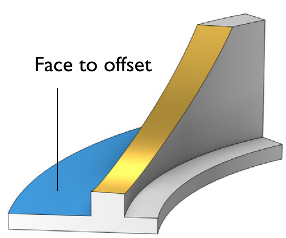
Create step faces: Where faces meet with continuous tangent
Except if faces at ends of edge can be merged

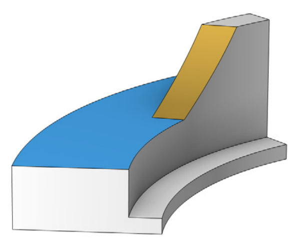
Create step faces: No

When **Create step faces** is set to **Where faces meet with continuous tangent** you can select the checkbox **Except if faces at ends of edge can be merged** (cleared by default) to extend and merge, if possible, the neighboring faces at the two ends of those bounding edges where faces meet with continuous tangent.

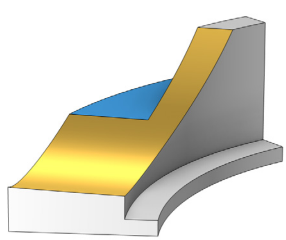
Use the **Overflow handling** setting to determine how to handle the case when an offset face moves beyond a nearby face in such a way that a wound (hole) appears. The following options are available:

- **Automatic** (default). The operation determines a suitable method to cover the wound caused by the overflow.
- **Extend offset faces**. With this option the offset faces are extended to cover the wound caused by the overflow.
- **Extend other faces**. With this option the faces that are not offset are extended to cover the wound caused by the overflow.
- **Cap faces**. The wound caused by the overflow is covered by creating new faces based on the edges surrounding the wound.
- **Disallow**. This option will return an error message if overflow happens.

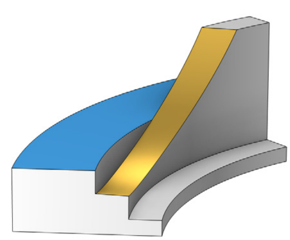




Overflow handling: Extend offset faces



Overflow handling: Extend other faces



Overflow handling: Cap faces

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS ON INPUT OBJECTS

If you have Named Selections that include entities on the input objects, select the **Propagate selections to resulting objects** (selected by default) checkbox to update the selections to corresponding entities on the output objects, when possible. Clear the checkbox to not propagate the selection to the resulting objects. Selecting this option can be useful in combination with selecting the **Keep input objects** checkbox so that the selections refer only to the input objects.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Projection

Use the **Projection** feature to compute the projection of 3D objects and entities to a work plane. This can be useful when you need to reference existing 3D objects in the 2D drawing on the work plane. You can also add a new 2D or 2D axisymmetric component and add the **Projection** node there. In that case you can select the work plane to use for the projection from the 3D component's geometry sequence, but first make sure that in the 3D component's **Geometry** node the **Geometry representation** is set to the **CAD kernel**.

To add a projection to a **Work Plane** node's **Plane Geometry** sequence, from the **Plane Geometry** toolbar select **Projection** ( ), or right-click a **Plane Geometry** node under a **Work Plane** node and select **Projection** ( ). Enter the properties of the projection using the following sections:

PROJECTION

From the **Project** list, choose **All objects** (the default) to project all 3D geometry objects to the work plane, or choose **Selected objects**, **Selected domains**, **Selected boundaries**, **Selected edges**, or **Selected vertices** to project only the objects or entities that you add to the **Entities to project** list that appears. For **Selected Objects**, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Work Plane** node, choose **Manual** to select objects or entities, or choose one of the selection nodes from the list next to **Entities to project**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Entities to project** selections.

When projecting objects, domains, and boundaries you can select the **Projection type** to project the **Outline** (default) of the selected objects and entities, or to project the **Edges and vertices** only, or the **Outline, edges, and vertices**.

Projecting the outline for surface and solid objects results in the edges that form the boundary of the shadow of the object's faces. In this case the edges are projected only if they coincide with the outline. If you project the outline of a mixed object that contains isolated edges and vertices, only the outline of the faces is generated, the isolated edges and vertices are not projected.

For point and curve objects, projecting the outline is the same as projecting the edges and vertices.

You can change the settings for the **Repair tolerance** list if you experience problems with the projection operation. Geometric entities that have a distance less than the repair tolerance are merged.

- The default value in the **Repair tolerance** list is **Automatic**, which means a relative repair tolerance of 10^{-6}.
- Choose **Relative** to enter a value for the **Relative repair tolerance** field (the default is determined by the main **Geometry** node's setting). This value is relative to the largest absolute value of the coordinates of all input objects.
- Choose **Absolute** to enter a value for the **Absolute repair tolerance** field (the default is determined by the main **Geometry** node's setting; SI unit: m). This value uses the same unit as the geometry sequence's length unit.

When you build this feature, the relative and absolute repair tolerances are set to the values that are used for the last projected object (with a precision of two digits). This is useful to find out the tolerance used for the last projected object. After the feature is built, you can set the **Repair tolerance** to either **Relative** or **Absolute**, then check the values displayed in the **Relative repair tolerance** or **Absolute repair tolerance** fields.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, boundaries, and points — that are applicable) in subsequent nodes in the plane geometry sequence. To also make all or one of the types of resulting entities (objects, boundaries, and points) available as selections in applicable selection lists in the main **Geometry** node's geometry sequence, choose an option from the **Show in 3D** list: **All levels**, **Object selection** (default), **Boundary selection**, or **Point selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the plane geometry sequence.

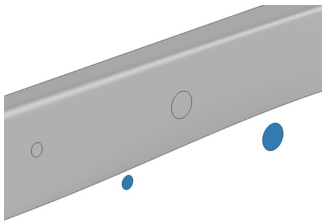
ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects.

For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Project to Faces

Use the **Project to Faces** ( ) feature to create imprints on faces by projecting edges onto the face.



To add the feature to a geometry sequence, in the **Geometry** toolbar, **Booleans and Partitions** menu, click **Project to Faces**. You can also right-click the **Geometry** node and add this node from the **Booleans and Partitions** submenu.

TARGET

Select the faces that you want to project to in the **Graphics** window. The faces appear in the **Faces to project to** list. If the geometry sequence includes user-defined selections above the **Project to Faces** node, choose **Manual** to select faces, or choose one of the selection nodes from the list. Click the **Activate Selection** button to toggle between turning ON and OFF the **Faces to project to** selections.

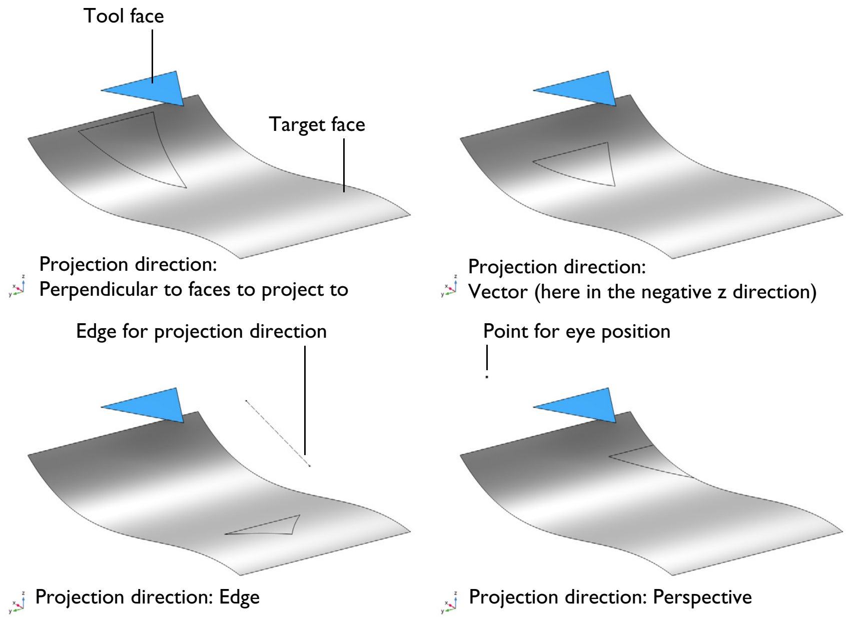
Select the **Imprint** checkbox (default) to create imprints and partition the target faces. Clear this checkbox to keep the target faces unmodified, and to return the projected edges/vertices as separate objects with one output object for each object in the **Faces to project to** selection. This checkbox is disabled and imprints are always created on the target faces if the Project to Faces feature is added after a Form Union/Form Assembly node in the geometry sequence.

TOOL

From the **Geometric entity level** list, select **Edge** (the default) to project selected edges, or select **Boundary** to project the edges of selected faces. In both cases, choose the tool entities and add them to the **Entities to project** list below.

OPTIONS

Select a **Projection direction** — **Perpendicular to faces to project to** (the default), **Vector**, **Edge**, or **Perspective**.



- For **Perpendicular to faces to project to**, note that the projection direction can vary along the target faces to match the face normal direction.
- For **Vector**, enter the vector components for the projection direction in the **x**, **y**, **z** fields.
- For **Edge**, choose a straight edge as the projection direction and add it to the **Straight edge** list.
- For **Perspective**, choose a point for the eye position and add it to the **Eye position** list. Note that projected entities are not created on target faces that are shadowed by faces on the object that the target faces belong to.

Enter a value or expression to specify the **Maximum distance** (SI unit: m; default `Inf`). Projected entities that are not entirely within this distance from the tool entities will be discarded from the result.

By changing the settings for the **Repair tolerance** list you can control the snapping of the projected entities to the existing edges and vertices on the target faces. Projected entities that are within a distance less than the repair tolerance from exiting edges or vertices on the target are snapped together.

- The default value in the **Repair tolerance** list is **Automatic**, which determines an appropriate repair tolerance internally.
- Choose **Relative** to enter a value for the **Relative repair tolerance** field (the default is determined by the main **Geometry** node's setting). This value is relative to the largest absolute value of the coordinates of all input objects.
- Choose **Absolute** to enter a value for the **Absolute repair tolerance** field (the default is determined by the main **Geometry** node's setting; SI unit: m). This value uses the same unit as the geometry sequence's length unit.

When you build this feature, the relative and absolute repair tolerances are set to the values that are used (with a precision of two digits), which can be useful when you have set **Repair tolerance** to **Automatic**.

Select the **Extend projected edges** checkbox (cleared by default) to make sure that all projected edges are connected to another edge at both ends.

SELECTIONS OF RESULTING ENTITIES

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Thicken

In the **Geometry** toolbar, **Conversions** menu, click **Thicken** ( ) to convert a surface object to a solid object by giving it a thickness in the normal direction. You can also right-click the **Geometry** node and add this node from the **Conversions** submenu.

INPUT

From the **Input objects** list, choose **Manual** (default) to select the objects you want to thicken in the **Graphics** window. Click the **Activate Selection** button to toggle between turning ON and OFF the **Input objects** selections. Alternatively, choose **All objects** to select all objects or choose **All nonconstruction objects** to automatically select all objects that have not been marked as Construction Geometry. If the geometry sequence includes user-defined selections above the **Thicken** node, these will also be available in the **Input objects** list. Each input object is thickened independently.

Select the **Keep input objects** checkbox to use the selected geometry objects for further geometry operations.

OPTIONS

Select an **Offset** — **Symmetric** (the default) or **Asymmetric**.

If **Symmetric** is selected enter a **Total thickness**.

If **Asymmetric** is selected enter an **Upside thickness** and a **Downside thickness**.

Select the **Fillet offset edges** checkbox to fillet each convex edge joining two offset faces, using the offset distance as the fillet radius. This option applies only when the offset is single sided; that is, when either the up or down thickness is set to 0.

For either choice, select the **Direction of side faces** — **Normal** (the default) or **Vector**. For **Vector**, and based on space dimension, enter values or expressions for **x**, **y**, and **z** (SI unit: m). Note that this setting influences only the direction of the side faces, the specified thickness is always applied in the normal direction.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS ON INPUT OBJECTS

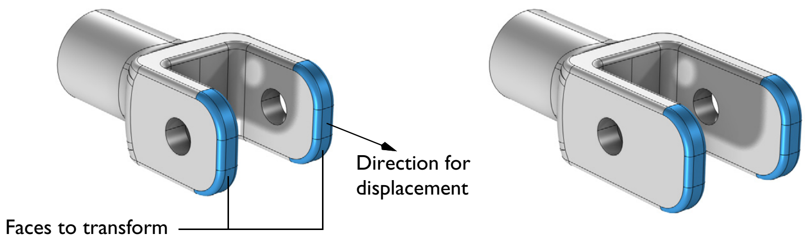
If you have Named Selections that include entities on the input objects, select the **Propagate selections to resulting objects** (selected by default) checkbox to update the selections to corresponding entities on the output objects, when possible. Clear the checkbox to not propagate the selection to the resulting objects. Selecting this option can be useful in combination with selecting the **Keep input objects** checkbox so that the selections refer only to the input objects.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

Transform Faces

Using the Transform Faces operation you can apply a linear transform (consisting of displacement, rotation, and isotropic scaling) on a selection of faces on 3D geometry objects. The transformed faces and their adjacent faces are extended or trimmed to cover any wounds that may result. Alternatively, the wounds are filled by inserting step faces.



To add a Transform Faces node to a geometry sequence, from the **Geometry** toolbar, **Editing** ( ) menu, select **Transform Faces**( ). You can also right-click the **Geometry** node and add this node from the context menu.

Enter the properties of the Transform Faces operation using the following sections:

FACES

Select the faces that you want to transform in the **Graphics** window. They then appear in the **Faces to transform** list. If the geometry sequence includes user-defined selections above the **Transform Faces** node, choose **Manual** to select faces, or choose one of the selection nodes from the list next to **Faces to transform**.

Click the **Activate Selection** button to toggle between turning ON and OFF the **Faces to offset** selections.

Select the **Keep input objects** checkbox if you want to use the selected geometry objects for further geometry operations.

COORDINATE SYSTEM

The coordinate system in which the point coordinates, displacements, and axis of rotation are interpreted for the linear transformation. From the **Work plane** list, select **xy-plane** (the default, for a standard global Cartesian coordinate system) or select any work plane defined above this node in the geometry sequence. If you choose a work plane, the work plane and its coordinate system appears in the **Graphics** window, using an extra coordinate triad with the directions **xw**, **yw**, and **zw** (which are then used to specify the coordinates, displacements, and the rotation axis position).

CENTER

Specify a point on the axis of rotation, and the centerpoint of the scaling by specifying **x**, **y**, and **z**; **xw**, **yw**, and **zw** if a work plane is selected as the coordinate system. This is the point that stays fixed during the scaling (that is, the point that the scaled faces approach when the scale factor goes to zero).

ROTATION

From the **Specify** list, choose **Axis of rotation** (the default), **Euler angles (Z-X-Z)**, or **Edge** as the way to specify the rotation.

- For **Axis of rotation**, select an **Axis type**: **x-axis**, **y-axis**, **z-axis** (the default), **Cartesian**, or **Spherical**. For any choice, enter an **Angle** (SI unit: degrees; default 0) to specify the rotation. If **Cartesian** is selected, enter Cartesian coordinates values for **x**, **y**, and **z** (default values 0, 0, and 1, respectively, corresponding to the global z-axis) to specify the axis vector. If **Spherical** is selected, specify the axis vector using spherical angles **theta** and **phi** in degrees (default: 90 and 0, respectively).
- For **Euler angles (Z-X-Z)**, enter values for the intrinsic Z-X-Z Euler angles α, β, and γ in the corresponding text fields (in degrees; the default values are 0).
- For **Edge**, choose a straight edge as the axis of rotation and add it to the **Straight edge** list. Then enter an **Angle** (SI unit: degrees; default 0) to specify the rotation angle.

SCALING

Specify the isotropic scaling factor in the **Factor** field.

DISPLACEMENT

Enter values or expressions to specify the **x**, **y**, and **z** displacements (SI unit: m).

OPTIONS

To decide how the step faces, which connect the bounding edges of the transformed faces with the adjacent faces, are created, select one of the available options for **Create step faces**:

- Select **Yes** to create step faces for all bounding edges of the transformed faces.
- Select **Where faces meet with continuous tangent** (default) to create step faces at all bounding edges where the selected faces meet the adjacent faces with tangent continuity.
- Select **No** to not create step faces.

When **Create step faces** is set to **Yes** you can select the checkbox **Merge step faces with adjacent face**, cleared by default, to extend and merge, if possible, the neighboring faces with the step faces.

When **Create step faces** is set to **Where faces meet with continuous tangent** you can select the checkbox **Except if faces at ends of edge can be merged**, cleared by default, to extend and merge, if possible, the neighboring faces at the two ends of those bounding edges where faces meet with continuous tangent.

Use the **Overflow handling** setting to determine how to handle the case when a transformed face moves beyond a nearby face in such a way that a wound (hole) appears. The following options are available:

- **Automatic** (default). The operation determines how to cover the wound caused by the overflow.
- **Extend transformed faces**. With this option the transformed faces are extended to cover the wound caused by the overflow.
- **Extend other faces**. With this option the faces that are not transformed are extended to cover the wound caused by the overflow.
- **Cap faces**. The wound caused by the overflow is covered by creating new faces based on the edges surrounding the wound.
- **Disallow**. This option will return an error message if overflow happens.

SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no contribution), or click the **New** button to create a new cumulative selection (see Cumulative Selections in the *COMSOL Multiphysics Reference Manual*).

Select the **Resulting objects selection** checkbox to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

SELECTIONS ON INPUT OBJECTS

If you have Named Selections that include entities on the input objects, select the **Propagate selections to resulting objects** (selected by default) checkbox to update the selections to corresponding entities on the output objects, when possible. Clear the checkbox to not propagate the selection to the resulting objects. Selecting this option can be useful in combination with selecting the **Keep input objects** checkbox so that the selections refer only to the input objects.

ASSIGNED ATTRIBUTES

From the **Construction geometry** list choose **On** to make the resulting objects available only in the feature's geometry sequence. The default option **Inherit from input** means that the resulting objects become construction geometry if all input objects are construction geometry. Choose **Off** to never output construction geometry objects. For more information see Construction Geometry in the *COMSOL Multiphysics Reference Manual*.

3

Programming and Command Reference

In this section you find detailed COMSOL® API reference information for the geometry features in the Design Module.

Defeaturing Tools

To remove unnecessary details in objects imported from a 3D CAD file, and detect interferences between objects, you can use the defeaturing tools. You access these by typing:

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets");
model.component(<ctag>).geom(<tag>).defeaturing("Holes");
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges");
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces");
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces");
model.component(<ctag>).geom(<tag>).defeaturing("Spikes");
model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces");
model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces");
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences");
```

Using the defeaturing tools you can search for small details, without altering your geometry. If you find small details that you want to remove, a defeaturing tool can create a feature that removes the details from the geometry.

The features corresponding to the defeaturing tools are `DeleteFillets`, `DeleteHoles`, `DeleteShortEdges`, `DeleteSliverFaces`, `DeleteSmallFaces`, `DeleteSpikes`, `ReplaceFaces`, and `DetachFaces`. If you already know which details you need to remove, it is also possible to create these features directly using the standard create syntax.

This section includes these topics:

- Defeaturing Tools — Finding and Deleting Small Details
- Defeaturing Tools — Detach Faces
- Defeaturing Tools — Detect Interferences
- Defeaturing Tools — Replace Faces

Defeaturing Tools — Finding and Deleting Small Details

The defeaturing tools `Fillets`, `Holes`, `ShortEdges`, `SliverFaces`, `SmallFaces`, and `Spikes` search for and delete details smaller than a given size. First select the objects you want to examine by typing, for example,

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      selection("input").set(<onames>);
```

where <onames> is a string array contains the object names.

Set the maximum size of the details (fillets in this case) you want to remove by typing

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      set("entsize",size);
```

The defeaturing tools `Fillets` and `Holes` also support specifying a minimum radius, to do this type (for fillets in this case):

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      set("minentsize",minsize);
```

To find the details in the specified size interval, type

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      find();
```

The found details appear in the selection

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      detail();
```

To get the number of found details, type

```
int nd = model.component(<ctag>).geom(<tag>).
         defeaturing("Fillets").detail().size();
```

To get the names of the found details, type

```
String[] filletNames = model.component(<ctag>).geom(<tag>).
         defeaturing("Fillets").detail().groupNames();
```

In general, a detail (fillet in this case) consists of a number of geometric entities. For example, a fillet consists of a number of faces. To get the entity numbers in the `n`th detail, type

```
int[] entities = model.component(<ctag>).geom(<tag>).
               defeaturing("Fillets").detail().groupEntities(n);
```

To get the object that contains the `n`th detail, type

```
String oname = model.component(<ctag>).geom(<tag>).
               defeaturing("Fillets").detail().groupObject(n);
```

To delete all details found, type

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      deleteAll(<ftag>);
```

This adds a feature, tagged <ftag>, that performs the deletion operation to the geometry sequence, after the current feature, and build this feature. In this case, it adds a `DeleteFillets` feature.

To delete a subset of the details found, type, for example

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      detail().setGroup(2,5);
```

to delete fillets number 2 and 5. You can also use, for example,

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      detail().addGroup(7,8);
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      detail().removeGroup(3);
```

to add and remove details from the selection. Perform the deletion by typing

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      delete(<ftag>);
```

This adds a `DeleteFillets` feature tagged *<ftag>* after the current feature in the geometry sequence.

DEFEATURING METHODS

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).find()` searches for small details, for a defeaturing feature *<ftag>*.

`model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing(`*tooltag*`).find()` searches for small details, for a defeaturing tool *tooltag*.

`model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing(`*tooltag*`).detail().selMethod` manipulates the selection of details to remove, for a defeaturing tool *tooltag*.

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).detail().selMethod` manipulates the selection of details to remove, for a defeaturing feature *<ftag>*.

`model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing(`*tooltag*`).delete(`*<ftag>*`)` creates a defeaturing feature of type *tooltag*, tagged *<ftag>*, with the properties currently specified in the defeaturing tool. The property `delete` of the created feature is set to `selected`. If the feature *<ftag>* can be built, it is inserted in the geometry sequence after the current feature, otherwise the feature is discarded.

`model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing(`*tooltag*`).deleteAll(`*<ftag>*`)` creates a defeaturing feature of type *tooltag*, tagged *<ftag>*, with the properties currently specified in the defeaturing tool. The property `delete` of the created feature is set to `all`. If the feature *<ftag>* can be built, it is inserted in the geometry sequence after the current feature, otherwise the feature is discarded.

DEFEATURING SELECTION METHODS

For a defeaturing selection `sel` the following methods are available, in addition to the methods available for a general geometry selection.

Geometry Object Selection Methods in the *COMSOL Multiphysics Programming Reference Manual*

The `find` method on the corresponding feature or defeaturing tool provides the defeaturing selection with a list of details. Each detail is a group of geometric entities. Group numbers, *<groups>*, is an array of integers that index into the list of details.

You can select groups either by explicitly referring to group numbers, or by selecting geometric entities. In the latter case, any group that has nonempty intersection with the provided entity selection is selected.

`int[] sel.group(<groups>)` returns the group numbers for the selected groups.

`sel.addGroup(<groups>)` adds the specified groups to the selection.

`sel.setGroup(<groups>)` sets the selection groups.

`sel.removeGroup(<groups>)` removes the specified groups from the selection.

`String[] sel.groupNames()` returns a list of names of the groups found.

`String sel.groupObject(<group>)` returns the name of the geometry object that contains the specified detail group.

`int[] sel.groupEntities(<group>)` returns the entity numbers of the specified detail group.

`int sel.size()` returns the number of detail groups found.

Defeaturing Tools — Detach Faces

Use the `DetachFaces` tool to detach faces from a solid object (the parent) to form a new solid object (the child). Select the faces to detach and properties for the operation like in the corresponding feature `DetachFaces`. The detach operation is performed when you issue the command

```
model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces").
      delete(<ftag>);
```

Access the `DetectInterferences` tool by the command

```
GeomDefeature tool = model.component(<ctag>).geom(<tag>).
              defeaturing("DetectInterferences");
```

To access the input objects selection, use

```
GeomObjectSelection input = tool.selection("input");
```

To set the tolerance for detecting intersections, use

```
tool.set("abstol", value);
```

To set the tolerance for detecting gaps, use

```
tool.set("gaptol", value);
```

To find interferences, type

```
tool.find();
```

To access the resulting interferences, use

```
GeomObjectGroupSelection interf = tool.detail();
```

Each node listed in Interfering faces list in the user interface corresponds to a group of faces in `GeomObjectGroupSelection`. To access the data for a group, use its integer group index:

```
String label = interf.groupNames()[group];
String objName = interf.groupObject(group);
int[] faces = interf.groupEntities(group);
double gapSize = interf.entSize(group)[0];
```

The face numbers in the groups do not refer to the input objects. Rather, they refer to objects in a local state, where the tool has imprinted edges where objects interfere. To hide faces in the local state, first type

```
tool.localState(true);
```

to enter the local state. After doing the hiding, type

```
tool.localState(false);
```

to exit the local state.

See also `DetectInterferences`.

Defeaturing Tools — Replace Faces

Use the `ReplaceFaces` tool to delete faces and replace them either with a new face or by growing or shrinking the adjacent faces. Select the faces to replace and properties for the operation like in the corresponding feature `ReplaceFaces`. The faces are replaced when you issue the command

```
model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces").
      delete(<ftag>);
```

This adds a `ReplaceFaces` feature tagged *<ftag>* after the current feature in the geometry sequence.

Summary of Commands

- Angle
- Array
- BezierPolygon
- CapFaces
- Chamfer
- Chamfer3D
- Check
- Circle
- CircularArc
- Coincident
- Concentric
- ConvertToCOMSOL
- Copy
- CubicBezier
- DeleteFillets
- DeleteHoles
- DeleteShortEdges
- DeleteSliverFaces
- DeleteSmallFaces
- DeleteSpikes
- DetachFaces
- DetectInterferences
- DirectedDistance
- Distance
- Ellipse
- EqualDistance
- EqualRadius
- Export, ExportFinal
- Fillet
- Fillet3D
- Horizontal
- Import 3D CAD
- InterpolationCurve
- Knit
- LineSegment
- Loft
- Midsurface
- Mirror
- Move
- OffsetFaces
- Parallel
- Perpendicular
- Point
- Polygon
- Position
- Projection
- ProjectToFaces
- QuadraticBezier
- QuadraticBezier
- Radius
- Rectangle
- Repair
- ReplaceFaces
- Rotate
- Scale
- Square
- TangentConstraint
- Thicken
- TotalEdgeLength
- TransformFaces
- Vertical
- XDistance
- YDistance

Commands Grouped by Function

Commands for Creating and Modifying Geometry in 2D

FUNCTION	PURPOSE
`Projection`	Project 3D objects and entities to a 2D work plane

The Design Module enables on/off properties for constraining the other properties of 2D geometric primitive and operation commands. The property name is obtained by appending `constr` to available property names. The default is `on` in operation features and usually `off` in primitive features. A constrained property cannot be modified by constraint and dimension commands. Properties that do not have a `constr` property are always constrained. With the Design Module some features also have properties for controlling a corresponding symbol in the Graphics window.

The following 2D geometry features get additional properties with the Design Module.

- `Array`
- `BezierPolygon`
- `Chamfer`
- `Circle`
- `CircularArc`
- `Copy`
- `CubicBezier`
- `Ellipse`
- `Fillet`
- `InterpolationCurve`
- `LineSegment`
- `Mirror`
- `Move`
- `Point`
- `Polygon`
- `QuadraticBezier`
- `Rectangle`
- `Rotate`
- `Scale`
- `Square`

Commands for Creating Constraints in 2D

FUNCTION	PURPOSE
`Coincident`	Constrain two geometric entities to coincide with each other
`Concentric`	Constrain circular edges and vertices to have the same center
`EqualDistance`	Constrain the distances between two pairs of geometric entities to be equal
`EqualRadius`	Constrain two circular edges to have the same radius
`Horizontal`	Constrain straight edges to be parallel to the x-axis
`Parallel`	Constrain straight edges to be parallel
`Perpendicular`	Constrain two straight edges to be perpendicular
`TangentConstraint`	Constrain two edges to be tangent
`Vertical`	Constrain straight edges to be parallel to the y-axis

Commands for Creating Dimensions in 2D

FUNCTION	PURPOSE
`Angle`	Constrain the angle between two edges
`DirectedDistance`	Constrain the distance between two geometric entities in a given direction
`Distance`	Constrain the distance between two geometric entities
`Position`	Constrain the x- and y-coordinates of a point
`Radius`	Constrain the radius of a circular edge
`TotalEdgeLength`	Constrain the total length for a set of edges
`XDistance`	Constrain the distance in the x direction between entities
`YDistance`	Constrain the distance in the y direction between entities

Commands for Defeaturing

FUNCTION	PURPOSE
DeleteFillets	Find and delete fillets in CAD objects
DeleteHoles	Find and delete holes in CAD objects
DeleteShortEdges	Find and delete short edges in CAD objects
DeleteSliverFaces	Find and delete sliver faces in CAD objects
DeleteSmallFaces	Find and delete small faces in CAD objects
DeleteSpikes	Find and delete spikes in CAD objects
DetachFaces	Detach faces from CAD objects to form a new solid
DetectInterferences	Detect intersections, touches, gaps, and containments between CAD objects
ReplaceFaces	Delete faces from CAD objects and heal the wounds

Commands for File Import, Export, Conversion, and Repair

FUNCTION	PURPOSE
Check	Check the validity of CAD objects
ConvertToCOMSOL	Convert CAD Import Module geometry objects to COMSOL objects
Export, ExportFinal	Export geometry objects to a 3D CAD file
Import 3D CAD	Import geometry objects from a 3D CAD file
Knit	Knit surface CAD objects to form solids or surface objects
Repair	Repair CAD objects

Commands for Creating and Modifying Geometry in 3D

FUNCTION	PURPOSE
CapFaces	Add cap faces to fill holes in CAD geometries
Chamfer3D	Chamfer edges in 3D geometry objects
Fillet3D	Fillet edges in 3D geometry objects
Loft	Create a lofted surface through a set of profile curves

FUNCTION	PURPOSE
`Midsurface`	Generate a surface object that is the midsurface of a solid object in 3D
`OffsetFaces`	Offset faces of 3D objects in the normal direction
`ProjectToFaces`	Project edges to faces in 3D
`Thicken`	Generate a solid object by thickening a surface object in 3D
`TransformFaces`	Transform faces of 3D objects by displacing, rotating, and scaling

Commands in Alphabetical Order

Angle

Constrains two edges to meet at a given angle.

DESCRIPTION

TABLE 3-1: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge1`	Selection		First edge
`reverse1`	`on` | `off`	`off`	Reverse the direction of the first ray
`edge2`	Selection		Second edge
`reverse2`	`on` | `off`	`off`	Reverse the direction of the second ray
`angle`	double	`0`	Angle
`angleconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if createpar is on
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowradius`	double	`Empty`	Radius of circular arrow symbol in the Graphics window
`labelpos`	double	`0.5`	Relative label position along the arrow symbol in Graphics window
`arrowint`	`on` | `off`	`on`	Display internal or external arrow in Graphics window

SEE ALSO

`TangentConstraint`

Array

The following additional properties are available with the Design Module.

TABLE 3-2: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`displconstr`	String[2]	`{on, on}`	Constrain the displacement. Constrained properties cannot be modified by constraint and dimension functions.

BezierPolygon

The following additional properties are available with the Design Module.

TABLE 3-3: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`pconstr`	String[]	`{}`	Constrain the control points. Constrained properties cannot be modified by constraint and dimension functions.

CapFaces

PURPOSE

Add cap faces to objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"CapFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
        create(<ftag>,"CapFaces")
```

creates a `CapFaces` feature. A cap face is created for each loop of edges in the input selection. The cap faces are joined with the original objects. If new domains are created by the cap faces, these domains are made solid.

The input selection can contain more than one edge loop, but no two loops can have edges or vertices in common.

The input selection can contain edges from more than one object. In this case, each object is processed individually.

TABLE 3-4: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		The input edges.
`groupadjedg`	`on` | `off`	`off`	Extend edge selection to adjacent edge loop or chain.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The following attributes are available:

TABLE 3-5: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

Chamfer

The following additional properties are available with the Design Module.

TABLE 3-6: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`distconstr`	`on` | `off`	`off`	Constrain the distance. Constrained properties cannot be modified by constraint and dimension functions.

Chamfer3D

Chamfer edges in 3D geometry objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Chamfer3D");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Chamfer3D")
```

creates a `Chamfer3D` feature.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("edge")` to select the edge to chamfer. The default selection is empty.

TABLE 3-7: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Edges to chamfer.
`groupcontang`	`on` | `off`	`off`	Group edges to chamfer by continuous tangent
`angletol`	double	5 [deg]	Angular tolerance used when `groupcontang` is `on`.
`radius`	double	0	Radius.
`propagate`	`on` | `off`	`on`	Propagate to tangent edges.
`preserveoverlapped`	`on` | `off`	`off`	Preserve overlapped entities.
`yshaped`	`on` | `off`	`off`	Y-shaped chamfer.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.

TABLE 3-7: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

For information about the `selresult`, `selresultshow` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

The following attributes are available:

TABLE 3-8: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

EXAMPLE

Chamfer a subset of edges on a block:

```
Model model = ModelUtil.create("Model1");
model.component().create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").geomRep("cadps");
model.component("comp1").geom("geom1").create("blk1", "Block");
model.component("comp1").geom("geom1").
      create("cha1", "Chamfer3D");
model.component("comp1").geom("geom1").feature("cha1").
      selection("edge").set("blk1", new int[]{2, 4, 6, 8});
model.component("comp1").geom("geom1").feature("cha1").
      set("radius", "0.1");
model.component("comp1").geom("geom1").run();
```

SEE ALSO

`Fillet3D`

Check

Check the validity of CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Check");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property)
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).feature().`
`create(<ftag>,"Check")` creates a check feature tagged *<ftag>*. The following properties are available.

TABLE 3-9: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	`Selection`		Names of input objects

SEE ALSO

`Repair`

Circle

The following additional properties are available with the Design Module.

TABLE 3-10: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`rconstr`	on | off	`off`	Constrain the radius. Constrained properties cannot be modified by constraint and dimension functions.
`angleconstr`	on | off	`on`	Constrain the sector angle. Constrained properties cannot be modified by constraint and dimension functions.
`posconstr`	String[2]	`{off, off}`	Constrain the position. Constrained properties cannot be modified by constraint and dimension functions.

TABLE 3-10: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`rotconstr`	`on` | `off`	`on`	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.
`arrowangdispl`	double	`0`	Angular displacement of arrow symbol relative to middle of edge.
`labelradius`	double	`0.5`	Relative label position along arrow symbol.

CircularArc

The following additional properties are available with the Design Module.

TABLE 3-11: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`centerconstr`	String[2]	`{off, off}`	Constrain the center. Constrained properties cannot be modified by constraint and dimension functions.
`point1constr`	String[2]	`{off, off}`	Constrain the starting point. Constrained properties cannot be modified by constraint and dimension functions.
`point2constr`	String[2]	`{off, off}`	Constrain the endpoint. Constrained properties cannot be modified by constraint and dimension functions.
`rconstr`	`on` | `off`	`off`	Constrain the radius. Constrained properties cannot be modified by constraint and dimension functions.
`angle1constr`	`on` | `off`	`off`	Constrain the start angle. Constrained properties cannot be modified by constraint and dimension functions.
`angle2constr`	`on` | `off`	`off`	Constrain the end angle. Constrained properties cannot be modified by constraint and dimension functions.

Coincident

PURPOSE

Constrain two geometric entities to coincide with each other.

DESCRIPTION

TABLE 3-12: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or edge)
`entity2`	Selection		Second entity (vertex or edge)
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`symbolentity`	1 | 2	1	The entity the symbol in the Graphics window is attached to
`symboledgpar`	double	`0.5`	Normalized edge parameter for positioning the symbol in the Graphics window
`symboledgdispl`	double	20	Symbol displacement in left normal direction from edge (in pixels)
`symbolvtxdispl`	double[2]	`{20, 20}`	Symbol displacement from vertex (in pixels)

Concentric

PURPOSE

Constrain circular edges and vertices to have the same center.

DESCRIPTION

TABLE 3-13: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Circular edges
`vertex`	Selection		Vertices
`symboledg`	Selection		Edge the symbol in the Graphics window is attached to
`symboledgpar`	double	`0.5`	Normalized edge parameter for positioning the symbol in the Graphics window
`symboledgdispl`	double	`20`	Symbol displacement in left normal direction from edge (in pixels)
`symbolvtx`	Selection		Vertex the symbol in the Graphics window is attached to
`symbolvtxdispl`	double[2]	`{20, 20}`	Symbol displacement from vertex (in pixels)

ConvertToCOMSOL

Convert CAD objects to COMSOL objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"ConvertToCOMSOL");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
        create(<ftag>,"ConvertToCOMSOL")
```

creates a `ConvertToCOMSOL` feature.

TABLE 3-14: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Names of input objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The following attributes are available:

TABLE 3-15: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

SEE ALSO

Import 3D CAD

Copy

The following additional properties are available with the Design Module.

TABLE 3-16: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`displxconstr`	`on` | `off`	`on`	Constrain the x-displacement. Constrained properties cannot be modified by constraint and dimension functions.
`displyconstr`	`on` | `off`	`on`	Constrain the y-displacement. Constrained properties cannot be modified by constraint and dimension functions.

CubicBezier

The following additional properties are available with the Design Module.

TABLE 3-17: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`pconstr`	String[4]	`{off, off, off, off}`	Constrain the control points. Constrained properties cannot be modified by constraint and dimension functions.

DeleteFillets

Find and delete fillets in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DeleteFillets");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").find();
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      detail();
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      deleteAll(<ftag>);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets").
      delete(<ftag>)
```

creates a `DeleteFillets` feature tagged *<ftag>* with the specified properties. The property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

`model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing("Fillets").` `deleteAll(`*<ftag>*`)` works as the `delete` method, but the property `delete` is set to `all`.

It is also possible to create the `DeleteFillets` feature using the standard `create` method. The following properties are available.

TABLE 3-18: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all fillets of given size, or a selection. Only available for the feature.
`minentsize`	double	`0`	Minimum fillet radius.
`entsize`	double	`1e-3`	Maximum fillet radius.

TABLE 3-18: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Names of input objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).find()` searches the input objects for fillets with radius less than `entsize`.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).detail()` returns a selection object where you can select a subset of the fillets found.

The `find` and `detail` methods of

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillets")
```

have the corresponding functionality for the defeaturing tool.

Only faces that can be deleted without invalidating the object are deleted. If a fillet was not possible to delete, a warning is given, accessible through `model.geom(<tag>).feature(<ftag>).problem()`.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-19: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	`none`	Output variables

EXAMPLE

The following example imports the CAD object in the COMSOL Multiphysics geometry file `defeaturing_demo_3.mphbin` and finds all fillets with radius less than $4 \cdot 10^{-3}$. The first of these fillets is deleted.

```
Model model = ModelUtil.create("Model1");
```

```
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_3.mphbin");
model.component("comp1").geom("geom1").run("imp1");
model.component("comp1").geom("geom1").feature().
      create("dfi1","DeleteFillets");
model.component("comp1").geom("geom1").feature("dfi1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("dfi1").
      set("entsize",4e-3);
model.component("comp1").geom("geom1").feature("dfi1").find();
model.component("comp1").geom("geom1").feature("dfi1").detail().
      setGroup(1);
model.component("comp1").geom("geom1").run();
```

SEE ALSO

`ReplaceFaces`

DeleteHoles

Find and delete holes in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DeleteHoles");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("Holes").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("Holes").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("Holes").find();
model.component(<ctag>).geom(<tag>).defeaturing("Holes").detail();
model.component(<ctag>).geom(<tag>).defeaturing("Holes").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("Holes").
      deleteAll(<ftag>);
```

DESCRIPTION

model.component(*<ctag>*).geom(*<tag>*).defeaturing("Holes").
delete(*<ftag>*) creates a DeleteHoles feature tagged *<ftag>* with the specified properties. The property delete is set to selected. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

model.component(*<ctag>*).geom(*<tag>*).defeaturing("Holes").
deleteAll(*<ftag>*) works as the delete method, but the property delete is set to all.

It is also possible to create the DeleteHoles feature using the standard create method. The following properties are available.

TABLE 3-20: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
delete	all | selected	selected	Delete all holes of given size, or a selection. Only available for the feature
minentsize	double	0	Minimum hole radius
entsize	double	1e-3	Maximum hole radius
input	Selection		Names of input objects

TABLE 3-20: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selresult	on | off	off	Create selections of all resulting objects
selresultshow	all | obj | dom | bnd | edg | pnt | off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features
contributeto	String	none	Tag of cumulative selection to contribute to

model.component(*<ctag>*).geom(*<tag>*).feature(*<ftag>*).find() searches the input objects for holes with radius less than entsize.

model.component(*<ctag>*).geom(*<tag>*).feature(*<ftag>*).detail() returns a selection object where you can select a subset of the holes found.

The find and detail methods of model.component(*<ctag>*).geom(*<tag>*).defeaturing("Holes") have the corresponding functionality for the defeaturing tool.

Only faces that can be deleted without invalidating the object are deleted. If a hole was not possible to delete, a warning is given, accessible through model.component(*<ctag>*).geom(*<tag>*).feature(*<ftag>*).problem().

COMPATIBILITY

The following property is no longer supported:

TABLE 3-21: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
Out	stx | ftx | ctx | ptx	none	Output variables

EXAMPLE

The following example imports the CAD object in the COMSOL Multiphysics geometry file defeaturing_demo_3.mphbin and finds all holes with radius less than $4 \cdot 10^{-2}$. The first four of these holes are deleted.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
```

```
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_3.mphbin");
model.component("comp1").geom("geom1").run("imp1");
model.component("comp1").geom("geom1").feature().
      create("dho1","DeleteHoles");
model.component("comp1").geom("geom1").feature("dho1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("dho1").
      set("entsize",4e-2);
model.component("comp1").geom("geom1").feature("dho1").find();
model.component("comp1").geom("geom1").feature("dho1").
      detail().setGroup(1, 2, 3, 4);
model.component("comp1").geom("geom1").run();
```

SEE ALSO

ReplaceFaces

DeleteShortEdges

Find and delete short edges in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DeleteShortEdges");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
       set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
      find();
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
      detail();
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").
      deleteAll(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").delete(<ftag>)` creates a `DeleteShortEdges` feature tagged `<ftag>` with the specified properties. The property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

`model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges").deleteAll(<ftag>)` works as the `delete` method, but the property `delete` is set to `all`.

It is also possible to create a `DeleteShortEdges` feature using the standard `create` method. The following properties are available.

TABLE 3-22: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all edges of given size, or a selection. Only available for the feature.
`entsize`	double	`1e-3`	Maximum edge length

TABLE 3-22: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Names of input objects
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).find()` searches the input objects for edges of length less than `entsize`.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).detail()` returns a selection object where you can select a subset of the edge sets found.

The `find` and `detail` methods of `model.component(<ctag>).geom(<tag>).defeaturing("ShortEdges")` have the corresponding functionality for the defeaturing tool.

Only edges that can be deleted without invalidating the object are deleted. If an edge was not possible to delete, a warning is given, accessible through `model.component(<ctag>).geom(<tag>).feature(<ftag>).problem()`.

COMPATIBILITY

The lengths of the edges are no longer returned.

The following property is no longer supported:

TABLE 3-23: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	`none`	Output variables

EXAMPLE

The following example imports the file `defeaturing_demo_4.x_b` and finds all edges with length less than $3 \cdot 10^{-3}$. The first of these edges is deleted.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
```

```
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_4.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("dse1","DeleteShortEdges");
model.component("comp1").geom("geom1").feature("dse1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("dse1").
      set("entsize",3e-3);
model.component("comp1").geom("geom1").feature("dse1").find();
model.component("comp1").geom("geom1").feature("dse1").
      detail().setGroup(1);
model.component("comp1").geom("geom1").runAll();
```

DeleteSliverFaces

Find and delete sliver faces in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(gname).feature().
      create(<ftag>,"DeleteSliverFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      find();
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      detail();
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").
      deleteAll(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").delete(<ftag>)` creates a `DeleteSliverFaces` feature tagged `<ftag>` with the specified properties. The property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

`model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces").deleteAll(<ftag>)` works as the delete method, but the property `delete` is set to `all`.

It is also possible to create a `DeleteSliverFaces` feature using the standard `create` method. The following properties are available.

TABLE 3-24: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all sliver faces of given width, or a selection. Only available for the feature.
`entsize`	double	`1e-3`	Maximum face width.
`input`	Selection		Names of input objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

Sliver faces are narrow but long faces with large aspect ratio, which usually give rise to extremely fine local meshes in their vicinity.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).find()` searches the input objects for faces with width less than `entsize`.

`model.component(<ctag>).geom(<tag>).feature(<ftag>).detail()` returns a selection object where you can select a subset of the faces found.

The `find` and `detail` methods of `model.component(<ctag>).geom(<tag>).defeaturing("SliverFaces")` have the corresponding functionality for the defeaturing tool.

Only faces that can be deleted without invalidating the object are deleted. If a face was not possible to delete, a warning message is given.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-25: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx` | `status`	`none`	Output variables

EXAMPLE

The following example imports the geometry model from the file `defeaturing_demo_5.x_b`, finds sliver faces narrower than $2 \cdot 10^{-3}$, and deletes the first of these.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_5.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("dsl1","DeleteSliverFaces");
model.component("comp1").geom("geom1").feature("dsl1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("dsl1").
      set("entsize",2e-3);
model.component("comp1").geom("geom1").feature("dsl1").find();
model.component("comp1").geom("geom1").feature("dsl1").detail().
      setGroup(1);
model.component("comp1").geom("geom1").runAll();
```

SEE ALSO

`ReplaceFaces`, `DeleteSmallFaces`

DeleteSmallFaces

Find and delete small faces in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(gname).feature().
      create(<ftag>,"DeleteSmallFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      find();
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      detail();
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").
      deleteAll(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").delete(<ftag>)` creates a `DeleteSmallFaces` feature tagged *<ftag>* with the specified properties. The property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

`model.component(<ctag>).geom(<tag>).defeaturing("SmallFaces").deleteAll(<ftag>)` works as the `delete` method, but the property `delete` is set to `all`.

It is also possible to create a `DeleteSmallFaces` feature using the standard `create` method. The following properties are available.

TABLE 3-26: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all small faces of given size, or a selection. Only available for the feature.
`entsize`	double	`1e-3`	Maximum face size.
`input`	Selection		Names of input objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

A small face is a face that fits within a sphere of specified radius, given in the property `entsize`.

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).find()` searches the input objects for faces with size less than `entsize`.

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).detail()` returns a selection object where you can select a subset of the faces found.

The `find` and `detail` methods of `model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing("SmallFaces")` have the corresponding functionality for the defeaturing tool.

Only faces that can be deleted without invalidating the object are deleted. If a face was not possible to delete, a warning message is given, accessible through `model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).problem()`.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-27: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx` | `status`	`none`	Output variables.

EXAMPLE

The following example imports the geometry model from the file `defeaturing_demo_6.x_b`, finds sliver faces narrower than 10^{-3}, and deletes the first of these.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_6.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("df1","DeleteSmallFaces");
model.component("comp1").geom("geom1").feature("df1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("df1").find();
model.component("comp1").geom("geom1").feature("df1").detail().
      setGroup(1);
model.component("comp1").geom("geom1").run();
```

SEE ALSO

`ReplaceFaces`, `DeleteSliverFaces`

DeleteSpikes

Find and delete spikes in CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DeleteSpikes");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).find();
model.component(<ctag>).geom(<tag>).feature(<ftag>).detail();

model.component(<ctag>).geom(<tag>).defeaturing("Spikes").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("Spikes").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("Spikes").find();
model.component(<ctag>).geom(<tag>).defeaturing("Spikes").detail();
model.component(<ctag>).geom(<tag>).defeaturing("Spikes").
      delete(<ftag>);
model.component(<ctag>).geom(<tag>).defeaturing("Spikes").
      deleteAll(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("DeleteSpikes").delete(<ftag>)` creates a `DeleteSpikes` feature tagged *<ftag>* with the specified properties. The property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

`model.component(<ctag>).geom(<tag>).defeaturing("DeleteSpikes").deleteAll(<ftag>)` works as the `delete` method, but the property `delete` is set to `all`.

It is also possible to create a `DeleteSpikes` feature using the standard `create` method. The following properties are available.

TABLE 3-28: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all spikes of given width, or a selection. Only available for the feature.
`entsize`	double	`1e-3`	Maximum spike width.
`input`	Selection		Names of input objects.

TABLE 3-28: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

A spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges.

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).find()` searches the input objects for spikes of width less than `entsize`.

`model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).detail()` returns a selection object where you can select a subset of the spikes found.

The find and detail methods of `model.component(`*<ctag>*`).geom(`*<tag>*`).defeaturing("Spikes")` have the corresponding functionality for the defeaturing tool.

Only spikes that can be deleted without invalidating the object are deleted. If a spike was not possible to delete, a warning message is given, accessible through `model.component(`*<ctag>*`).geom(`*<tag>*`).feature(`*<ftag>*`).problem()`.

COMPATIBILITY

The width of each spike is no longer returned.

The following property is no longer supported:

TABLE 3-29: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx` | `status`	`none`	Output variables.

EXAMPLE

The following example imports the geometry model from the file `defeaturing_demo_7.x_b`, finds all spikes narrower than 10^{-4}, and deletes the first of these.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_7.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("dsp1","DeleteSpikes");
model.component("comp1").geom("geom1").feature("dsp1").
      selection("input").
      set("imp1");
model.component("comp1").geom("geom1").feature("dsp1").
      set("entsize",1e-4);
model.component("comp1").geom("geom1").feature("dsp1").find();
model.component("comp1").geom("geom1").feature("dsp1").detail().
      setGroup(1);
model.component("comp1").geom("geom1").runAll();
```

SEE ALSO

`DeleteShortEdges`, `DeleteSliverFaces`

DetachFaces

Detach faces from CAD objects to form a new (child) solid.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DetachFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);

model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces").
      selection(property);
model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces").
      delete(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces").delete(<ftag>)` creates a `DetachFaces` feature tagged *<ftag>* with the specified properties. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

It is also possible to create a `DetachFaces` feature using the standard `create` method.

TABLE 3-30: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	`Selection`		Faces to detach.
`healchild`	`fill` | `patchchild` | `patchparent`	`patchparent`	Healing method used on the child object.
`healparent`	`fill` | `patch`	`patch`	Healing method used on the parent object.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The faces in the property `input` are detached from their *parent* object. A new solid, the *child* object, are formed from the detached faces. The output objects are the healed parent and child objects.

The property `healparent` determines how the parent object is healed to form a new solid after detaching the faces. The value `fill` means that a new face is formed based on the surrounding edges of each wound. The value `patch` means that the surrounding faces of each wound are grown or shrunk.

The property `healchild` determines how the child solid is constructed from the detached faces. The value `fill` means that a new face is formed based on the surrounding edges of each wound. The value `patchchild` means that the detached faces are grown or shrunk to form a solid. The value `patchparent` means that the parent faces surrounding the detached faces are grown or shrunk to form a solid together with the detached faces.

EXAMPLE

The following example imports the COMSOL Multiphysics geometry file `defeaturing_demo_2.mphbin` and detaches a hole defined by a set of faces:

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "defeaturing_demo_2.mphbin");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("det1","DetachFaces");
model.component("comp1").geom("geom1").feature("det1").
      selection("input").set("imp1",6,7,8,9,11,12,13);
model.component("comp1").geom("geom1").runAll();
```

COMPATIBILITY

The following property is no longer supported:

TABLE 3-31: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	`none`	Output variables

SEE ALSO

`ReplaceFaces`

DetectInterferences

Detect intersections, touches, gaps, and containments between CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences").selection(property);
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences").set(property,<value>);
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences").selection(property);
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences").find();
model.component(<ctag>).geom(<tag>).
      defeaturing("DetectInterferences").detail();
```

DESCRIPTION

See Defeaturing Tools — Detect Interferences.

Available properties:

TABLE 3-32: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Input objects
`abstol`	double	0.1[mm]	Absolute tolerance for detecting intersections
`gaptol`	double	1[mm]	Absolute tolerance for detecting gaps
`showingraphics`	`interferingonly` | `selected` | `other` | `both` | `all`	`selected`	Objects to show in graphics
`groupbyobject`	boolean	`false`	Group interferences by object in GUI
`showintersections`	boolean	`true`	List intersections in GUI
`showtouches`	boolean	`true`	List touches in GUI
`showgaps`	boolean	`true`	List gaps in GUI
`showcontainments`	boolean	`true`	List containments in GUI

DirectedDistance

PURPOSE

Constrain the distance in a given direction between two geometric entities to a given value.

DESCRIPTION

TABLE 3-33: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or curved edge)
`entity2`	Selection		Second entity (vertex or curved edge)
`direction`	`vector` | `parallel` | `perpendicular`	`vector`	Direction
`vector`	double[2]	`{1, 0}`	Direction vector (used when direction is vector)
`edge`	Selection		Straight edge determining the direction (used when direction is parallel or perpendicular)
`distance`	double	`0`	Distance
`distanceconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if createpar is on
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`Empty`	Displacement of arrow symbol in the normal direction
`labelpos`	double	`0.5`	Relative label position along arrow symbol
`arrowint`	`on` | `off`	`on`	Internal or external arrow symbol

SEE ALSO

`Distance`, `XDistance`, `YDistance`

PURPOSE

Constrain the distance between two geometric entities to a given value.

DESCRIPTION

TABLE 3-34: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or edge)
`entity2`	Selection		Second entity (vertex or edge)
`distance`	double	`0`	Distance
`distanceconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if createpar is on
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`Empty`	Displacement of arrow symbol in the normal direction
`labelpos`	double	`0.5`	Relative label position along arrow symbol
`arrowint`	`on` | `off`	`on`	Internal or external arrow symbol

SEE ALSO

`DirectedDistance`, `XDistance`, `YDistance`, `EqualDistance`, `TotalEdgeLength`

Ellipse

The following additional properties are available with the Design Module.

TABLE 3-35: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`semiaxesconstr`	String[2]	`{off, off}`	Constrain the semiaxes. Constrained properties cannot be modified by constraint and dimension functions.
`angleconstr`	`on` | `off`	`on`	Constrain the sector angle. Constrained properties cannot be modified by constraint and dimension functions.
`posconstr`	String[2]	`{off, off}`	Constrain the position. Constrained properties cannot be modified by constraint and dimension functions.
`rotconstr`	`on` | `off`	`on`	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.

EqualDistance

PURPOSE

Constrain the distances between two pairs of geometric entities to be equal.

DESCRIPTION

TABLE 3-36: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or edge)
`entity2`	Selection		Second entity (vertex or edge)
`entity3`	Selection		Third entity (vertex or edge)
`entity4`	Selection		Fourth entity (vertex or edge)
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`helppoint3`	double[2]	`{0, 0}`	Third help point
`helppoint4`	double[2]	`{0, 0}`	Fourth help point
`arrowdispl`	double[2]	`Empty`	Displacement of arrow symbols in the normal direction
`labelpos`	double[2]	`{0.5, 0.5}`	Relative label position along arrow symbols
`arrowint`	String[2]	`{on, on}`	Internal or external arrow symbols
`label`	String		Label text

SEE ALSO

`Distance`

EqualRadius

PURPOSE

Constrain two circular edges to have the same radius.

DESCRIPTION

TABLE 3-37: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge1`	Selection		First circular edge
`edge2`	Selection		Second circular edge
`arrowangdispl`	double[2]	`{0, 0}`	Angular displacement of arrow symbols relative to middle of edges
`labelradius`	double[2]	`{0.5, 0.5}`	Relative label position along arrow symbols
`label`	String		Label text

SEE ALSO

`Radius`

Export, ExportFinal

Using the CAD Import Module, Design Module, or a LiveLink product for CAD software, export selected geometry objects or the finalized geometry to a 3D CAD format, such as ACIS, Parasolid, STEP, and IGES.

To export selected geometry objects to a file, first select the objects to export using

```
model.component(<ctag>).geom(<tag>).export().selection().set(<obj
names>);
```

where *<objnames>* is a string array of object names.

Set the file format using

```
model.component(<ctag>).geom(<tag>).export().setType(<format>);
```

where *<format>* determines the file format. See Table 3-38 for valid type value names available with the CAD Import Module, Design Module, or a LiveLink product for CAD software.

TABLE 3-38: FILE FORMATS SUPPORTED FOR EXPORT.

FILE FORMAT	FILE EXTENSION	TYPE VALUE
Parasolid Binary (3D)	`.x_b, .xmt_bin`	`parasolidbin`
Parasolid Text (3D)	`.x_t, .xmt_txt`	`parasolidascii`
ACIS Binary (3D)	`.sab`	`acisbin`
ACIS Text (3D)	`.sat`	`acisascii`

TABLE 3-38: FILE FORMATS SUPPORTED FOR EXPORT.

FILE FORMAT	FILE EXTENSION	TYPE VALUE
IGES File (3D)	`.igs`, `.iges`	`iges`
STEP File (3D)	`.step`, `.stp`	`step`

Check which file format is set for the export using

```
String formatType =
model.component(<ctag>).geom(<tag>).export().getType();
```

To export the file enter

```
model.component(<ctag>).geom(<tag>).export(<filename>);
```

To export the finalized geometry to a file, enter

```
model.component(<ctag>).geom(<tag>).exportFinal(<filename>);
```

where *<filename>* is a string.

EXPORTING TO AN ACIS FILE

When exporting to an ACIS file you can set the ACIS file format version using

```
model.component(<ctag>).geom(<tag>).export().setAcisVersion(<vers
ion>);
```

where *<version>* is a string `4.0`, `7.0`, or `2016 1.0`. Default is `2016 1.0`.

Check which ACIS file format version is set for the export using

```
model.component(<ctag>).geom(<tag>).export().getAcisVersion();
```

EXPORTING TO A PARASOLID FILE

The Parasolid text or binary file generated by the export is of version 37.

When exporting to a Parasolid format, a unit conversion can optionally be performed during export. Use the following method to select the export length unit:

```
model.component(<ctag>).geom(<tag>).export().setLengthUnit(<unit>
);
```

where *<unit>* is either `fromgeom` (default) to disable unit conversion or a COMSOL Multiphysics length unit, such as `m` for meters or `in` for inches. To get the current value of the export length unit type:

```
model.component(<ctag>).geom(<tag>).export().getLengthUnit();
```

To decide how the nonmanifold objects are exported use the following method:

```
model.component(<ctag>).geom(<tag>).export().setSplitInManifold(<value>);
```

where *<value>* is either `true` (default) to split the objects into manifold objects during the export, or `false` to export the unmodified objects. To get the current method used for exporting nonmanifold objects use:

```
model.component(<ctag>).geom(<tag>).export().getSplitInManifold();
```

EXPORTING TO AN IGES FILE

When exporting to the IGES format, a unit conversion can optionally be performed during export. Use the following method to select the export length unit:

```
model.component(<ctag>).geom(<tag>).export().setLengthUnitIGES(<unit>);
```

where *<unit>* is either `fromgeom` (default) to disable unit conversion or a supported length unit: `uin`, `um`, `mil`, `mm`, `cm`, `in`, `ft`, `m`, `km`, `mi`.To get the current value of the export length unit type:

```
model.component(<ctag>).geom(<tag>).export().getLengthUnitIGES();
```

EXPORTING TO A STEP FILE

When exporting to the STEP format, a unit conversion can optionally be performed during export. Use the following method to select the export length unit:

```
model.component(<ctag>).geom(<tag>).export().setLengthUnitSTEP(<unit>);
```

where *<unit>* is either `fromgeom` (default) to disable unit conversion or a supported length unit: `nm`, `uin`, `um`, `mil`, `mm`, `cm`, `in`, `dm`, `ft`, `m`, `km`, `mi`. To get the current value of the export length unit type:

```
model.component(<ctag>).geom(<tag>).export().getLengthUnitSTEP();
```

SEE ALSO

Import 3D CAD

Horizontal

PURPOSE

Constrains straight edges to be parallel to the x-axis.

DESCRIPTION

TABLE 3-39: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Straight edges
`symboledg`	Selection		Edge the symbol in the Graphics window is attached to
`symboledgpar`	double	`0.5`	Normalized edge parameter for positioning the symbol
`symboledgdispl`	double	`20`	Symbol displacement in left normal direction from edge (in pixels)

SEE ALSO

`Angle`, `Parallel`, `Vertical`

Fillet

The following additional properties are available with the Design Module.

TABLE 3-40: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`radiusconstr`	`on` | `off`	`off`	Constrain the radius. Constrained properties cannot be modified by constraint and dimension functions.

Fillet3D

Fillet edges in 3D geometry objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Fillet3D");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Fillet3D")
```

creates a `Fillet3D` feature.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("edge")` to select the edge to fillet. The default selection is empty.

TABLE 3-41: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`fillettype`	`constant radius` | `constant width` | `variable radius`	`constant radius`	Type of fillet
`edge`	Selection		Edges to fillet
`groupcontang`	`on` | `off`	`off`	Group edges to fillet by continuous tangent
`angletol`	double	5 [deg]	Angular tolerance used when `groupcontang` is `on`.
`radius`	double	0	Radius
`width`	double	0	Width used if `fillettype` is `constantwidth`.
`tableedge`	int[]		Edge column of the Radii table
`tableparam`	double[] | String[]		Parameter column of the Radii table, containing relative arc length parameters (between 0 and 1) for the corresponding edge.
`tableradius`	double[] | String[]		Radius column of the Radii table. Empty values are allowed, meaning that the radius specification is ignored.
`tableclamp`	boolean[] | String[]		Clamp column of the Radii table.
`propagate`	`on` | `off`	`on`	Propagate to tangent edges. Not used when `fillettype` is `variableradius`.
`preserveoverlapped`	`on` | `off`	`off`	Preserve overlapped entities

TABLE 3-41: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
yshaped	on | off	off	Y-shaped fillet. Not used when filletype is constantwidth.
filletsharp	on | off	off	Fillet sharp edges at vertices. Not used when filletype is constantwidth.
selresult	on | off	off	Create selections of all resulting objects.
selresultshow	all | obj | dom | bnd | edg | pnt | off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
contributeto	String	none	Tag of cumulative selection to contribute to.

For information about the selresult, selresultshow and contributeto properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

The following attributes are available:

TABLE 3-42: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
construction	on | off | inherit	inherit	Designate the resulting objects as construction geometry. Use inherit to set the construction geometry attribute only if all input objects are construction geometry.

EXAMPLE

Fillet a subset of edges on a block:

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").geomRep("cadps");
model.component("comp1").geom("geom1").create("blk1", "Block");
model.component("comp1").geom("geom1").
      create("fil1", "Fillet3D");
```

```
model.component("comp1").geom("geom1").feature("fil1").
      selection("edge").set("blk1", new int[]{2, 4, 6, 8});
model.component("comp1").geom("geom1").feature("fil1").
      set("radius", "0.1");
model.component("comp1").geom("geom1").run();
```

SEE ALSO

Chamfer3D

Import 3D CAD

Import geometry objects from a 3D CAD file using the CAD Import Module, Design Module, or a LiveLink product for CAD software.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Import");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      loadAssemblyTree();
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      importData();
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).feature().`
`create(<ftag>,"Import")` creates an import feature. When the property `filename` is set to a filename recognized as a 3D CAD file, the property `type` is set to `cad`. The following properties are available.

TABLE 3-43: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`check`	`on` | `off`		Check imported objects for errors.
`deletedetails`	`on` | `off`	`on`	Delete small geometric details from the geometry.
`filename`	`String`		Filename.

TABLE 3-43: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`fillholes`	`on` | `off`	`off`	Attempt to generate new faces to replace missing geometry if the property knit is solid or surface.
`fixerrors`	`auto` | `on` | `off`	`auto`	Attempt to repair topological errors. Use `auto` to repair only if errors are detected.
`healedges`	`on` | `off`	`off`	Attempt to replace tolerant edges with accurately calculated edges.
`importbodynames`	`auto` | `on` | `off`	`auto`	Include the body name in the object name. This property is available only when `filename` is set to a STEP file extension. Use `auto` to include the body name only for multibody parts.
`importtol`	double	`1e-5`	Absolute repair tolerance.
`keepbnd`	`on` | `off`	`on`	Import surface objects.
`keepfree`	`on` | `off`	`off`	Import curve and point objects.
`keepsolid`	`on` | `off`	`on`	Import solid objects.
`knit`	`solid` | `surface` | `off`	`solid`	Knit together surface objects to form solids or surface objects.
`minimizetol`	`on` | `off`	`off`	Attempt to reduce edge and vertex tolerances.
`removeredundant`	`on` | `off`	`off`	Remove redundant edges and vertices.
`simplify`	`on` | `off`	`off`	Simplify the underlying curve and surface manifolds of geometric entities.
`type`	`cad`		Type of import.
`unit`	`source` | `current`	`source`	Take length unit from file or from the current geometry unit.
`unitecurves`	`on` | `off`	`on`	Unite curve objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.

TABLE 3-43: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The file to import is specified by `filename`, which can have of any of the following formats:

TABLE 3-44: SUPPORTED 3D CAD FILE FORMATS.

FILE FORMAT	NOTE	FILE EXTENSION
ACIS®	1, 2	`.sat`, `.sab`
AutoCAD®	1, 3	`.dwg`, `.dxf`
CATIA® V5	3, 4	`.CATPart`, `.CATProduct`
IGES	1, 2	`.igs`, `.iges`
Inventor®	1, 3	`.ipt`, `.iam`
NX™	1, 3	`.prt`
Parasolid®	1	`.x_t`, `.x_b`
PTC Creo Parametric™	1, 2	`.prt`, `.asm`
PTC Pro/ENGINEER®	1, 2	`.prt`, `.asm`
SOLIDWORKS®	1, 3, 5	`.sldprt`, `.sldasm`
STEP	1, 2	`.step`, `.stp`

Note 1: This format requires a license for the CAD Import Module, Design Module, or a LiveLink product for a CAD package.

Note 2: This format is available only on supported Windows® and Linux operating systems with Intel® 64-bit processors and on macOS.

Note 3: This format is available only on supported Windows® and Linux operating systems with Intel® 64-bit processors.

Note 4: This format requires, in addition to the CAD Import Module, Design Module, or a LiveLink product for a CAD package, a license for the File Import for CATIA V5 module.

Note 5: Files saved with an educational version of the SOLIDWORKS® software are not supported.

The imported geometry objects are represented using the Parasolid geometry kernel, which is the geometry kernel utilized by the CAD Import Module and the LiveLink products for CAD software.

When importing CAD assemblies saved in the file formats of the CATIA V5, Inventor, NX, PTC Creo Parametric, PTC Pro/ENGINEER, and SOLIDWORKS software it is possible to set up the import for importing only specified components. The method

```
model.geom(gname).feature(<ftag>).loadAssemblyTree()
```

loads the assembly tree from the chosen assembly file, after which the following properties become available:

TABLE 3-45: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`assemnodename`	String[]		The names of the nodes in the assembly tree
`assemnodefile`	String[]		The file names of the nodes in the assembly tree.
`assemnodedepth`	int[]		The depths of the nodes in the assembly tree.
`assemnodetransform`	double[][]		The linear transforms of the nodes in the assembly tree. Each linear transform Ax+b is 9 numbers (in the order column 1, column 2, column 3) for the 3-by-3 rotation matrix A followed by 3 numbers for the translation vector b.
`assemnodeimport`	String[]		The import flag ("on" or "off") for the nodes in the assembly tree.

The properties `assemnodename`, `assemnodefile`, `assemnodedepth`, and `assemnodetransform` are available for reading and should not be modified.

The method

```
model.geom(gname).feature(<ftag>).importData()
```

imports the file again, even if the feature is built.

The import can generate object, boundary, edge, and point selections based on material, layer, and color assignments in the 3D CAD file. The following properties are available for working these selections:

TABLE 3-46: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selcadshownamesfromfileobj`	boolean	`false`	Show the object selection names from the file in the GUI.
`selcadnameobj`	String[]	Empty	Names of object selections in 3D CAD import.
`selcadnameinfileobj`	String[]	Empty	Original names of object selections in 3D CAD import. Read-only.
`selcadkeepobj`	`on` | `off`	Empty	Keep object selections in 3D CAD import.
`selcadshowobj`	`on` | `off`	Empty	Show object selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
`selcadcontributetoobj`	String[]	Empty	Tags of cumulative selection to contribute to (or `none` to not contribute), for object selections in 3D CAD import.
`selcadtagobj`	String[]	Empty	Tags of object selections (read only, hidden in GUI) in 3D CAD import.
`selcadcolorobj`	String[]	Empty	Colors of object selections (read only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be `none` (in which case it will be displayed in yellow).

TABLE 3-46: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selindividualintable`	boolean	`false`	Show individual object selections and, for the knit case, individual original object selections in the CAD-tables.
`selcadshownamesfromfilebnd`	boolean	`false`	Show the boundary selection names from the file in the GUI.
`selcadnamebnd`	String[]	Empty	Names of boundary selections in 3D CAD import.
`selcadnameinfilebnd`	String[]	Empty	Original names of boundary selections in 3D CAD import. Read only.
`selcadkeepbnd`	`on` | `off`	Empty	Keep boundary selections in 3D CAD import.
`selcadshowbnd`	`on` | `off`	Empty	Show boundary selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
`selcadcontributetobnd`	String[]	Empty	Tags of cumulative selection to contribute to (or `none` to not contribute), for boundary selections in 3D CAD import.
`selcadtagbnd`	String[]	Empty	Tags of boundary selections (read-only, hidden in GUI) in 3D CAD import.

TABLE 3-46: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selcadcolorbnd`	String[]	Empty	Colors of boundary selections (read only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be `none` (in which case it will be displayed in yellow).
`selcadshownamesfromfileedg`	boolean	`false`	Show the edge selection names from the file in the GUI.
`selcadnameedg`	String[]	Empty	Names of edge selections in 3D CAD import.
`selcadnameinfileedg`	String[]	Empty	Original names of edge selections in 3D CAD import. Read only.
`selcadkeepedg`	`on` | `off`	Empty	Keep edge selections in 3D CAD import.
`selcadshowedg`	`on` | `off`	Empty	Show edge selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
`selcadcontributetoedg`	String[]	Empty	Tags of cumulative selection to contribute to (or `none` to not contribute), for edge selections in 3D CAD import.
`selcadtagedg`	String[]	Empty	Tags of edge selections (read only, hidden in GUI) in 3D CAD import.

TABLE 3-46: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selcadcoloredg`	String[]	Empty	Colors of edge selections (read only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be `none` (in which case it will be displayed in yellow).
`selcadshownamesfromfilepnt`	boolean	`false`	Show the point selection names from the file in the GUI.
`selcadnamepnt`	String[]	Empty	Names of point selections in 3D CAD import.
`selcadnameinfilepnt`	String[]	Empty	Original names of point selections in 3D CAD import. Read only.
`selcadkeeppnt`	`on` | `off`	Empty	Keep point selections in 3D CAD import.
`selcadshowpnt`	`on` | `off`	Empty	Show point selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
`selcadcontributetopnt`	String[]	Empty	Tags of cumulative selection to contribute to (or `none` to not contribute), for point selections in 3D CAD import.

TABLE 3-46: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selcadtagpnt`	String[]	Empty	Tags of point selections (read only, hidden in GUI) in 3D CAD import.
`selcadcolorpnt`	String[]	Empty	Colors of point selections (read only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be `none` (in which case it will be displayed in yellow).

The following attributes are available:

TABLE 3-47: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off`	`off`	Designate the resulting objects as construction geometry.

COMPATIBILITY

The following properties are no longer supported:

TABLE 3-48: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`coercion`	`solid` | `face` | `off`	`solid`	Alias for knit. face is equivalent to surface.
`repair`	`on` | `off`	`on`	Replaced by deletedetails.

SEE ALSO

`Export`, `ExportFinal`

InterpolationCurve

The following additional properties are available with the Design Module.

TABLE 3-49: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`tableconstr`	String[]	`{}`	Constrain the interpolation points. Constrained properties cannot be modified by constraint and dimension functions.
`starttangconstr`	String[2]	`{off, off}`	Constrain the start tangent. Constrained properties cannot be modified by constraint and dimension functions.
`endtangconstr`	String[2]	`{off, off}`	Constrain the end tangent. Constrained properties cannot be modified by constraint and dimension functions.

Knit

Knit surface CAD objects to form solids or surface objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Knit");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property)
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

`model.component(<ctag>).geom(<ftag>).feature().create(<ftag>,"Knit")` creates a knit feature tagged `<ftag>`. The following properties are available.

TABLE 3-50: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`fillholes`	`on` | `off`	`off`	Attempt to generate new faces to replace missing geometry
`input`	`Selection`		Names of input surface objects.
`repairtol`	`double`	`1e-5`	Absolute repair tolerance.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

This function also removes gaps and spikes that are within the absolute tolerance specified in the property `repairtol`.

The following attributes are available:

TABLE 3-51: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-52: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	`none`	Output variables.

EXAMPLE

The following example imports the file `repair_demo_2.x_b`, and knits the surface objects into a solid. A gap is also removed during the operation.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename","repair_demo_2.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("knit1","Knit");
model.component("comp1").geom("geom1").feature("knit1").
      selection("input").set("imp1");
model.component("comp1").geom("geom1").feature("knit1").
      set("repairtol",1e-3);
model.component("comp1").geom("geom1").runAll();
```

SEE ALSO

`Repair`

LineSegment

The following additional properties are available with the Design Module.

TABLE 3-53: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`coord1constr`	String[2]	`{off, off}`	Constrain the coordinates of the starting point. Constrained properties cannot be modified by constraint and dimension functions.
`coord2constr`	String[2]	`{off, off}`	Constrain the coordinates of the endpoint. Constrained properties cannot be modified by constraint and dimension functions.

Loft

Create a lofted surface through a set of profile curves.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Loft");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Loft")
```

creates a `Loft` feature.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("profile")` to select loft profiles that are not specified as start or end profiles. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("startprofile")` to select the loft start profile. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("endprofile")` to select the loft start profile. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("guide")` to select guide curves that are not specified as start or end guide curves. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("startguide")` to select the start guide curve. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("endguide")` to select the end guide curve. The default selection is empty.

TABLE 3-54: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`periodic`	`on` | `off`	`off`	Periodic loft.
`unite`	`on` | `off`	`on`	Unite with input objects.
`crossfaces`	`on` | `off`	`off`	Keep intermediate profile faces.
`type`	`solid` | `surface`	`solid`	Object type.
`facepartioning`	`minimal` | `columns` | `grid`	`minimal`	Face partitioning.
`removeredundant`	`on` | `off`	`off`	Remove vertices that separate edges with identical curves before the loft.
`profile`	Selection		Profiles.
`startprofile`	Selection		Start profile selection.
`startprofiledir`	`notprescribed` | `parallel` | `perpendicular` | `atangle`	`notprescribed`	Loft direction.
`startprofilerel`	`profilefaces` | `adjacent` | `profileedges`	`adjacent`	Relative to.
`startprofileangle`	double	0	Angle.
`endprofile`	Selection		End profile selection.
`endprofiledir`	`notprescribed` | `parallel` | `perpendicular` | `atangle`	`notprescribed`	Loft direction.
`endprofilerel`	`profilefaces` | `adjacent` | `profileedges`	`adjacent`	Relative to.
`endprofileangle`	double	0	Angle.
`guide`	Selection		Guide curves.
`startguide`	Selection		Start guide curve.
`startguidedir`	`notprescribed` | `parallel`	`notprescribed`	Loft surface direction.

TABLE 3-54: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
endguide	Selection		Start guide curve.
endguidedir	notprescribed | parallel	notprescribed	Loft surface direction.
selresult	on | off	off	Create selections of all resulting objects.
selresultshow	all | obj | dom | bnd | edg | pnt | off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
propagatesel	on | off	on	Propagate selections from input objects to resulting objects.
contributeto	String	none	Tag of cumulative selection to contribute to.

For information about the selresult, selresultshow and contributeto properties, search the online help in COMSOL Multiphysics to locate and search the documentation.

The following attributes are available:

TABLE 3-55: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
construction	on | off | inherit	inherit	Designate the resulting objects as construction geometry. Use inherit to set the construction geometry attribute only if all input objects are construction geometry.

EXAMPLE

Create a cylinder from two disc profiles. This illustrates two different ways of specifying the profile.

```
Model model = ModelUtil.create("Model1");
model.component().create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").geomRep("cadps");
```

```
model.component("comp1").geom("geom1").feature().
      create("wp1", "WorkPlane");
model.component("comp1").geom("geom1").feature("wp1").geom().
      create("c1", "Circle");
model.component("comp1").geom("geom1").create("copy1", "Copy");
model.component("comp1").geom("geom1").feature("copy1").
      selection("input").set(new String[]{"wp1"});
model.component("comp1").geom("geom1").feature("copy1").
      set("displz", "1");
model.component("comp1").geom("geom1").create("loft1", "Loft");
model.component("comp1").geom("geom1").feature("loft1").
      selection("profile").set(new String[]{"wp1"});
model.component("comp1").geom("geom1").feature("loft1").
      selection("startprofile").init(1);
model.component("comp1").geom("geom1").feature("loft1").
      selection("startprofile").
      set("copy1", new int[]{1, 2, 3, 4});
model.component("comp1").geom("geom1").run();
```

Midsurface

Generate a surface object that is the midsurface of a solid object in 3D.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Midsurface");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Midsurface")
```

creates a `Midsurface` feature.

Use

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection("input")
```

to select the objects for the midsurface operation. The default selection is empty.

TABLE 3-56: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`repair`	`on` | `off`	`on`	Repair overlaps
`split`	`on` | `off`	`off`	Split in smooth components
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Cumulative selection to contribute to

For information about the `selresult`, `selresultshow` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

The following attributes are available:

TABLE 3-57: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

EXAMPLE

Generate the midsurface of a thin block:

```
Model model = ModelUtil.create("Model1");
model.component().create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").geomRep("cadps");
model.component("comp1").geom("geom1").create("blk1", "Block");
model.component("comp1").geom("geom1").feature("blk1").
      set("size", new String[]{"1", "1", "0.1"});
model.component("comp1").geom("geom1").
      create("mid1", "Midsurface");
model.component("comp1").geom("geom1").feature("mid1").
      selection("input").set(new String[]{"blk1"});
model.component("comp1").geom("geom1").run();
```

SEE ALSO

Thicken

Mirror

The following additional properties are available with the Design Module.

TABLE 3-58: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`posconstr`	String[2]	`{on, on}`	Constrain point on line of reflection. Constrained properties cannot be modified by constraint and dimension functions.
`axisconstr`	String[2]	`{on, on}`	Constrain normal vector to line of reflection. Constrained properties cannot be modified by constraint and dimension functions.

Move

The following additional properties are available with the Design Module.

TABLE 3-59: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`displxconstr`	`on` | `off`	`on`	Constrain the x-displacement. Constrained properties cannot be modified by constraint and dimension functions.
`displyconstr`	`on` | `off`	`on`	Constrain the y-displacement. Constrained properties cannot be modified by constraint and dimension functions.

OffsetFaces

PURPOSE

Offset faces of 3D objects in the normal direction.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"OffsetFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property)
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"OffsetFaces")
```

creates an offset faces feature tagged *<ftag>*. The following properties are available.

TABLE 3-60: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
face	Selection	Empty	Faces to offset
keep	on | off	off	Keep input objects
subtractinput	on | off	off	Offset the faces, and from the resulting object subtract the corresponding input object
distance	double	0	Offset distance
reverse	on | off	off	Reverse offset direction
fillet	on | off	off	Fillet convex edges
stepfaces	contang | no	contang	Create step faces
exceptendfacesmerged	on | off	off	Except if faces at ends of edge can be merged. Only used when stepfaces is contang
perpstepedges	on | off	off	Create perpendicular step edges for surface objects
overflow	auto | extendoffset | extendother | capfaces | error	auto	Overflow handling
selresult	on | off	off	Create selections of all resulting objects

TABLE 3-60: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections, if `selresult` is `on`, of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry
`color`	`none` | `custom` | `integer`between 1 and the number of colors in the current theme	`none`	The color of the selection, either given as an integer indicating a color in the color theme, or as a custom color in the customcolor property. Coloring is only available when selresult in active
`customcolor`	RGB-triplet	Next available theme color	The color to use. Active when color is set to custom
`contributeto`	String	`none`	Tag of cumulative selection to contribute to
`propagatesel`	`on` | `off`	`on`	Propagate selections from input objects to resulting objects.

The following attributes are available:

TABLE 3-61: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

SEE ALSO

`TransformFaces`

Parallel

PURPOSE

Constrains straight edges to be parallel.

DESCRIPTION

TABLE 3-62: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
edge	Selection		Straight edges
symboledg	Selection		Edge the symbol in the Graphics window is attached to
symboledgpar	double	0.5	Normalized edge parameter for positioning the symbol
symboledgdispl	double	20	Symbol displacement in left normal direction from edge (in pixels)

SEE ALSO

Angle

Perpendicular

PURPOSE

Constrains two straight edges to be orthogonal.

DESCRIPTION

TABLE 3-63: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
entity1	Selection		First edge
entity2	Selection		Second edge
symbolentity	1 | 2	1	Edge the symbol in the Graphics window is attached to
symboledgpar	double	0	Normalized edge parameter for positioning the symbol
symboledgdispl	double	20	Symbol displacement in left normal direction from edge (in pixels)

SEE ALSO

Angle

Point

The following additional properties are available with the Design Module.

TABLE 3-64: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`pconstr`	String[2]	`{off, off}`	Constrain the coordinates. Constrained properties cannot be modified by constraint and dimension functions.

Polygon

The following additional properties are available with the Design Module.

TABLE 3-65: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`tableconstr`	String[]	`{}`	Constrain the polygon points. Constrained properties cannot be modified by constraint and dimension functions.

Position

PURPOSE

Constrain a vertex to have given coordinates.

DESCRIPTION

TABLE 3-66: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`vertex`	Selection		Vertex to fix
`pos`	double[2]	`{0, 0}`	Coordinates
`posconstr`	String[]	`{"on","on"}`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`{"",""}`	Specify parameter name if createpar is on
`symbolvtxdispl`	double[2]	`{20, 20}`	Displacement of the symbol in the Graphics window from the vertex (in pixels)

Projection

PURPOSE

Project 3D objects and entities to a 2D work plane.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature(<wptag>).
      geom().create(<ftag>,"Projection");
model.component(<ctag>).geom(<tag>).feature(<wptag>).
      geom().feature(<ftag>).selection("input");
model.component(<ctag>).geom(<tag>).feature(<wptag>).
      geom().feature(<ftag>).set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<wptag>).
      geom().feature(<ftag>).getType(property)
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

Use

```
model.component(<ctag>).geom(<tag>).feature(<wptag>).geom().
      create(<ftag>,"Projection")
```

to create a projection feature tagged *<ftag>* in the 2D sequence of the work plane feature *<wptag>*. It can compute the projection of 3D objects and entities to the work plane.

By default, you get the projection for all 3D objects that were generated by the features preceding the work plane feature. To select a subset of these objects or to select entities, set the `project` property to the appropriate entity level, and use the property `input` to select the 3D objects or entities.

The following properties are available.

TABLE 3-67: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`absrepairtool`		`...geom(<tag>).absRepairTol()`	Absolute repair tolerance.
`input`	Selection		Entities to project. Used when project is not `all`.
`project`	`all` | `obj` | `dom` | `bnd` | `edg` | `vtx`	`all`	Project all objects or selected objects or entities
`projectiontype`	`edgvtx` | `outline` | `all`	`outline`	Projection type. Used when project is `all`, `dom`, or `bnd`.
`repairtol`	double	`...geom(<tag>).repairTol()`	Relative repair tolerance, relative to size of each input object.
`repairtoltype`	`auto` | `relative` | `absolute`	`...geom(<tag>).repairTolType()`	Repair tolerance type: `automatic`, `relative`, or `absolute`.
`workplane`	`String`		Work plane to project onto.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `bnd` | `pnt` | `off`	`bnd`	Show selections, if selresult is on, in physics, materials, and so on; or in 3D from a plane geometry. obj is not available in a component's geometry.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The following attributes are available:

TABLE 3-68: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

SEE ALSO

`CrossSection`, `WorkPlane`

ProjectToFaces

Project edges onto faces in 3D to create imprints.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"ProjectToFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"ProjectToFaces")
```

creates a `ProjectToFaces` feature.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("target")` to select the faces to project to. The default selection is empty.

Use `model.component(<ctag>).geom(<tag>).feature(<ftag>).selection("tool")` to select the tool edges or faces. The default selection is empty.

TABLE 3-69: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`target`	Selection		Faces to project to
`imprint`	`on` | `off`	`on`	Imprint projected entities on target faces.
`tool`	Selection		Entities to project
`projdir`	`perptotarget` | `vector` | `edge` | `perspective`	`perptotarget`	Projection direction
`dirvector`	double[3]	`0,0,0`	Direction vector used when `projdir` is `vector`
`edge`	Selection		Straight edge used when `projdir` is `edge`
`eyevertex`	Selection		Vertex for eye position used when `projdir` is `perspective`
`maxdist`	double	`Inf`	Maximum distance for projection
`repairtoltype`	`auto` | `relative` | `absolute`	`auto`	Repair tolerance
`repairtol`	`double`		Relative repair tolerance used when `repairtoltype` is `relative`
`absrepairtol`	`double`		Absolute repair tolerance used when `repairtoltype` is `absolute`
`extendedges`	`on` | `off`	`off`	Extend projected edges.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.

TABLE 3-69: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`color`	`none` | `custom` | integer between 1 and the number of colors in the current theme	`none`	The color of the selection, either given as an integer indicating a color in the color theme, or as a custom color in the `customcolor` property. Coloring is only available when `selresult` is active.
`customcolor`	RGB-triplet	`Next available theme color`	The color to use. Active when `color` is set to `custom`.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.
`propagatesel`	`on` | `off`	`on`	Propagate selections from input objects to resulting objects.

For information about the `selresult`, `selresultshow` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

The following attributes are available:

TABLE 3-70: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

QuadraticBezier

The following additional properties are available with the Design Module.

TABLE 3-71: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`pconstr`	String[3]	`{off, off, off}`	Constrain the control points. Constrained properties cannot be modified by constraint and dimension functions.

Radius

PURPOSE

Constrain a circular edge to have a given radius.

DESCRIPTION

TABLE 3-72: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Circular edge
`radius`	double	0	Radius
`radiusconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (`on`) or measuring dimension (`off`)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if `createpar` is `on`
`arrowangdispl`	double	0	Angular displacement of arrow symbol relative to middle of edge
`labelradius`	double	`0.5`	Relative label position along arrow symbol

SEE ALSO

`EqualRadius`

Rectangle

The following additional properties are available with the Design Module.

TABLE 3-73: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`sizeconstr`	String[2]	`{off, off}`	Constrain the width and the height. Constrained properties cannot be modified by constraint and dimension functions.
`posconstr`	String[2]	`{off, off}`	Constrain the position. Constrained properties cannot be modified by constraint and dimension functions.

TABLE 3-73: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`rotconstr`	`on` | `off`	`off`	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.
`arrowdispl`	double[2]	Empty	Displacement of arrow symbols in the normal direction.
`labelpos`	double[2]	`{0.5, 0.5}`	Relative label position along arrow symbols.
`arrowint`	String[2]	`{on, on}`	Internal or external arrow symbols.

Repair

Repair CAD objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Repair");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property)
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Repair")
```

creates a repair feature tagged *<ftag>*. The following properties are available.

TABLE 3-74: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Names of input objects.
`check`	`on` | `off`	`on`	Check the input objects for errors.
`deletedetails`	`on` | `off`	`on`	Delete small geometric details from the geometry.

TABLE 3-74: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
fixerrors	on | off	on	Attempt to repair errors.
healedges	on | off	off	Attempt to replace tolerant edges with accurately calculated edges.
minimizetol	on | off	off	Attempt to reduce edge and vertex tolerances.
repairtol	double	1e-5	Absolute repair tolerance
selresult	on | off	off	Create selections of all resulting objects.
selresultshow	all | obj | dom | bnd | edg | pnt | off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
simplify	on | off	off	Simplify the underlying curve and surface manifolds of geometric entities
repairfacetoface	on | off	off	Repair face-to-face inconsistencies in solid objects
contributeto	String	none	Tag of cumulative selection to contribute to.

The function tries to remove or repair the following geometric and topological errors:

- Entities with invalid sense
- Invalid edge and vertex tolerances
- Invalid manifolds
- Self-intersecting manifolds
- Non-G1 manifolds
- Missing edge or vertex manifolds
- Missing vertex
- Vertices not on curve of edge
- Edges and vertices not on surface of face
- Surface self-intersections that lie outside the face

- Edge intersections that have no vertex
- Removal of discontinuities by either splitting or smoothing.

The function can also remove small geometric details including short edges, small faces, sliver faces, and spikes.

The following attributes are available:

TABLE 3-75: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-76: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	none	Output variables

EXAMPLE

The following example imports the file `repair_demo_2.x_b`, and repairs the resulting objects.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename", "repair_demo_2.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
      create("rep1","Repair");
model.component("comp1").geom("geom1").feature("rep1").
      selection("input"). set("imp1");
model.component("comp1").geom("geom1").feature("rep1").
      set("repairtol",1e-3);
model.component("comp1").geom("geom1").runAll();
```

SEE ALSO

`Check`, `Knit`

ReplaceFaces

Delete faces from CAD objects and heal the wounds by creating new faces.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"ReplaceFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);

model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces").
      selection(property)
model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces").
      delete(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("ReplaceFaces").delete(<ftag>)` creates a `ReplaceFaces` feature tagged *<ftag>* with the specified properties. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

It is also possible to create a `ReplaceFaces` feature using the standard `create` method.

TABLE 3-77: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Faces to replace.
`heal`	`cap` | `extend`	`extend`	Healing method.
`throughhole`	`on` | `off`	`off`	Heal as if the removed faces are a through hole.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.

TABLE 3-77: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The faces in the property input are deleted from their objects. The resulting object is healed so that a solid object is obtained. If heal is `cap`, a new face is formed based on the surrounding edges of each wound. If heal is `extend`, the surrounding faces of each wound are grown or shrunk to heal the wound.

When you replacing faces that form through holes, set the `throughhole` property to `on` to indicate that the two wounds from where the hole entered and exited the geometry are to be healed independently instead of as a single wound. If `throughhole` is `off`, the wound would be healed with a single new face that would just recreate the hole.

EXAMPLE

The following example imports the file `defeaturing_demo_2.mphbin`, and removes a hole from the geometry model.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
      create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
      set("filename","defeaturing_demo_2.mphbin");
model.component("comp1").geom("geom1").run("imp1");
model.component("comp1").geom("geom1").feature().
      create("rfa1","ReplaceFaces");
model.component("comp1").geom("geom1").feature("rfa1").
      selection("input").set("imp1",6,7,8,9,11,12,13);
model.component("comp1").geom("geom1").run();
```

COMPATIBILITY

The following property is no longer supported:

TABLE 3-78: OBSOLETE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`Out`	`stx` | `ftx` | `ctx` | `ptx`	`none`	Output variables

SEE ALSO

`DeleteFillets`, `DeleteSliverFaces`, `DeleteSmallFaces`, `DetachFaces`

Rotate

The following additional properties are available with the Design Module.

TABLE 3-79: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`rotconstr`	`on` | `off`	`on`	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.
`posconstr`	String[2]	`{on, on}`	Constrain the center of rotation. Constrained properties cannot be modified by constraint and dimension functions.

Scale

The following additional properties are available with the Design Module.

TABLE 3-80: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`posconstr`	String[2]	`{on, on}`	Constrain the center of scaling. Constrained properties cannot be modified by constraint and dimension functions.

Square

The following additional properties are available with the Design Module.

TABLE 3-81: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`sizeconstr`	`on` | `off`	`off`	Constrain the side length. Constrained properties cannot be modified by constraint and dimension functions.
`posconstr`	String[2]	`{off, off}`	Constrain the position. Constrained properties cannot be modified by constraint and dimension functions.
`rotconstr`	`on` | `off`	`off`	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.
`arrowdispl`	double	Empty	Displacement of arrow symbol in the normal direction.
`labelpos`	double	`0.5`	Relative label position along arrow symbol.
`arrowint`	`on` | `off`	`on`	Internal or external arrow symbol.

TangentConstraint

PURPOSE

Constrain two edges to have a point of tangency. For each edge, you can require that the tangency occurs at an adjacent vertex.

DESCRIPTION

TABLE 3-82: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge1`	Selection		First edge
`point1`	`vertex` | `anywhere`	`anywhere`	Point of tangency for first edge
`vertex1`	Selection		Vertex for point of tangency (used when `point1` is vertex)
`edge2`	Selection		Second edge
`point2`	`vertex` | `anywhere`	`anywhere`	Point of tangency for second edge
`vertex2`	Selection		Vertex for point of tangency (used when `point2` is vertex)
`helppoint1`	double[2]	`{0, 0}`	First help point (used when `point1` is anywhere)
`helppoint2`	double[2]	`{0, 0}`	Second help point (used when `point1` is anywhere)
`symbolentity`	1 | 2	1	Edge the symbol in the Graphics window is attached to
`symboledgpar`	double	0	Normalized edge parameter for positioning the symbol
`symboledgdispl`	double	20	Symbol displacement in left normal direction from edge (in pixels)

SEE ALSO

`Angle`

Thicken

Generate a solid object by thickening in the normal direction a surface object in 3D.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Thicken");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"Thicken")
```

creates a Thicken feature.

Use model.component(*<ctag>*).geom(*<tag>*).feature(*<ftag>*).selection("input") to select the objects for the thicken operation. The default selection is empty.

TABLE 3-83: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		Input objects
offset	symmetric | asymmetric	symmetric	Type of offset specification
totalthick	double	0	Total thickness (for symmetric offset)
upthick	double	0	Upside thickness (for asymmetric offset)
downthick	double	0	Downside thickness (for asymmetric offset)
direction	normal | vector	normal	Direction of the side faces
dirvector	double[3]	{0,0,1}	Direction vector for side faces
fillet	on | off	off	Fillet offset edges
selresult	on | off	off	Create selections of all resulting objects.

TABLE 3-83: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. `dom`, `bnd`, and `edg` are not available in all features.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.
`propagatesel`	`on` | `off`	`on`	Propagate selections from input objects to resulting objects.

For information about the `selresult`, `selresultshow` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

The following attributes are available:

TABLE 3-84: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

EXAMPLE

Create a cylinder by thickening a disc shaped surface:

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").geomRep("cadps");
model.component("comp1").geom("geom1").feature().
      create("wp1", "WorkPlane");
model.component("comp1").geom("geom1").feature("wp1").geom().
      create("c1", "Circle");
model.component("comp1").geom("geom1").create("thi1", "Thicken");
model.component("comp1").geom("geom1").feature("thi1").
      selection("input").set(new String[]{"wp1"});
model.component("comp1").geom("geom1").feature("thi1").
      set("totalthick", "0.1");
model.component("comp1").geom("geom1").feature("thi1").
      set("direction", "vector");
model.component("comp1").geom("geom1").feature("thi1").
      set("dirvector", new String[]{"1", "0", "1"});
```

```
model.component("comp1").geom("geom1").run();
```

SEE ALSO

Midsurface

TotalEdgeLength

PURPOSE

Constrain a set of edges to have a given total length. The edges must form a chain and all lie on the same line, circle, or spline.

DESCRIPTION

TABLE 3-85: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Edges
`length`	double	0	Total length of edges
`lengthconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if `createpar` is on
`labeldispl`	double	Empty	Displacement of the label in the Graphics window in left normal direction
`labelpos`	double	`0.5`	Relative position of the label in the Graphics window along edges

SEE ALSO

Distance

PURPOSE

Apply a linear transform (consisting of displacement, rotation, and isotropic scaling) to faces of 3D objects.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"TransformFaces");
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      selection(property);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      set(property,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getType(property)
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      setAttribute(attribute,<value>);
model.component(<ctag>).geom(<tag>).feature(<ftag>).
      getAttribute(attribute);
```

DESCRIPTION

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"TransformFaces")
```

creates a transform faces feature tagged `<ftag>`. The following properties are available.

TABLE 3-86: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`face`	Selection	Empty	Faces to transform
`keep`	`on` | `off`	`off`	Keep input objects
`workplanesrc`	`this` | Part Instance feature tag	`this`	Part Instance feature to take the work plane from
`workplane`	`xyplane` | work plane feature tag	`xyplane`	Work Plane feature that defines the coordinate system. The default, `xyplane`, is the global Cartesian coordinate system
`center`	double[3]	`{0,0,0}`	Point on the axis of rotation, and the center point of scaling
`displ`	double[3]	`{0,0,0}`	Displacement
`specify`	`axis` | `eulerang` | `edge`	`axis`	Specify an axis of rotation, Euler angles (Z-X-Z), or a straight edge

TABLE 3-86: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`axistype`	`x` | `y` | `z` | `cartesian` | `spherical`	`z`	Coordinate system used for axis. Used if `specify` is set to `axis`
`ax3`	double[3]	`{0,0,1}`	Axis vector, used if `axistype` is `cartesian`
`ax2`	double[2]	`{0,0}`	Spherical angles theta and phi, used if `axistype` is `spherical`
`axis`	double	`{0,0,1}`	Rotation axis. Vector has length 3 if `axistype` is `cartesian`, and length 2 if `axistype` is `spherical`. Alias for ax2 and ax3
`eulerang`	double[3]	`{0,0,0}`	Intrinsic Z-X-Z Euler angles α, β, and γ, used if specify is set to `eulerang`
`edge`	Selection		Edge selection. Used when `specify` is set to `edge`
`rot`	double	`0`	Rotation angle. Used if `specify` is set to `axis` or `edge`
`factor`	double	`1`	Scale factor
`stepfaces`	`yes` | `contang` | `no`	`contang`	Create step faces
`mergestepfaces`	`on` | `off`	`off`	Merge step faces with adjacent face. Only used when `stepfaces` is yes
`exceptendfacesmerged`	`on` | `off`	`off`	Except if faces at ends of edge can be merged. Only used when `stepfaces` is `contang`
`perpstepedges`	`on` | `off`	`off`	Create perpendicular step edges for surface objects
`overflow`	`auto` | `extendtransformed` | `extendother` | `capfaces` | `error`	`auto`	Overflow handling
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects

TABLE 3-86: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selresultshow`	`all` | `obj` | `dom` | `bnd` | `edg` | `pnt` | `off`	`dom`	Show selections, if `selresult` is `on`, of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry
`color`	`none` | `custom` | integer between 1 and the number of colors in the current theme	`none`	The color of the selection, either given as an integer indicating a color in the color theme, or as a custom color in the `customcolor` property. Coloring is only available when `selresult` is active
`customcolor`	RGB-triplet	Next available theme color	The color to use. Active when `color` is set to `custom`
`contributeto`	String	`none`	Tag of cumulative selection to contribute to
`propagatesel`	`on` | `off`	`on`	Propagate selections from input objects to resulting objects.

The following attributes are available:

TABLE 3-87: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
`construction`	`on` | `off` | `inherit`	`inherit`	Designate the resulting objects as construction geometry. Use `inherit` to set the construction geometry attribute only if all input objects are construction geometry.

SEE ALSO

`OffsetFaces`

Vertical

PURPOSE

Constrains straight edges to be parallel to the y-axis.

DESCRIPTION

TABLE 3-88: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Straight edges
`symboledg`	Selection		Edge the symbol in the Graphics window is attached to
`symboledgpar`	double	`0.5`	Normalized edge parameter for positioning the symbol
`symboledgdispl`	double	20	Symbol displacement in left normal direction from edge (in pixels)

SEE ALSO

`Angle`, `Horizontal`, `Parallel`

XDistance

PURPOSE

Constrain the distance in the *x* direction between two geometric entities to a given value.

DESCRIPTION

TABLE 3-89: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or curved edge)
`entity2`	Selection		Second entity (vertex or curved edge)
`distance`	double	`0`	Distance
`distanceconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if createpar is on
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	Empty	Displacement of arrow symbol in the Graphics window in the normal direction
`labelpos`	double	`0.5`	Relative label position along arrow symbol
`arrowint`	`on` | `off`	`on`	Internal or external arrow symbol

SEE ALSO

`YDistance`, `DirectedDistance`, `Distance`

YDistance

PURPOSE

Constrain the distance in the y direction between two geometric entities to a given value.

DESCRIPTION

TABLE 3-90: AVAILABLE PROPERTIES.

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or curved edge)
`entity2`	Selection		Second entity (vertex or curved edge)
`distance`	double	`0`	Distance
`distanceconstr`	`on` | `off`	`on`	Determine whether the dimension is a constraining dimension (on) or measuring dimension (off)
`createpar`	`on` | `off`	`off`	Generate parameter from dimension
`parname`	String	`""`	Specify parameter name if `createpar` is on
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	Empty	Displacement of arrow symbol in the Graphics window in the normal direction
`labelpos`	double	`0.5`	Relative label position along arrow symbol
`arrowint`	`on` | `off`	`on`	Internal or external arrow symbol

SEE ALSO

`XDistance`, `DirectedDistance`, `Distance`

Index