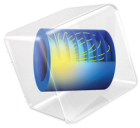

Design Module

User's Guide

5.5

COMSOL

Design Module User's Guide

Version: COMSOL 5.5

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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

GEOMETRY FEATURE	ICON	DESCRIPTION
Directed Distance		Constrain the distance between two geometric entities in a given direction
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
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
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
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
Chamfer		Create a bevel on selected edges
Convert to COMSOL		Convert to the COMSOL kernel representation
Delete Faces		Delete and replace faces
Delete Fillets		Find and delete fillets
Delete Holes		Find and delete holes
Delete Short Edges		Find and delete short edges
Delete Sliver Faces		Find and delete sliver faces

GEOMETRY FEATURE	ICON	DESCRIPTION
Delete Small Faces		Find and delete small faces
Delete Spikes		Find and delete spikes from faces
Detach Faces		Detach faces and form a new object from them
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
Repair		Repair and removal of small details
Thicken		Create a solid by offsetting selected surfaces

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. • On 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. • On 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

- From the **Home** toolbar, **Windows** menu, click ( ) **Applications Libraries**.
- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select ( ) **Update COMSOL Application Library**.

Mac

Linux

Select **Application Libraries** from the main **File>** or **Windows>** menus.

To include the latest versions of model examples, from the **Help** menu select ( ) **Update COMSOL Application Library**.

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

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2

Geometry Tools and Features

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

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 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

To 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**. 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 therefor not recommended.

To enable constraints and dimensions by default in new models make sure that the preference check box **Geometry>Use constraints and dimensions>In new models** is selected.

When you open an existing model, the constraint and dimension features in the model are normally active. To change this behavior clear the preference check box **Geometry>Use constraints and dimensions>When opening a model that uses constraints and dimensions**. 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
- Part Instance
- Partition Objects, Partition Domains, and Partition Edges
- Tangent

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 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 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 either the toolbar buttons from the **Sketch** toolbar, 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 non-sketch 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 on the **Sketch** toolbar 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 afterwards to adjust its position. To select a different type of constraint from the one suggested, click the button of another constraint 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.

- Click a button (other than **Constraint**) from the **Constraint** group of the **Sketch** toolbar, to select a specific constraint. If you have already selected a sufficient number of entities before clicking the button for 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 on 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 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 afterwards to adjust its position. The smart dimension mode remains active after each applied dimension until you click the **Dimension** button again.

- Click a button (other than **Dimension**) from the **Dimensions** group of the **Sketch** toolbar, to select a specific dimension. If you have already selected a sufficient number of entities before clicking the button for 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 upon.
- 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 and their associated geometric entities are highlighted in turquoise.
- 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 an artichoke green 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, i.e. 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 this state the geometric entities are colored magenta in the Graphics window.

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.
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 an error 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.

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, the help points are based on the position of the mouse click for the edge selection. 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** check box 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** check box 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.
- **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**.
- **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/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 applied to two straight edges, the distance dimension constrains the edges to be parallel. For singe 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.
- **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/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 section:

- **Edge Selection**: Change the selected entities.
- **Graphics**: Edit the label displayed with the arrow symbol for the constraint.

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.

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.

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 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.

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.
- **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 an 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.
- **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 ( ) on 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 models**.

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**.

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** check box 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.

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 above mentioned 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** ( ). In the **Import** section of the Settings window, select **3D CAD file** from the **Geometry import** 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 re-imported when you click a build button.

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,	.sat, .sab	up to 2019 1.0
AutoCAD®	1, 2	.dwg, .dxf	2.5-2019
CATIA® V5	2, 3	.CATPart, .CATProduct	R8 to R2019
PTC® Creo® Parametric™	1	.prt, .asm	1.0-6.0
IGES	1	.igs, .iges	up to 5.3
Inventor® assembly	1, 2	.iam	11, 2008-2019
Inventor® part	1, 2	.ipt	6 to 11, 2008-2019
NX®	1, 4	.prt	up to 1847

TABLE 2-3: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
Parasolid®	1	.x_t, .x_b	up to V32.0
PTC® Pro/ENGINEER®	1	.prt, .asm	16 to Wildfire 5
SOLIDWORKS®	1, 2, 5	.sldprt, .sldasm	98-2019
STEP	1	.step, .stp	AP203E1, AP214

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

Note 2: Available only on a supported Windows operating system.

Note 3: 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 4: Support for the NX® file format is available only on a supported Windows or Linux operating system.

Note 5: Embedded parts in assemblies are not supported. To import 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 re-imported 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 re-importing 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 re-importing 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.

OBJECTS TO IMPORT

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

If the **Surfaces** check box 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.
- Choose **Knit surfaces** 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 surfaces** options select the **Fill holes** check box to generate new faces to replace missing geometry.

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

IMPORT OPTIONS

The **Absolute import tolerance** is a length measured in the geometry's unit after the import. When importing 3D CAD files, the program merges geometric entities with a distance smaller that this tolerance.

If you select the **Check imported objects for errors** check box, a warning appears if the imported objects contain errors.

If you select the **Repair imported objects** check box, the software tries to repair defects and remove details smaller than the **Absolute import tolerance**.

If you select the **Remove redundant edges and vertices** check box, the software tries to remove redundant edges and vertices.

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** check box 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** check box 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*, etc., 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** check box above the table.
- **Keep**: Select the check box in this column to make the selection available in selection lists for subsequent nodes in the geometry sequence.
- **Physics**: Select the check box 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.

Exporting Objects to 3D CAD Formats

With a license for the Design Module you can export 3D geometry objects to the ACIS®, IGES, Parasolid®, and STEP formats. To do this:

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

Then, in 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, then click **Save** to close the Export Geometry window.

Next, select **Export selected objects** to export only chosen geometry objects or select **Export entire finalized geometry** to export the resulting geometry of a Form Union or Form Assembly operation.

Note that it is not possible to export to the formats mentioned here the result of virtual geometry operations that come after a Form Union or Form Assembly node in the geometry sequence.

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.

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 non-manifold 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 non-manifold objects are always split.

Finally, to export the geometry, click the **Export** button.

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.

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.

Using the Defeaturing Tools

This section describes the defeaturing tools for removing details from imported 3D CAD geometry. With the defeaturing tools 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.

To access the defeaturing tools, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Fillets**, **Delete Holes**, **Delete Short Edges**, **Delete Sliver Faces**, **Delete Small Faces**, **Delete Spikes**, **Delete Faces**, or **Detach Faces** from the submenu. You can also right-click the **Geometry** node and select the same options from the context menu.

When you are on the Tools window for a defeaturing operation, you can switch to another defeaturing tool by clicking one of the corresponding buttons at the top of the page. Upon completion of the defeaturing operation a corresponding feature node, which you can modify, appears in the geometry sequence.

Finding and Deleting Small Details

You can use any of the **Delete Fillets** ( ), **Delete Holes** ( ), **Delete Short Edges** ( ), **Delete Sliver Faces** ( ), **Delete Small Faces** ( ), and **Delete Spikes** ( ) tools to search for and delete details smaller than a given size. First activate the **Input objects** selection by clicking the **Active** button to toggle between and . Select the objects you want to examine in the Graphics window.

In the field **Maximum fillet radius**, **Maximum hole radius**, **Maximum edge length**, **Maximum face width**, **Maximum face size**, or **Maximum spike width**, enter the maximum size of the details you want to delete. When you click the **Find** button, a list of details that are smaller than the given size are shown in the list below. 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.

Delete Faces

The **Delete Faces** ( ) page is used to delete faces and replace them either with a new face (if **Heal method** is **Fill**) or a by growing or shrinking the adjacent faces (if **Heal method** is **Patch**). Select the faces you want to delete in the Graphics window. They appear in the **Faces to delete** list. Select the **Heal as through hole** check box if you have selected faces that make up a hole that you want to delete. When you click the **Delete Selected** button, the selected faces are deleted, 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.

Detach Faces

The **Detach Faces** ( ) page is used to detach faces from a solid object (the parent) to form a new solid object (the child). Select the faces you want to detach in the Graphics window. They appear in the **Faces to detach** list.

The **Parent heal method** list determines how to replace the detached faces in the parent object: **Fill** means that a new face is constructed, and **Patch** means that the adjacent faces are grown or shrunk to heal the wound.

The **Child heal method** list controls how to construct the child solid from the detached faces: **Fill** means that a new face is formed based on the surrounding edges of each wound, **Patch from child** means that the detached faces grow or shrink to form a solid, and **Patch from parent** 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. 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.

Geometry Features

In this section:

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 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 display in the **Bounding edges** list.

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.

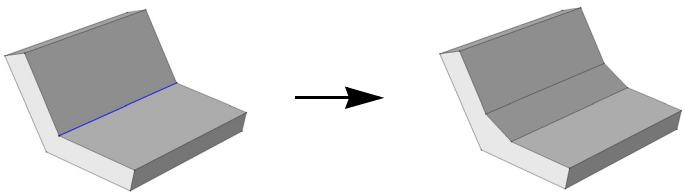
SELECTIONS OF RESULTING ENTITIES

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** check box 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.

Chamfer

To chamfer corners in 3D geometry objects, on the **Geometry** toolbar, click **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 **Active** button to toggle between turning ON and OFF the **Edges to chamfer** selections.

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 check boxes as needed.

- If the **Propagate to tangent edges** check box 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** check box 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** check box 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** check box 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 Faces

To delete and replace faces from an object, from the **Geometry** toolbar, **CAD Defeaturing** menu, select **Delete Faces** (). This opens the Delete Faces window. When the

deletion operation has been performed, you can modify it by editing the corresponding Delete Faces node that appears in the geometry branch by clicking it.

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

DELETE FACES

In the **Faces to delete** list, select the faces you want to delete. In the **Heal method** list, select the method to use for replacing the deleted faces: **Fill** means that the deleted faces are replaced with a new face, while **Patch** means that the adjacent faces are grown or shrunk to heal the wound. Select the **Heal as through hole** check box if you have selected faces that make up a hole that you want to delete.

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** check box 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 Fillets

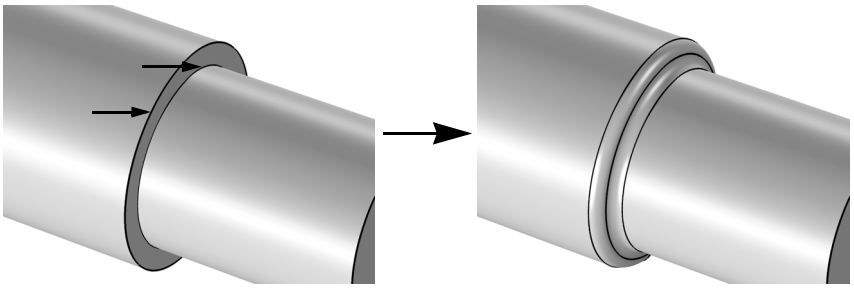
To delete fillets from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Fillets** ( ). This opens the **Delete Fillets** window, see Finding and Deleting Small Details. When the deletion operation has been performed, you can

modify it by editing the corresponding Delete Fillets node that appears in the geometry branch by clicking it.

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

Note that fillets 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 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 gap. 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.



DELETE FILLETS

In the **Input objects** list, select the objects you want to delete fillets from. In the field **Maximum fillet radius**, enter the maximum size of the fillets you want to delete. When you click the **Find Fillets** button, a list of fillets with radius smaller that the given value is shown in the **Fillet selection** list. If **Deletion type** is **All fillets**, all such fillets are deleted. You can delete a subset of these fillets by clicking 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** check box 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

To delete cylindrical holes from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Holes** (). This opens the **Delete Holes** window, see Finding and Deleting Small Details. When the deletion operation has been performed, you can modify it by editing the corresponding Delete Fillets node that appears in the geometry branch by clicking it.

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

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.

DELETE HOLES

In the **Input objects** list, select the objects you want to delete holes from. In the field **Maximum hole radius**, enter the maximum size of the holes you want to delete. When you click the **Find Holes** button, a list of holes with radius smaller that the given value is shown in the **Hole selection** list. If **Deletion type** is **All holes**, all such holes are deleted. You can delete a subset of these holes by clicking 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** check box 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

To delete short edges from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Short Edges** ( ). This opens the **Delete Short Edges** window, see Finding and Deleting Small Details. When the deletion operation has been performed, you can modify it by editing the corresponding **Delete Short Edges** node that appears in the geometry branch by clicking it.

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

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.

DELETE SHORT EDGES

In the **Input objects** list, select the objects you want to delete short edges from. 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. If **Deletion type** is **All short edges**, all such edges are deleted. You can delete a subset of these edges by clicking 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** check box 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

To delete sliver faces from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Sliver Faces** ( ). This opens the **Delete Sliver Faces** window, see Finding and Deleting Small Details. When the deletion operation has been performed, you can modify it by editing the corresponding Delete Sliver Faces node that appears in the geometry branch by clicking it.

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

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.

DELETE SLIVER FACES

In the **Input objects** list, select the objects you want to delete sliver faces from. 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. If **Deletion type** is **All sliver faces**, all such faces are deleted. You can delete a subset of these faces by clicking 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** check box 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

To delete small faces from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Small Faces** ( ). This opens the **Delete Small Faces** window, see Finding and Deleting Small Details. When the deletion operation has been performed, you can modify it by editing the corresponding Delete Small Faces node that appears in the geometry branch by clicking it.

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

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.

DELETE SMALL FACES

In the **Input objects** list, select the objects you want to delete small faces from. 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. If **Deletion type** is **All small faces**, all such faces are deleted. You can delete a subset of these faces by clicking 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** check box 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. To delete spikes from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Spikes** ( ). This opens the **Delete Spikes** window, see Finding and Deleting Small Details. When the deletion operation has been performed,

you can modify it by editing the corresponding **Delete Spikes** node that appears in the geometry branch by clicking it.

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

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.

DELETE SPIKES

In the **Input objects** list, select the objects you want to delete spikes from. 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. If **Deletion type** is **All spikes**, all such spikes are deleted. You can delete a subset of these spikes by clicking 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** check box 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

To detach faces from an object (the parent) and form a new object (the child), from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Detach Faces** ( ). This opens the Detach Faces window. When the detach operation has been performed, you can modify it by editing the corresponding **Detach Faces** node that appears in the geometry branch by clicking it.

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

DETACH FACES

Select the faces you want to detach in the **Graphics** window. They appear in the **Faces to detach** list.

The **Parent heal method** list determines how to replace the detached faces in the parent object: **Fill** means that a new face is constructed, and **Patch** means that the adjacent faces grow or shrink to heal the wound.

The **Child heal method** list controls how to construct the child solid from the detached faces: **Fill** means that a new face is formed based on the surrounding edges of each wound, **Patch from child** means that the detached faces are grown or shrunk to form a solid, and **Patch from parent** means that the parent faces surrounding the detached faces are grown or shrunk to form a solid together with the detached 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*).

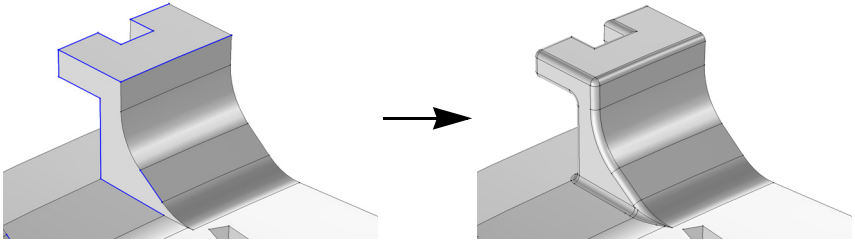
Select the **Resulting objects selection** check box to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, 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.

Fillet

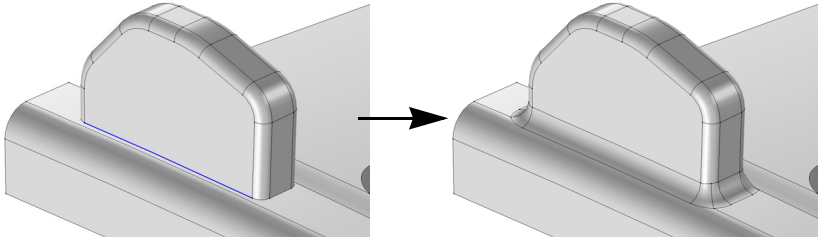
To fillet corners in 3D geometry objects, on the **Geometry** toolbar, click **Fillet** ( ). You can also right-click the **Geometry** node and add this node from the context menu.

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. When you build the feature, these adjacent faces are shrunk and a rolling ball fillet face is inserted in between. The new surface is generated by rolling a ball of the given radius so that it is tangent to both faces. The fillet face is therefore tangent to both original faces.



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 **Active** button to toggle between turning ON and OFF the **Edges to fillet** selections.

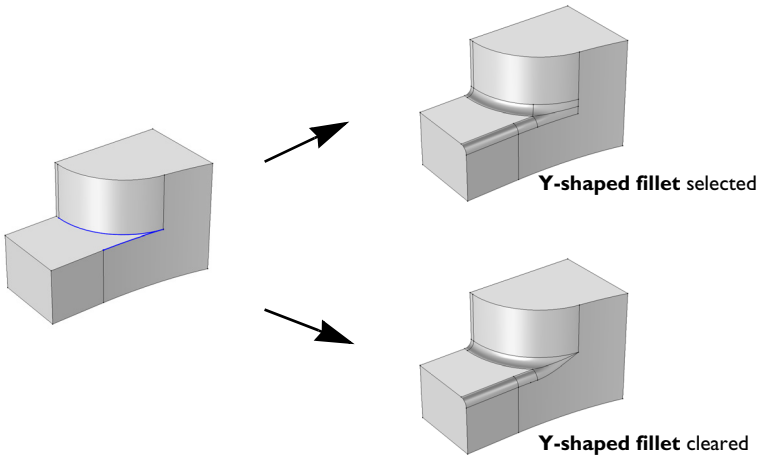
RADIUS

Enter the **Radius** of the fillet. 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.

OPTIONS

Select or clear the following check boxes as needed.

- If the **Propagate to tangent edges** check box is selected, the fillet is propagated to edges that have continuous tangent to the edges selected in **Edges to fillet**.
- If the **Preserve overlapped entities** check box 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** check box to get a *y*-shaped fillet at a vertex where three or more edges meet and there are two fillet surfaces of different convexity. In some cases, using this option is necessary for the operation to succeed.



- Select the **Fillet sharp edges at vertices** check box to get a smooth fillet surface at vertices where two filleted edges intersect at an angle.

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** check box 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.

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.

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

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** check box 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.

Loft

To create a lofted object from a set of profiles in 3D, on 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 check boxes as needed.

- Select the **Periodic loft** check box to create a periodic loft, for which the start and end profiles coincide.
- If the **Unite with input objects** check box 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.

- If the **Keep intermediate profile faces** check box 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.

Select the objects that you want to use as profiles in the **Graphics** window. 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. The selected objects appear in the **Profile objects** list. If the geometry sequence includes user-defined selections above the **Loft** node, choose **Manual** to select objects, or choose one of the selection nodes from the list next to **Profile objects**.

Click the **Active** button to toggle between turning ON and OFF the **Profile objects** selection.

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. 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 **Input objects**.

Click the **Active** 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.

Select the objects that you want to use as guides in the **Graphics** window. You can select a set of curve objects. In the non-periodic 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. The objects appear in the **Guide objects** list. If the geometry sequence includes user-defined

selections above the **Loft** node, choose **Manual** to select objects, or choose one of the selection nodes from the list next to **Guide objects**.

Click the **Active** button to toggle between turning ON and OFF the **Guide objects** selection.

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 **Active** 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** check box 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.

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.

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

INPUT

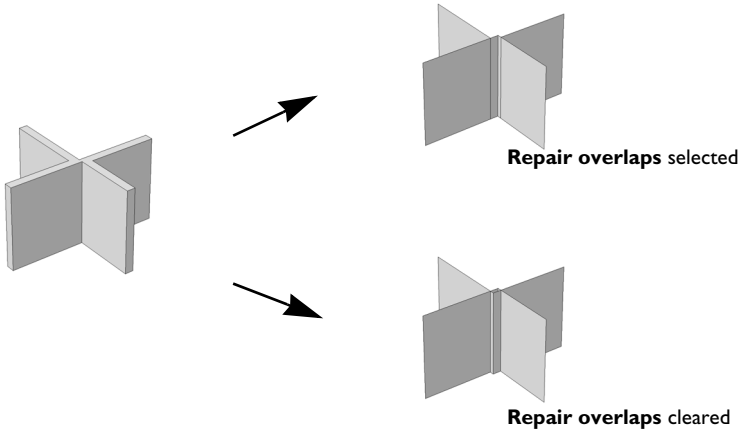
Select the objects that you want to use as input in the **Graphics** window. The objects appear in the **Input objects** list. A midsurface object is generated for each input object independently. If the geometry sequence includes user-defined selections above the **Midsurface** node, choose **Manual** to select objects, or choose one of the selection nodes from the list next to **Input objects**.

Click the **Active** button to toggle between turning ON and OFF the **Input objects** selection.

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

OPTIONS

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



Click to select the **Split in smooth components** check box 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** check box 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.

Repair

To repair objects, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Repair** ().

REPAIR

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

The software tries to repair defects and remove details smaller than the **Absolute import tolerance**. More precisely:

- 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
- Removal of surface self-intersections that lie outside the face
- Splitting at edge intersections which have no vertex
- Removal of discontinuities by either splitting or smoothing
- Remove small features (short edges, small faces, sliver faces, and spikes)

Select the option **Simplify curves and surfaces** to also simplify the underlying curve and surface manifolds of the geometric entities. Repairing objects with this option may help in some cases when Boolean operations on the objects fail.

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** check box 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.

Thicken

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

INPUT

Select the objects that you want to thicken in the **Graphics** window. The objects appear in the **Input objects** list. Each input object is thickened independently. If the geometry sequence includes user-defined selections above the **Thicken** node, choose **Manual** to select objects, or choose one of the selection nodes from the list next to **Input objects**.

Click the **Active** button to toggle between turning ON and OFF the **Input objects** selections.

Select the **Keep input objects** check box 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** check box 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 a **Direction** — **Normal** (the default) or **Vector**. For **Vector**, and based on space dimension, enter values or expressions for **x**, **y**, and **z** (SI unit: m)

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** check box 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.

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, 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("DeleteFaces");
model.component(<ctag>).geom(<tag>).defeaturing("DetachFaces");
```

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`, `DeleteFaces`, 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

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);
```

To find the details that are smaller than the given size, 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 nth detail, type

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

To get the object that contains the nth 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 non-empty 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 — Delete Faces

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

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

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

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>);
```

Summary of Commands

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

Commands Grouped by Function

Commands for Creating and Modifying Geometry in 2D

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
`Parallel`	Constrain straight edges to be parallel
`Perpendicular`	Constrain two straight edges to be perpendicular
`TangentConstraint`	Constrain two edges to be tangent

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
`DeleteFaces`	Delete faces from CAD objects and heal the wounds
`DeleteFillets`	Find and delete fillets in CAD objects

FUNCTION	PURPOSE
`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
`Export, ExportFinal`	Detach faces from CAD objects to form a new solid

Commands for File Import, Export, Conversion and Repair

FUNCTION	PURPOSE
`ConvertToCOMSOL`	Convert CAD Import Module geometry objects to COMSOL objects
`Export, ExportFinal`	Export geometry objects to a 3D CAD file
`Import`	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 Geometry Creation and Modification

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
`Midsurface`	Generate a surface object that is the midsurface of a solid object in 3D
`Thicken`	Generate a solid object by thickening a surface object in 3D

Commands in Alphabetical Order

Angle

PURPOSE

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
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowradius`	double	`NaN`	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);
```

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.
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.

Chamfer

The following additional properties are available with the Design Module.

TABLE 3-5: 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);
```

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 3-6: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Edges to chamfer.
`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.
`createselection`	`on` | `off`	`off`	Create selections.
`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 `createselection` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

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

Circle

The following additional properties are available with the Design Module.

TABLE 3-7: 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.
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 3-8: 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 3-9: 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 3-10: 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 Import Module geometry objects to COMSOL objects.

SYNTAX

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

DESCRIPTION

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

creates a `ConvertToCOMSOL` feature.

TABLE 3-11: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	Selection		Names of input objects.
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.

TABLE 3-11: 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.

SEE ALSO

Import

Copy

The following additional properties are available with the Design Module.

TABLE 3-12: 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 3-13: 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.

DeleteFaces

Delete faces from CAD objects and heal the wounds.

SYNTAX

```
model.component(<ctag>).geom(<tag>).feature().
      create(<ftag>,"DeleteFaces");
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("DeleteFaces").
      selection(property)
model.component(<ctag>).geom(<tag>).defeaturing("DeleteFaces").
      set(property,<value>);
model.component(<ctag>).geom(<tag>).defeaturing("DeleteFaces").dele
  te(<ftag>);
```

DESCRIPTION

`model.component(<ctag>).geom(<tag>).defeaturing("DeleteFaces").delete(<ftag>)` creates a `DeleteFaces` 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 `DeleteFaces` feature using the standard `create` method.

TABLE 3-14: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`input`	`Selection`		Faces to delete.
`heal`	`fill` | `patch`	`patch`	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 3-14: 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 `fill`, a new face is formed based on the surrounding edges of each wound. If heal is `patch`, the surrounding faces of each wound are grown or shrunk to heal the wound.

When you delete 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("dfa1","DeleteFaces");
model.component("comp1").geom("geom1").feature("dfa1").
      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 3-15: OBSOLETE PROPERTIES

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

SEE ALSO

DeleteFillets, DeleteSliverFaces, DeleteSmallFaces, Export, ExportFinal

DeleteFillets

Find and delete fillets in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-16: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all fillets of given size, or a selection. Only available for the feature.
`entsize`	`double`	`1e-3`	Maximum fillet radius.
`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.geom(`*`<tag>`*`).feature(`*`<ftag>`*`).find()` searches the input objects for fillets with radius less than `entsize`.

`model.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.geom(`*`<tag>`*`).defeaturing("Fillets")` has 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 3-17: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
      "defeaturing_demo_3.mphbin");
model.geom("geom1").run("imp1");
model.geom("geom1").feature().create("dfi1","DeleteFillets");
model.geom("geom1").feature("dfi1").selection("input").
      set("imp1");
model.geom("geom1").feature("dfi1").set("entsize",4e-3);
model.geom("geom1").feature("dfi1").find();
model.geom("geom1").feature("dfi1").detail().setGroup(1);
model.geom("geom1").run();
```

SEE ALSO

DeleteFaces

DeleteHoles

Find and delete holes in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-18: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`delete`	`all` | `selected`	`selected`	Delete all holes of given size, or a selection. Only available for the feature.
`entsize`	`double`	`1e-3`	Maximum hole radius.
`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.geom(<tag>).feature(<ftag>).find()` searches the input objects for holes with radius less than `entsize`.

`model.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.geom(<tag>).defeaturing("Holes")` has 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.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 holes with radius less than $4{\cdot}10^{-2}$. The first four of these holes are deleted.

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
      "defeaturing_demo_3.mphbin");
model.geom("geom1").run("imp1");
model.geom("geom1").feature().create("dho1","DeleteHoles");
model.geom("geom1").feature("dho1").selection("input").
      set("imp1");
model.geom("geom1").feature("dho1").set("entsize",4e-2);
model.geom("geom1").feature("dho1").find();
model.geom("geom1").feature("dho1").detail().setGroup(1, 2, 3, 4);
model.geom("geom1").run();
```

SEE ALSO

DeleteFaces

DeleteShortEdges

Find and delete short edges in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-20: 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
`input`	`Selection`		Names of input objects
`selresult`	`on` | `off`	`off`	Create selections of all resulting objects.

TABLE 3-20: 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.

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

`model.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.geom(<tag>).defeaturing("ShortEdges")` has 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.geom(<tag>).feature(<ftag>).problem()`.

COMPATIBILITY

The lengths of the edges are no longer returned.

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 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
     "defeaturing_demo_4.x_b");
model.geom("geom1").runAll();
```

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

DeleteSliverFaces

Find and delete sliver faces in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-22: 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.geom(`*`<tag>`*`).feature(`*`<ftag>`*`).find()` searches the input objects for faces with width less than `entsize`.

`model.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.geom(`*`<tag>`*`).defeaturing("SliverFaces")` has 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 3-23: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
     "defeaturing_demo_5.x_b");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("dsl1","DeleteSliverFaces");
model.geom("geom1").feature("dsl1").selection("input").
set("imp1");
model.geom("geom1").feature("dsl1").set("entsize",2e-3);
model.geom("geom1").feature("dsl1").find();
model.geom("geom1").feature("dsl1").detail().setGroup(1);
model.geom("geom1").runAll();
```

SEE ALSO

DeleteFaces, DeleteSmallFaces

DeleteSmallFaces

Find and delete small faces in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-24: 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.

TABLE 3-24: 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.

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

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

model.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.geom(*<tag>*).defeaturing("SmallFaces") has 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.geom(*<tag>*).feature(*<ftag>*).problem().

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_6.x_b, finds sliver faces narrower than 10^{-3}, and deletes the first of these.

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
```

```
model.geom("geom1").feature("imp1").set("filename",
     "defeaturing_demo_6.x_b");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("df1","DeleteSmallFaces");
model.geom("geom1").feature("df1").selection("input").
      set("imp1");
model.geom("geom1").feature("df1").find();
model.geom("geom1").feature("df1").detail().setGroup(1);
model.geom("geom1").run();
```

SEE ALSO

DeleteFaces, DeleteSliverFaces

DeleteSpikes

Find and delete spikes in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.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.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 3-26: 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.
`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.geom(`*`<tag>`*`).feature(`*`<ftag>`*`).find()` searches the input objects for spikes of width less than `entsize`.

`model.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.geom(`*`<tag>`*`).defeaturing("Spikes")` has 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.geom(`*`<tag>`*`).feature(`*`<ftag>`*`).problem()`.

COMPATIBILITY

The width of each spike is no longer returned.

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_7.x_b`, finds all spikes narrower than 10^{-4}, and deletes the first of these.

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

SEE ALSO

`DeleteShortEdges`, `DeleteSliverFaces`

DetachFaces

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

SYNTAX

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

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

DESCRIPTION

`model.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 3-28: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
     "defeaturing_demo_2.mphbin");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("det1","DetachFaces");
model.geom("geom1").feature("det1").selection("input").
      set("imp1",6,7,8,9,11,12,13);
model.geom("geom1").runAll();
```

COMPATIBILITY

The following property is no longer supported:

TABLE 3-29: OBSOLETE PROPERTIES

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

SEE ALSO

DeleteFaces

DirectedDistance

PURPOSE

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

DESCRIPTION

TABLE 3-30: 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
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`NaN`	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`

Distance

PURPOSE

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

DESCRIPTION

TABLE 3-31: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`entity1`	Selection		First entity (vertex or edge)
`entity2`	Selection		Second entity (vertex or edge)
`distance`	double	`0`	Distance
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`NaN`	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 3-32: 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.

TABLE 3-32: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
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 3-33: 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]	`NaN`	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 3-34: 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(<objnames>);
```

where *<objnames>* is a string array of object names. Then export them by entering

```
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.

In the above commands the file extension in the *<filename>* string determines the file format, which can be of any of the following:

TABLE 3-35: SUPPORTED FILE FORMATS

FILE FORMAT	FILE EXTENSION
Parasolid Binary (3D)	.x_b, .xmt_bin
Parasolid Text (3D)	.x_t, .xmt_txt

TABLE 3-35: SUPPORTED FILE FORMATS

FILE FORMAT	FILE EXTENSION
ACIS Binary (3D)	`.sab`
ACIS Text (3D)	`.sat`
IGES File (3D)	`.igs`, `.iges`
STEP File (3D)	`.step`, `.stp`

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`.

EXPORTING TO A PARASOLID FILE

The Parasolid text or binary file generated by the export is of version 31.0.

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 non-manifold 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.

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(<u
nit>);
```

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

Fillet

The following additional properties are available with the Design Module.

TABLE 3-36: 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.geom(<tag>).feature().create(<ftag>,"Fillet3D");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
```

DESCRIPTION

```
model.geom(<tag>).feature().create(<ftag>,"Fillet3D")
```

creates a `Fillet3D` feature.

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

TABLE 3-37: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`edge`	Selection		Edges to fillet
`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 fillet
`filletsharp`	`on` | `off`	`off`	Fillet sharp edges at vertices
`createselection`	`on` | `off`	`off`	Create selections
`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 `createselection` and `contributeto` properties, search the online help in COMSOL Multiphysics to locate and search all the documentation.

EXAMPLE

Fillet a subset of edges on a block:

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").geomRep("cadps");
model.geom("geom1").create("blk1", "Block");
model.geom("geom1").create("fil1", "Fillet3D");
model.geom("geom1").feature("fil1").selection("edge").set("blk1",
new int[]{2, 4, 6, 8});
model.geom("geom1").feature("fil1").set("radius", "0.1");
model.geom("geom1").run();
```

SEE ALSO

Chamfer3D

Import

Import geometry objects from a 3D CAD file using the CAD Import Module, Design Module, or a LiveLink product for CAD software.

SYNTAX

```
model.geom(<tag>).feature().create(<ftag>,"Import");
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
model.geom(<tag>).feature(<ftag>).importData();
```

DESCRIPTION

`model.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 3-38: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
check	on | off		Check imported objects for errors.
filename	String		Filename.
fillholes	on | off	off	Attempt to generate new faces to replace missing geometry if the property knit is solid or surface
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.
removeredundant	on | off	off	Remove redundant edges and vertices.
repair	on | off	on	Repair imported objects.
type	cad		Type of import.

TABLE 3-38: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`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.
`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 3-39: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTE	FILE EXTENSION
Autodesk Inventor	1, 3	`.ipt, .iam`
CATIA V5	2, 3	`.CATPart, .CATProduct`
IGES	1	`.igs, .iges`
Parasolid	1	`.x_t, .x_b`
PTC Pro/ENGINEER	1	`.prt, .asm`
SAT (ACIS)	1	`.sat, .sab`
SOLIDWORKS	1, 3	`.sldprt, .sldasm`
STEP	1	`.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 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 3: Only supported on Windows.

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.

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 3-40: 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.

TABLE 3-40: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`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).
`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.

TABLE 3-40: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`selcadtagbnd`	String[]	empty	Tags of boundary selections (read-only, hidden in GUI) in 3D CAD import.
`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 3-40: 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 3-40: 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).

COMPATIBILITY

The following property is no longer supported:

TABLE 3-41: OBSOLETE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`coercion`	`solid` | `face` | `off`	solid	Alias for knit. face is equivalent to surface.

SEE ALSO

`Export, ExportFinal`

InterpolationCurve

The following additional properties are available with the Design Module.

TABLE 3-42: AVAILABLE PROPERTIES

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

TABLE 3-42: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
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.geom(<tag>).feature().create(<ftag>,"Knit");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property)
```

DESCRIPTION

`model.geom(<ftag>).feature().create(<ftag>,"Knit")` creates a knit feature tagged *<ftag>*. The following properties are available.

TABLE 3-43: 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`.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-44: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
                                        "repair_demo_2.x_b");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("knit1","Knit");
model.geom("geom1").feature("knit1").selection("input").
                                     set("imp1");
model.geom("geom1").feature("knit1").set("repairtol",1e-3);
model.geom("geom1").runAll();
```

SEE ALSO

`Repair`

LineSegment

The following additional properties are available with the Design Module.

TABLE 3-45: 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.geom(<tag>).feature().create(<ftag>,"Loft");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
```

DESCRIPTION

`model.geom(<tag>).feature().create(<ftag>,"Loft")`

creates a `Loft` feature.

Use `model.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.geom(<tag>).feature(<ftag>).selection("startprofile")` to select the loft start profile. The default selection is empty.

Use `model.geom(<tag>).feature(<ftag>).selection("endprofile")` to select the loft start profile. The default selection is empty.

Use `model.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.geom(<tag>).feature(<ftag>).selection("startguide")` to select the start guide curve. The default selection is empty.

Use `model.geom(<tag>).feature(<ftag>).selection("endguide")` to select the end guide curve. The default selection is empty.

TABLE 3-46: 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.
`profile`	`Selection`		Profiles.
`startprofile`	`Selection`		Start profile selection.

TABLE 3-46: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
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.
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.
contributeto	String	none	Tag of cumulative selection to contribute to.

For information about the createselection and contributeto properties, search the online help in COMSOL Multiphysics to locate and search the documentation.

EXAMPLE

Create a cylinder from two disc profiles. This illustrates two different ways of specifying the profile.

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1", 3);
model.geom("geom1").geomRep("cadps");
model.geom("geom1").feature().create("wp1", "WorkPlane");
model.geom("geom1").feature("wp1").geom().create("c1", "Circle");
model.geom("geom1").create("copy1", "Copy");
model.geom("geom1").feature("copy1").selection("input").set(new
String[]{"wp1"});
model.geom("geom1").feature("copy1").set("displz", "1");
model.geom("geom1").create("loft1", "Loft");
model.geom("geom1").feature("loft1").selection("profile").set(new
String[]{"wp1"});
model.geom("geom1").feature("loft1").selection("startprofile").in
it(1);
model.geom("geom1").feature("loft1").selection("startprofile").se
t("copy1", new int[]{1, 2, 3, 4});
model.geom("geom1").run();
```

Midsurface

Generate a surface object that is the midsurface of a solid object in 3D.

SYNTAX

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

DESCRIPTION

```
model.geom(<tag>).feature().create(<ftag>,"Midsurface")
```

creates a `Midsurface` feature.

Use `model.geom(<tag>).feature(<ftag>).selection("input")` to select the objects for the midsurface operation. The default selection is empty.

TABLE 3-47: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`repair`	`on` | `off`	`on`	Repair overlaps
`split`	`on` | `off`	`off`	Split in smooth components
`createselection`	`on` | `off`	`off`	Create selections
`contributeto`	`String`	`none`	Cumulative selection to contribute to

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

EXAMPLE

Generate the midsurface of a thin block:

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1", 3);
model.geom("geom1").geomRep("cadps");
model.geom("geom1").create("blk1", "Block");
model.geom("geom1").feature("blk1").set("size", new String[]{"1",
"1", "0.1"});
model.geom("geom1").create("mid1", "Midsurface");
model.geom("geom1").feature("mid1").selection("input").set(new
String[]{"blk1"});
model.geom("geom1").run();
```

SEE ALSO

`Thicken`

Mirror

The following additional properties are available with the Design Module.

TABLE 3-48: 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 3-49: 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.

Parallel

PURPOSE

Constrains straight edges to be parallel.

DESCRIPTION

TABLE 3-50: 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 3-51: 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 3-52: 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 3-53: 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 3-54: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`vertex`	Selection		Vertex to fix
`pos`	double[2]	`{0, 0}`	Coordinates
`symbolvtxdispl`	double[2]	`{20, 20}`	Displacement of the symbol in the Graphics window from the vertex (in pixels)

QuadraticBezier

The following additional properties are available with the Design Module.

TABLE 3-55: 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 3-56: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
edge	Selection		Circular edge
radius	double	0	Radius
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 3-57: 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.
rotconstr	on | off	off	Constrain the rotation angle. Constrained properties cannot be modified by constraint and dimension functions.
arrowdispl	double[2]	NaN	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.geom(<tag>).feature().create(<ftag>,"Repair");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property)
```

DESCRIPTION

`model.geom(<tag>).feature().create(<ftag>,"Repair")` creates a repair feature tagged `<ftag>`. The following properties are available.

TABLE 3-58: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`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.
`simplify`	`on` | `off`	`off`	Simplify the underlying curve and surface manifolds of geometric entities
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

The function tries to remove or repair the following defects:

- 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

- Removal of surface self-intersections that lie outside the face
- Splitting at edge intersections which have no vertex
- Removal of discontinuities by either splitting or smoothing
- Remove small features (short edges, small faces, sliver faces, and spikes)

COMPATIBILITY

The following property is no longer supported:

TABLE 3-59: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
                                        "repair_demo_2.x_b");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("rep1","Repair");
model.geom("geom1").feature("rep1").selection("input").
                                    set("imp1");
model.geom("geom1").feature("rep1").set("repairtol",1e-3);
model.geom("geom1").runAll();
```

SEE ALSO

Knit

Rotate

The following additional properties are available with the Design Module.

TABLE 3-60: 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 3-61: 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 3-62: 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.

TABLE 3-62: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`arrowdispl`	double	`NaN`	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 3-63: 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 a surface object in 3D.

SYNTAX

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

DESCRIPTION

```
model.geom(<tag>).feature().create(<ftag>,"Thicken")
```

creates a `Thicken` feature.

Use `model.geom(<tag>).feature(<ftag>).selection("input")` to select the objects for the thicken operation. The default selection is empty.

TABLE 3-64: 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`	Offset direction
`dirvector`	`double[3]`	`{0,0,1}`	Offset direction vector
`fillet`	`on` | `off`	`off`	Fillet offset 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.
`contributeto`	String	`none`	Tag of cumulative selection to contribute to.

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

EXAMPLE

Create a cylinder by thickening a disc shaped surface:

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1", 3);
model.geom("geom1").geomRep("cadps");
model.geom("geom1").feature().create("wp1", "WorkPlane");
model.geom("geom1").feature("wp1").geom().create("c1", "Circle");
```

```
model.geom("geom1").create("thi1", "Thicken");
model.geom("geom1").feature("thi1").selection("input").set(new
String[]{"wp1"});
model.geom("geom1").feature("thi1").set("totalthick", "0.1");
model.geom("geom1").feature("thi1").set("direction", "vector");
model.geom("geom1").feature("thi1").set("dirvector", new
String[]{"1", "0", "1"});
model.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 3-65: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
edge	Selection		Edges
length	double	0	Total length of edges
labeldispl	double	NaN	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

XDistance

PURPOSE

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

DESCRIPTION

TABLE 3-66: 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
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`NaN`	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 3-67: 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
`helppoint1`	double[2]	`{0, 0}`	First help point
`helppoint2`	double[2]	`{0, 0}`	Second help point
`arrowdispl`	double	`NaN`	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`