



# LiveLink™ *for* Inventor®

User's Guide

# LiveLink™ for Inventor® User's Guide

© 2005–2021 COMSOL

Protected by patents listed on [www.comsol.com/patents](http://www.comsol.com/patents), and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,626,475; 8,949,089; 9,098,106; 9,146,652; 9,208,270; 9,323,503; 9,372,673; 9,454,625; 10,019,544; 10,650,177; 10,776,541, and 11,030,365. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement ([www.comsol.com/comsol-license-agreement](http://www.comsol.com/comsol-license-agreement)) and may be used or copied only under the terms of the license agreement. LiveLink™ for Inventor® is not affiliated with, endorsed by, sponsored by, or supported by Autodesk, Inc., and/or its affiliates and/or subsidiaries. Portions of this software are owned by Siemens Product Lifecycle Management Software Inc. © 1986–2021. All Rights Reserved. Portions of this software are owned by Spatial Corp. © 1989–2021. All Rights Reserved.

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

Version: COMSOL 6.0

## Contact Information

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

If you need to contact Support, an online request form is located at the COMSOL Access page at [www.comsol.com/support/case](http://www.comsol.com/support/case). Other useful links include:

- Support Center: [www.comsol.com/support](http://www.comsol.com/support)
- Product Download: [www.comsol.com/product-download](http://www.comsol.com/product-download)
- Product Updates: [www.comsol.com/support/updates](http://www.comsol.com/support/updates)
- COMSOL Blog: [www.comsol.com/blogs](http://www.comsol.com/blogs)
- Discussion Forum: [www.comsol.com/forum](http://www.comsol.com/forum)
- Events: [www.comsol.com/events](http://www.comsol.com/events)
- COMSOL Video Gallery: [www.comsol.com/videos](http://www.comsol.com/videos)
- Support Knowledge Base: [www.comsol.com/support/knowledgebase](http://www.comsol.com/support/knowledgebase)

Part number: CM022301

# C o n t e n t s

## Chapter 1: Introduction

<b>About the Product</b>	<b>8</b>
Overview of the Included Geometry Tools and Features. . . . .	8
Overview of the User's Guide. . . . .	10
Where Do I Access the Documentation and Application Libraries?. . . . .	10

## Chapter 2: The LiveLink™ Interface

<b>The LiveLink Add-In and the COMSOL Multiphysics Tab</b>	<b>17</b>
The COMSOL Parameter Selection Window . . . . .	17
The COMSOL Selections Window . . . . .	18
The COMSOL Synchronize Offline Window . . . . .	19
<b>Synchronizing the Geometry</b>	<b>21</b>
The LiveLink Node . . . . .	21
<b>Modeling with the Embedded COMSOL Modeling Environment</b>	<b>28</b>
Overview of the Embedded Environment . . . . .	28
Basic Steps to Build a Model . . . . .	29
Making Changes to the CAD Geometry . . . . .	30
Saving a Model. . . . .	31
Opening a Model. . . . .	33
<b>Connecting to COMSOL Server™ and Running Applications</b>	<b>36</b>
Overview. . . . .	36
Connecting to a COMSOL Server™ . . . . .	37
Running Applications with COMSOL Multiphysics®. . . . .	37

## Chapter 3: Geometry Tools and Features

<b>Geometry Representation</b>	<b>40</b>
Working with the CAD Kernel . . . . .	40
Converting Objects to COMSOL Kernel Representation . . . . .	41
Converting Objects to CAD Kernel Representation . . . . .	42
<b>Importing and Exporting CAD Files</b>	<b>44</b>
Importing 3D CAD Files . . . . .	44
Exporting Objects to 3D CAD Formats . . . . .	49
<b>Using the Defeaturing Tools</b>	<b>51</b>
Finding and Deleting Small Details . . . . .	51
Delete Faces . . . . .	52
Detach Faces . . . . .	52
Detect Interferences . . . . .	53
<b>Geometry Features</b>	<b>56</b>
Cap Faces . . . . .	56
Check . . . . .	57
Delete Faces . . . . .	58
Delete Fillets . . . . .	59
Delete Holes . . . . .	61
Delete Short Edges . . . . .	62
Delete Sliver Faces . . . . .	63
Delete Small Faces . . . . .	64
Delete Spikes . . . . .	65
Detach Faces . . . . .	67
Knit to Solid . . . . .	68
Projection . . . . .	69
Repair . . . . .	71

## Chapter 4: Programming and Command Reference

<b>Defeaturing Tools</b>	<b>74</b>
Defeaturing Tools — Finding and Deleting Small Details . . . . .	74
Defeaturing Tools — Delete Faces . . . . .	77
Defeaturing Tools — Detach Faces . . . . .	78
Defeaturing Tools — Detect Interferences . . . . .	78
<b>Summary of Commands</b>	<b>80</b>
<b>Commands Grouped by Function</b>	<b>81</b>
<b>Commands in Alphabetical Order</b>	<b>83</b>
CapFaces . . . . .	83
Check . . . . .	84
ConvertToCOMSOL . . . . .	85
DeleteFaces . . . . .	86
DeleteFilletts . . . . .	88
DeleteHoles . . . . .	90
DeleteShortEdges . . . . .	93
DeleteSliverFaces . . . . .	96
DeleteSmallFaces . . . . .	99
DeleteSpikes . . . . .	101
DetachFaces . . . . .	104
DetectInterferences . . . . .	107
Export, ExportFinal . . . . .	107
Import 3D CAD . . . . .	109
Knit . . . . .	117
LiveLinkInventor . . . . .	119
Projection . . . . .	122
Repair . . . . .	124



# Introduction

Welcome to the LiveLink™ *for* Inventor® 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 use geometry from the Inventor® CAD system for simulation, to import and export geometry using the most common 3D CAD file formats, and to repair, defeature, and modify geometry.

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 Product

## *Overview of the Included Geometry Tools and Features*




---

LiveLink™ for Inventor® enables modeling using 3D designs synchronized from the Autodesk® Inventor® CAD software. The included LiveLink™ interface builds on an associative transfer of the geometry from the CAD program to the COMSOL model. You can work with the two programs running side-by-side, or by taking advantage of the embedded COMSOL Multiphysics modeling environment, you can even create models without leaving the Inventor user interface.


















Through the LiveLink interface, you can modify the geometry in the CAD program by sending the name and value of a dimension or parameter to Inventor. There the geometry is updated and regenerated, before finally being transferred back to COMSOL Multiphysics. The associative transfer assures that all your settings on the geometry are retained, just as they were before the modification.

If you rather use file import to get your designs into COMSOL Multiphysics, the product also supports import of 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 dedicated geometric kernel, the *CAD kernel*, and 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
 <b>2D Geometry Features</b>		
Projection		Project 3D objects and entities to a work plane
 <b>3D Geometry Features</b>		



GEOMETRY FEATURE	ICON	DESCRIPTION
Cap Faces		Generate faces from edges to fill gaps and create solid objects, or to partition solids
Check		Check CAD objects for faults
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
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
Detect Interferences		Find interferences, such as intersections, gaps, touches and containments, between objects
Export		Export geometry objects to 3D CAD file formats
Import		Import geometry objects from 3D CAD file formats
Knit to Solid		Knit surface objects to form solid or surface object
LiveLink for Inventor		Synchronize geometry between Inventor and COMSOL
Repair		Repair and removal of small details

## Overview of the User's Guide

---

This documentation covers LiveLink™ for Inventor® 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 LiveLink™ for Inventor®*.

## 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	<p>To open the <b>Help</b> window:</p> <ul style="list-style-type: none"><li>• In the <b>Model Builder</b>, <b>Application Builder</b>, or <b>Physics Builder</b> click a node or window and then press F1.</li><li>• On any toolbar (for example, <b>Home</b>, <b>Definitions</b>, or <b>Geometry</b>), hover the mouse over a button (for example, <b>Add Physics</b> or <b>Build All</b>) and then press F1.</li><li>• From the <b>File</b> menu, click <b>Help</b> (  ).</li><li>• In the upper-right corner of the COMSOL Desktop, click the <b>Help</b>(  ) button.</li></ul>
-----	---

---


---

Mac	<p>To open the <b>Help</b> window:</p> <ul style="list-style-type: none"><li>• In the <b>Model Builder</b> or <b>Physics Builder</b> click a node or window and then press F1.</li><li>• On the main toolbar, click the <b>Help</b> (  ) button.</li><li>• From the main menu, select <b>Help&gt;Help</b>.</li></ul>
Linux	

---


### Opening the Documentation Window

---

Win	<p>To open the <b>Documentation</b> window:</p> <ul style="list-style-type: none"><li>• Press Ctrl+F1.</li><li>• From the <b>File</b> menu select <b>Help&gt;Documentation</b> (  ).</li></ul>
-----	---

---

---

Mac	<p>To open the <b>Documentation</b> window:</p> <ul style="list-style-type: none"><li>• Press Ctrl+F1.</li><li>• On the main toolbar, click the <b>Documentation</b> (  ) button.</li><li>• From the main menu, select <b>Help&gt;Documentation</b>.</li></ul>
Linux	

---

## 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 ():



- 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**.

---



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](mailto:info@comsol.com).

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to

support@comsol.com. An automatic notification and case number is sent to you by email.

#### **COMSOL WEBSITES**

COMSOL website	<a href="http://www.comsol.com">www.comsol.com</a>
Contact COMSOL	<a href="http://www.comsol.com/contact">www.comsol.com/contact</a>
COMSOL Access	<a href="http://www.comsol.com/access">www.comsol.com/access</a>
Support Center	<a href="http://www.comsol.com/support">www.comsol.com/support</a>
Product Download	<a href="http://www.comsol.com/product-download">www.comsol.com/product-download</a>
Product Updates	<a href="http://www.comsol.com/support/updates">www.comsol.com/support/updates</a>
COMSOL Blog	<a href="http://www.comsol.com/blogs">www.comsol.com/blogs</a>
Discussion Forum	<a href="http://www.comsol.com/community">www.comsol.com/community</a>
Events	<a href="http://www.comsol.com/events">www.comsol.com/events</a>
COMSOL Video Gallery	<a href="http://www.comsol.com/video">www.comsol.com/video</a>
Support Knowledge Base	<a href="http://www.comsol.com/support/knowledgebase">www.comsol.com/support/knowledgebase</a>



## The LiveLink™ Interface

When running the COMSOL Multiphysics® software and the Inventor® CAD software side-by-side you can associatively transfer geometry from the CAD software to COMSOL Multiphysics using the LiveLink™ interface. In the other direction, from COMSOL Multiphysics to the CAD software, the interface enables you to update the dimensions in the CAD file. If running the software side-by-side is not available, for example when you want to run your COMSOL Multiphysics simulation on a different platform from where your CAD software is installed, the LiveLink interface also provides a file based offline synchronization mode. After synchronization you can apply further geometry operations to prepare the synchronized geometry objects for analysis — for example, by removing details.

In an even closer mode of integration, the included One Window interface makes the windows and tools of COMSOL Multiphysics available to set up and run simulations right inside the Inventor® user interface. In this case you must perform all geometry modifications in the Inventor® part or assembly file that is associated with your model.

You may also analyze designs using simulation apps that connect to Inventor® by utilizing the LiveLink interface. With the provided tools you can easily connect to COMSOL Server™ from within Inventor® to browse and run simulation apps, including those that use geometry synchronized with the CAD software.

This section includes the following topics:

- [The LiveLink Add-In and the COMSOL Multiphysics Tab](#)
- [Synchronizing the Geometry](#)
- [Modeling with the Embedded COMSOL Modeling Environment](#)
- [Connecting to COMSOL Server™ and Running Applications](#)



# The LiveLink Add-In and the COMSOL Multiphysics Tab

When you install LiveLink™ for Inventor® an add-in is registered with Inventor® during the install process.




You can also install and register the LiveLink add-in by installing the COMSOL Client, available for download from <https://www.comsol.com>. Such an installation enables the functionality for preparing designs for synchronization, saving files for offline synchronization, and running simulation apps that synchronize geometry.

In the user interface of the CAD software, the add-in enables the COMSOL Multiphysics tab, which contains the functionality to configure the CAD design for synchronization:

- [The COMSOL Parameter Selection Window](#), where you can select parameters in Inventor® to control from the simulation
- [The COMSOL Selections Window](#), which you can use to define selections on the geometry in Inventor®
- [The COMSOL Synchronize Offline Window](#), where you can save synchronization files from Inventor®.

## *The COMSOL Parameter Selection Window*

In Inventor®, open the **COMSOL Parameter Selection** window by clicking the **Parameter Selection** button () located on the **COMSOL Multiphysics** tab. The window lists all parameters for the part or assembly, including *Model* and *User Parameters*. To link a parameter to COMSOL select the corresponding check box in the **Add to COMSOL Model** column. All types of parameters can be linked, but only parameters that are not defined by an equation formula are possible to control from a COMSOL model. Linking parameters that are defined by a formula enables using their values in COMSOL model definitions.

The list of linked parameters will be saved automatically in the Inventor<sup>®</sup> file the next time you save the file. To re-load the list of linked parameters from the saved file click the **Revert to Saved** button. Confirm your changes by clicking the **Done** button.

For additional information about parameters in CAD files see the Inventor<sup>®</sup> documentation.

### *The COMSOL Selections Window*

---

In the Inventor<sup>®</sup> user interface, click the **Selections** button on the **COMSOL Multiphysics** tab to open the **COMSOL Selections** window where you can set-up user-defined selections of geometric entities to be synchronized with the COMSOL<sup>®</sup> model.

Alternatively, to create a user-defined selection right-click a feature or component in the Inventor<sup>®</sup> model tree, or a geometrical entity in the graphics area, and choose **COMSOL Selections**. This also opens the **COMSOL Selections** window, and adds the new selection to the list of **Selections**. You can add additional items to the selection by selecting on the geometry or in the tree.

The **Selections** list contains the list of user-defined selections for the active file in Inventor<sup>®</sup>. For each selection from the list you can review and modify the entities included in the selection. These appear in the **Entities** list. You can also edit the selection name, which is displayed in the **Selection name** field.

The following is a list of geometric entities or model elements that can be selected on a Inventor<sup>®</sup> design:

- solid or surface bodies
- components that can be parts, or sub-assemblies of an assembly
- features
- entities, such as faces, edges, and points

To create a new selection click the **New Selection (+)** button. A selection can only contain entities of the same type, for example, only faces or only points. The first item added to a selections determines the type for the selection. Change the selection mode in Inventor<sup>®</sup> to select different entity types, and use the model tree to select assembly components, features and solid bodies. The toggle buttons above the **Selections** list filter which selections are displayed based on their type. To remove a selection from the **Selections** list right-click the selection, then select **Delete**. Selecting **Clear** deletes all selections from the list.

To remove an item from a selection right-click the item in the **Entities** list, then select **Delete**.

To turn off the synchronization of the material selections clear the **Auto-generate selections from materials** check box. After the next synchronization the selections based on materials will not appear in the COMSOL model.

Selections are automatically saved in the Inventor<sup>®</sup> file the next time you save the file. To re-load the selections saved in the Inventor<sup>®</sup> file click the **Revert to Saved** button.

To confirm changes and close the **COMSOL Selections** window click **Done**. This step is necessary before synchronizing between Inventor<sup>®</sup> and COMSOL to transfer the selections.

### *The COMSOL Synchronize Offline Window*

---

In the Inventor<sup>®</sup> user interface, click the **Synchronize Offline** button on the **COMSOL Multiphysics** tab to open the **COMSOL Synchronize Offline** window where you can save synchronization files that can be loaded by the LiveLink feature in a COMSOL model.

Before saving offline synchronizations files consider linking parameters for control from the COMSOL model, see [The COMSOL Parameter Selection Window](#), and configuring selections on the geometry that can make it more efficient to define physics, see [The COMSOL Selections Window](#).

From the **Synchronization** list select **Initial** (default) or **Requested**. If you select **Initial** the synchronization file will contain the current geometry. Select **Requested** to load a request file, and save a synchronization file with geometries generated for the parameter values in the request file.

#### **SETTINGS FOR INITIAL SYNCHRONIZATION**

Select to **Synchronize material data** together with the geometry. With this option the material properties defined in the synchronized part or assembly components are included in the synchronization file.

Select the type of objects to include in the synchronized geometry. Only the types of objects that are selected from the **Solids** and **Surfaces** check boxes are transferred during a synchronization. Performance can be improved for very large geometries that contain more than one type of objects, by turning off the synchronization of types of objects that are not necessary for the simulation. Note that you can also exclude selected objects from being synchronized by hiding them.

## SETTINGS FOR REQUESTED SYNCHRONIZATION

Click **Browse** to load a request file that was saved from a LiveLink feature in COMSOL Multiphysics. Once the file is loaded you can see information displayed about the requested document and the requested parameter values. Click one of the rows in the **Parameters** table to display a larger table. The fields in the **Requested Value** column will contain many values if the loaded request is for a parametric sweep, or if many requests have been saved in the request file. In the **Current Value** column you can see the values that the parameters currently have in the CAD model. An empty field in this column means that the corresponding parameter could not be found in the CAD model.

To generate a synchronization file according to the request do not change the default settings, i.e. **Synchronize with** should be set to **Requested document** and **Use parameter values** should be set to **Requested**.

To generate a synchronization file for the currently active file in Inventor choose **Active document** from the **Synchronize with** list. By setting **Use parameter values** to **Current** the synchronization file will contain the geometry built using the parameter values displayed in the **Current Value** column.

Click **Save** to save the synchronization file and close the window. This may take some time for parametric sweeps for which the geometry needs to be rebuilt many times.

For **Initial** synchronizations you can also click **Append** to add a synchronization to an already saved synchronization file. You can for example save a synchronization file, then manually change parameter values, rebuild the CAD model, then open this window again to append the new geometry to the previously saved file.

To close the window without saving click **Cancel**.

# Synchronizing the Geometry

To initiate the geometry synchronization between Inventor<sup>®</sup> and COMSOL Multiphysics<sup>®</sup> use the **LiveLink for Inventor** feature node. In case you are working in the embedded One Window modeling environment inside the Inventor<sup>®</sup> user interface, synchronization is automatic when starting a new model, or as soon as a change in the design is detected.

Before synchronization consider to review and change the settings for [The LiveLink Node](#), and to configure the synchronization of parameters and selections in the Inventor<sup>®</sup> file as described in the section [The LiveLink Add-In and the COMSOL Multiphysics Tab](#).

## *The LiveLink Node*

---

The **LiveLink for Inventor** feature, available from the **LiveLink** menu on the **Home** toolbar, synchronizes the geometry between Autodesk<sup>®</sup> Inventor<sup>®</sup> and COMSOL Multiphysics<sup>®</sup>.

For geometry synchronization to take place both COMSOL Multiphysics and Autodesk<sup>®</sup> Inventor<sup>®</sup> need to be running, and the CAD file needs to be open in the CAD software. When this is not possible, [Offline Synchronization](#) is available so that you can, from the CAD software, save synchronization files, which you can load into the LiveLink node.



A list of compatible versions of Autodesk<sup>®</sup> Inventor<sup>®</sup> can be found at: [www.comsol.com/system-requirements/module](http://www.comsol.com/system-requirements/module).

---

During synchronization the LiveLink<sup>™</sup> interface initiates a rebuild of the CAD geometry in Inventor<sup>®</sup> based on the parameter values set in the COMSOL Multiphysics<sup>®</sup> model, then associatively transfers the rebuilt geometry to the model to ensure that physics and other model settings are retained on the geometric entities where they were originally defined.

In the geometry sequence of the model the LiveLink node signifies a geometry synchronized from the CAD software, and in many aspects it is just like any other geometry operation. It can be combined with other operations that may appear both before and after the LiveLink node in the sequence. The exception to this is when

modeling in the embedded COMSOL environment inside the Inventor<sup>®</sup> user interface, where the Geometry from Inventor<sup>®</sup> node is the only allowed geometry operation, and it is automatically added to the geometry sequence as soon as a new model is created.

Synchronized geometry objects are represented using the Parasolid<sup>®</sup> geometry kernel inside COMSOL Multiphysics. Thus, you can apply all the tools and features for defeaturing and geometry modification as included with this LiveLink<sup>™</sup> product.

### THE SYNCHRONIZE SECTION

To specify which CAD document to synchronize use the **Synchronize with** list. Select **Active document** to synchronize the open and active CAD document in Inventor<sup>®</sup>. For the initial synchronization after adding a LiveLink<sup>™</sup> node, **Active document** is the only available setting. For subsequent synchronizations the **Specified document** option becomes selected instead. Using this setting the CAD document specified after **Document** will be synchronized provided that it is open in Inventor<sup>®</sup>. To synchronize a new CAD document switch to the **Active document** setting. The name of the CAD document is automatically determined during synchronization with the **Active document** setting.



When running parametric optimization studies or parametric sweeps, the CAD file needs to remain open in Inventor<sup>®</sup> until the solver completes.

---

The LiveLink interface also determines the **Table record** and the **View, Position, and Model state** representations used in the synchronized Inventor<sup>®</sup> document when **Synchronize with** is set to **Active document**. To make sure that the CAD document will be synchronized in the same state during subsequent synchronizations, the **Specified document** alternative can be used. With this option the interface automatically activates the last synchronized table record and representation. To be able to synchronize the CAD document in a different state, first make the switch to the desired table record or representation in Inventor<sup>®</sup>, then from the **Synchronize with** list select **Active document**.

For information on how to use tables to manages configurations for parts and assemblies, and how to use the different representations see the Inventor<sup>®</sup> documentation.

You can also select to **Synchronize material data** together with the geometry. With this option the interface imports the material properties defined in the synchronized part or assembly components, and the software creates corresponding **Material** nodes in the component. The input selection in the generated **Material** nodes is set to the material


selections that are also created during synchronization; see [Selections](#), for more information.

To initiate a synchronization click the **Synchronize** button. This sends parameter value pairs to Inventor<sup>®</sup>, then retrieves the regenerated geometry objects. Only visible objects are synchronized. Synchronization can also be triggered by clicking a build button when there are changed settings in the LiveLink node, or there are changed parameter values. Synchronization is automatic when running an optimization study or a parametric sweep.

Select the **Offline synchronization** check box to enable the **Offline Synchronization** section where you can load synchronization files that you have prepared in advance using the LiveLink tools in Inventor<sup>®</sup>. Such offline synchronization files can contain the geometry, selections and parameters for several synchronizations, including parameter sweeps.

#### OFFLINE SYNCHRONIZATION

Use offline synchronization when you do not have the possibility to run the CAD software and COMSOL Multiphysics on the same computer. For example, to run a parametric sweep when COMSOL Multiphysics is installed on a Linux or macOS computer, you can, in the CAD software, prepare and save in advance a synchronization file with the geometries needed for the sweep. In COMSOL Multiphysics, in the LiveLink node, you can load the synchronization file, and compute the parametric sweep.


To load a synchronization file, from the **Offline Synchronization** toolbar, click **Receive Synchronization File** (  ). Browse to the file and click open to load the geometry, parameters and selections from the file. Synchronization files may contain one or several synchronizations, including synchronizations generated in response to a requested parameter sweep.





To generate synchronization files use the functionality provided by the LiveLink add-in inside Inventor<sup>®</sup>, see [The COMSOL Synchronize Offline Window](#).

---

If the loaded synchronization file contains a parametric sweep the number of parameter tuples in the file is displayed under **Receive**. After the file is loaded, you can go to the **Study** node that contains the parametric sweep for which the synchronization was requested, and click **Compute** to run the parametric sweep.

In case a loaded synchronization file contains several synchronizations you can see the number of synchronizations left in the file under **Receive**. To load the next synchronization from the file click **Load Next Synchronization** (  ).

To request a synchronization click **Request Synchronization** (  ). You can, for example, under **Global Definitions>Parameters**, change the value of a parameter that is synchronized (see [Parameters](#)), and then generate a synchronization request. The response you will generate in Inventor<sup>®</sup> will contain the updated geometry according to the parameter value in the request. To save another request to the previously saved request file click **Append to Request File** (  ). To generate a request for a parametric sweep select one of the available sweeps from the **Parametric sweep to request** list, and then click **Request Synchronization** to save the request.

## PARAMETERS

Parameters that take part in the synchronization are specified in the **Parameters in CAD Package** section. Based on the transferred parameter and value pairs in the **Controllable parameters** table, the CAD model is automatically rebuilt in Inventor<sup>®</sup> and transferred back to COMSOL Multiphysics<sup>®</sup>. This way you can control dimensions of the CAD design, for both parts and assemblies, and their components.

Together with the updated geometry, updated values of the parameters listed in the **Read-only parameters** table are also transferred from the CAD software. Read-only parameters are parameters in Inventor<sup>®</sup> that are defined by a formula and therefore not possible to control without breaking the CAD design. However these parameters can be used to set up the simulation.

Clicking **Synchronize** also retrieves parameters that not yet appear in the tables under the **Parameter in CAD Package** section but have been selected to be linked to COMSOL<sup>®</sup> from the Inventor<sup>®</sup> file. As part of this process a corresponding global parameter is automatically generated in the model.

In both the **Controllable parameters** and **Read-only parameters** tables, the **CAD name** column holds the names of parameters in the CAD file, whereas the **COMSOL name** and **COMSOL value** columns contain the name and value, respectively, of corresponding global parameters in the model, defined under **Global Definitions>Parameters** in the model tree. Global parameters can be controlled by the parametric solver to perform parametric sweeps. During synchronization of controllable parameters COMSOL evaluates the corresponding global parameters and sends the resulting value to Inventor<sup>®</sup>. Click the symbol in the **Sync** column to turn on or off the synchronization of a parameter.




Unless a unit is specified the updated parameters are assumed to have units as defined in the Inventor® file.

### *Synchronizing Parameters*

You can type in parameters in the tables, or use the **COMSOL Parameter Selection** window in Inventor® to link parameters from the Inventor® file, for details see the section [The COMSOL Parameter Selection Window](#). Linked parameters can then be retrieved to the table, and global parameters are automatically generated for them. This process is automatic when modeling with COMSOL inside Inventor®.

To retrieve the linked parameters from the Inventor® file to the LiveLink node, and to generate corresponding global parameters in the model do one of the following:

- In the **Settings** window for **LiveLink for Inventor** click the **Synchronize** button:  
New parameters, which have been selected in the Inventor® file, but are not listed under **Parameters in CAD Package**, are transferred to the **Controllable parameters** table or the **Read-only parameters** table. For each new CAD parameter a global parameter is generated in the model. The global parameters are assigned the values of the corresponding CAD parameters. Following this, the geometry is regenerated in Inventor®, based on the parameters in the **Controllable parameters** table, and transferred to COMSOL.
- In the **Settings** window for **LiveLink for Inventor** click the **Update Parameters from CAD** button (  )  
New parameters, which have been selected in the Inventor® file, but are not listed under **Parameters in CAD Package** are transferred to the **Controllable parameters** table or the **Read-only parameters** table. For each new CAD parameter a global parameter is generated in the model. The global parameters are assigned the values of the corresponding CAD parameters. In addition, the values of global parameters, which are already linked to CAD parameters in the table, are updated to the current values of the CAD parameters.

Parameter names in the **CAD name** column appear just as in the Inventor® file, for example d1. When the parameter refers to components of the synchronized assembly the syntax also includes the name of the component, for example d1.part1.ipt. This makes it possible to control parameters from components of an assembly.

### **IMPORT OPTIONS**

In the **Length unit** list, select **From COMSOL** to scale the transferred objects to the length unit of the geometry in the current model. Select **From CAD document** to change the geometry's length unit to the unit in the CAD software.

### *Objects to Import*

CAD designs may include not only solid objects, but also surfaces and curves used during the construction of those solids. These construction objects may slow down the synchronization, and they may cause problems when combining the synchronized objects before meshing.

One way to prevent synchronization of construction surfaces and curves is to hide them in the Inventor<sup>®</sup> user interface. An alternative is to set the LiveLink interface to omit them from the synchronization. Only the types of objects that are selected from the **Solids** and **Surfaces** check boxes are transferred during a synchronization. Performance can be improved for very large geometries that contain more than one type of objects, by turning off the synchronization of types of objects that are not necessary for the simulation.

### *Import Options*

The **Absolute import tolerance** is a length measured in the geometry's unit after synchronization. The import operation merges geometric entities with a distance smaller than this tolerance.

If the **Check imported objects for errors** check box is selected, a warning appears if the transferred objects contain errors.

If the **Repair imported objects** check box is selected, the software tries to repair defects and remove details smaller than the **Absolute import tolerance** when transferring objects from Inventor<sup>®</sup>.

If the **Remove redundant edges and vertices** check box is selected, edges and vertices that are considered redundant, such as the edges of an imprint on a face, are removed during synchronization.

## **ASSOCIATIVITY**

The associativity between the geometry transferred to COMSOL Multiphysics and the geometry in Inventor<sup>®</sup> is preserved as long as the topology of the geometry does not change. If the topology changes, for example if new faces are introduced or faces are removed, the interface tries to preserve associativity for the portions of the geometry that remain unchanged.

To make sure that the geometry stays associative to the geometry in the CAD program even after saving and reopening your work in COMSOL Multiphysics and Inventor<sup>®</sup> it is recommended to save the Inventor<sup>®</sup> file in the state corresponding to the latest synchronization. To make sure that the geometry stays associative during several offline synchronizations it is recommended that you load the synchronization files in

the order that you have saved them in the CAD software, and that you define selections, see [The COMSOL Selections Window](#).

## **SELECTIONS**

The LiveLink™ interface automatically synchronizes selections for materials from the Inventor® document. The selections get their names from the names of materials assigned to objects or other geometric entities on the synchronized CAD geometry, and they appear in **Selections** tables according to the entity level they are defined on. Click on an entry in a table to see the included entities highlighted in the **Graphics** window. When setting up a simulation you can use selections for example for assigning materials on the finalized geometry. You can read about how to turn off the automatic generation of selections for materials in the section [The COMSOL Selections Window](#).

### *User Defined Selections*

In addition to automatically synchronizing selections for materials you can also define custom selections on the CAD design in the Inventor® user interface and synchronize those with the COMSOL model, for details see the section [The COMSOL Selections Window](#).

For CAD assemblies, the selections that you define in a component file are automatically added to the list of selections synchronized for the assembly. The selections loaded from the components are also correctly instantiated during synchronization. For example consider a CAD assembly that contains ten instances of a part file, which has a selection defined with a single face. After synchronizing the CAD assembly, the selection in the COMSOL Multiphysics model will contain ten faces, one for each instance of the part.

User defined selections are listed alongside material selections in the **Selections** tables, according to the entity level they are defined on. Click on a selection to see it highlighted on the geometry in the **Graphics** window.

## **ASSIGNED ATTRIBUTES**

Select the **Construction geometry** check box to make the resulting objects available only in the feature's geometry sequence. For more information see [Construction Geometry](#).

# Modeling with the Embedded COMSOL Modeling Environment

## Overview of the Embedded Environment

LiveLink™ for Inventor® enables modeling inside Inventor® by embedding the COMSOL Desktop into the Inventor user interface.

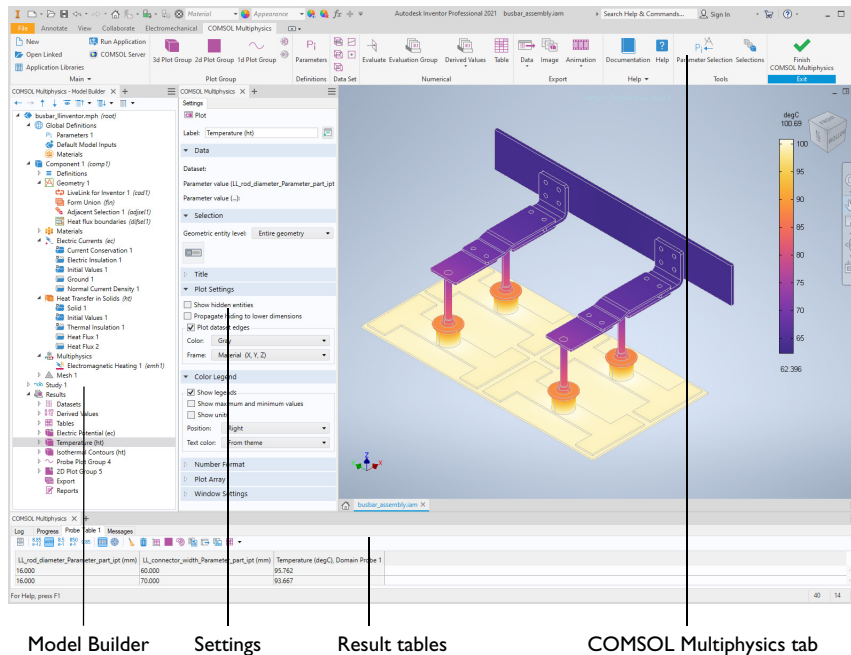


Figure 2-1: Components of the COMSOL modeling environment embedded into the Inventor user interface.

The embedded modeling environment contains the same components as the COMSOL Desktop in the following arrangement:

- The *Model Builder* appears in a window to the left of the Inventor application window.
- The *Application Libraries*, *Model Wizard*, *Material Library*, and *Settings* windows appear in a window to the right of the Model Builder.

- The *Log*, *Progress*, *Messages* and *Result tables* windows appear as tabs in window at the bottom of the Inventor application window.
- The **COMSOL Multiphysics** tab is added to the Inventor ribbon.

### *Basic Steps to Build a Model*

---

To start modeling with COMSOL Multiphysics embedded environment you need an Inventor file containing a 3D geometry.

- 1 Start by opening an Inventor part or assembly.
- 2 Switch to the **COMSOL Multiphysics** tab in the Inventor ribbon and click **New**.  
COMSOL Multiphysics is started up, and the COMSOL Multiphysics environment is initialized. When this is done the **Model Wizard** appears displaying the **Select Physics** window.

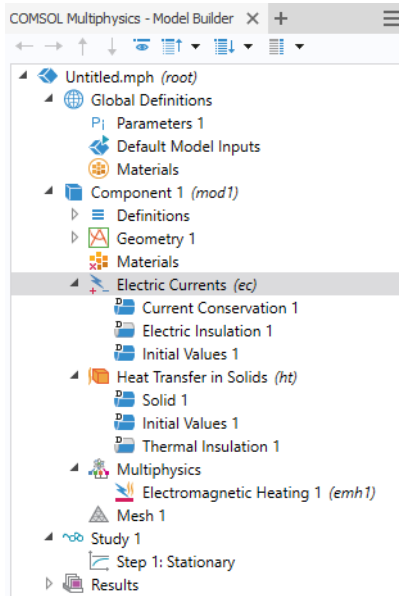


While modeling you are working in the COMSOL Multiphysics environment in Inventor. Closing this environment with the **Finish COMSOL Multiphysics** button returns you to the general working environment in Inventor, where you can continue working with the CAD design. To return to the COMSOL environment you can click the **COMSOL Multiphysics** button on the **Environments** tab.

- 3 Select one or several physics interfaces to add to the model, for example by double-clicking a physics interface.
- 4 Click the **Study** button to continue to the **Select Study** page.
- 5 Select a study type.
- 6 Click **Done** to confirm your selections and to close the **Model Wizard**.

After clicking **Done** the model is initialized and the geometry is synchronized with the model. Once this is done the nodes in the **Model Builder** automatically display their default sequences, depending on the choices you made for your model.

- 7 Click the **Expand All** button in the **Model Builder** to take a look at the default sequences for the model.



*Figure 2-2: The defaults sequences for the nodes in the Model Builder after creating a model with the Joule Heating interface with a Stationary study type that adds the Electric Currents interface, Heat Transfer in Solids interface, and Multiphysics couplings to the Model Builder.*



The geometry sequence automatically includes the **Geometry from Inventor** node, which corresponds to the operation that synchronizes the Inventor geometry to the COMSOL model. This is the only feature node that appears in the geometry sequence when you are modeling inside Inventor.

### *Making Changes to the CAD Geometry*

Once you have started modeling, you can switch from the COMSOL Multiphysics environment to the general Inventor<sup>®</sup> working environment to modify the geometry. Do this by:

- clicking the **Finish COMSOL Multiphysics** button on any of the available tabs on the ribbon.

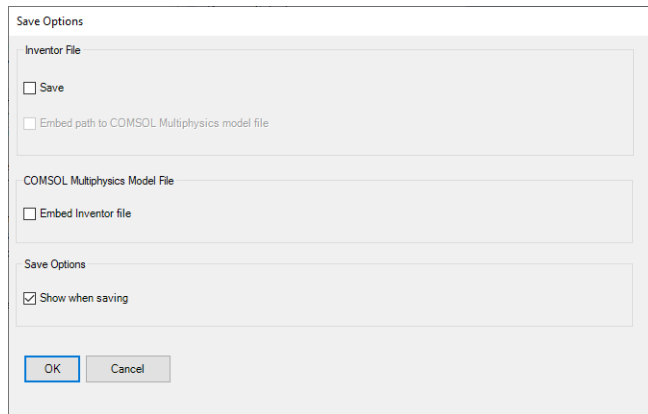
After you modify the geometry in the Inventor file, the changes are automatically synchronized to the COMSOL model as you switch to the COMSOL Multiphysics environment. You can do this by:

- clicking the **COMSOL Multiphysics** button on the **Environments** tab.

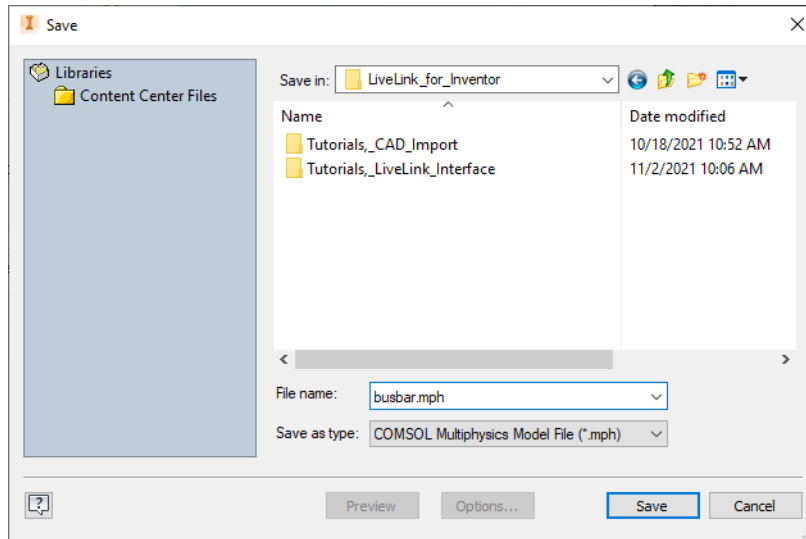
### *Saving a Model*

---

- 1 Save a COMSOL model just as you usually would save files in Inventor, for example by clicking the **Save** button on the *Quick Access Toolbar*. The **Save Options** dialog box appears.



- 2 Change the options if needed, according to the section [Save Options](#), then click **OK**, which opens the **Save** dialog box, if the model has not been saved before.



- 3 In the **Save** dialog box specify the name of the file.



Models saved from the embedded COMSOL interface in Inventor have the same file type (.mph) as other COMSOL model files. These files can be opened from the standalone COMSOL Desktop.

### SAVE OPTIONS

The settings in the **Save Options** dialog box let you control the type of information that is saved in the Inventor and the COMSOL model files and how the two are connected together.

The following options are available:

- Select **Save** to save the Inventor geometry in a state corresponding to the last synchronization with the COMSOL model.



- Only available when you are saving the Inventor file, the **Embed path to COMSOL Multiphysics Model File** option allows you to save the location of the COMSOL model inside the Inventor file.

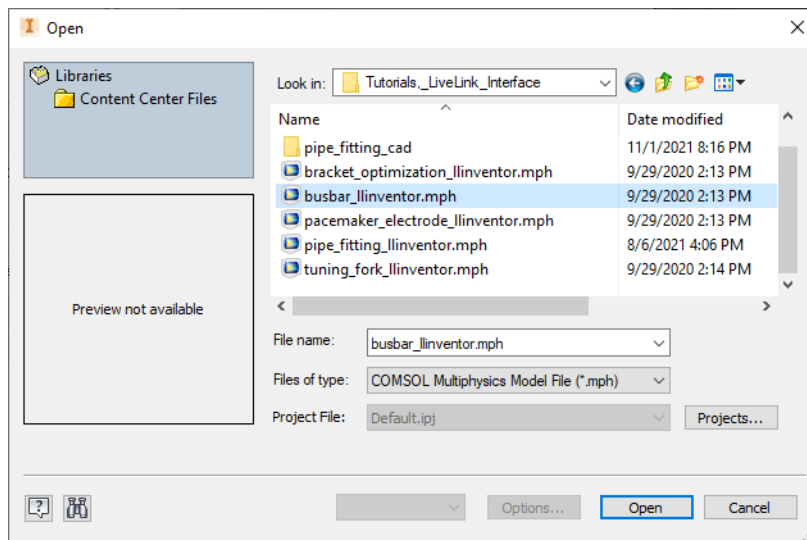


An Inventor file can contain references to several COMSOL model files. See [Opening a Model](#) on how to open COMSOL models that had their locations saved in the associated Inventor file.

- To create an easy to share package of the COMSOL Model and the associated Inventor file select **Embed Inventor file**. A copy of the Inventor file is saved inside the COMSOL model.
- By clearing the **Show when saving** check box you can disable the display of the **Save Options** dialog box when saving a COMSOL model file. You can still access these settings on the **Save** page of the **Preferences** dialog box, available if you expand the **Main** section of the **COMSOL Multiphysics** tab.

## Opening a Model

- 1 Open COMSOL models just as you would open a regular Inventor file, for example click the **Open** button on the *Quick Access Toolbar*.



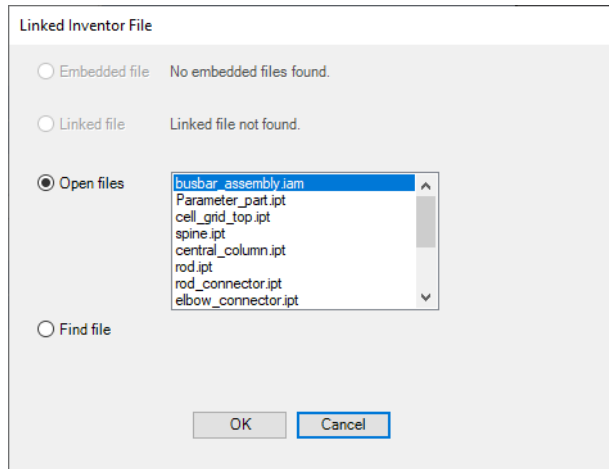
- 2 From the list of file types select COMSOL Multiphysics Model File (\*.mph).

**3** Select a file and click **Open**.

In case the COMSOL model file has been saved from inside the embedded modeling environment in Inventor you can continue with the steps in the following section, [Models Saved from inside Inventor](#). The section [Models Saved from the COMSOL Desktop](#) details the opening of COMSOL models saved from the standalone COMSOL GUI.

### MODELS SAVED FROM INSIDE INVENTOR

After you select an .mph file and click open in the **Open** dialog box in Inventor the **Linked Inventor File** dialog box appears.



The geometry in a COMSOL model created in Inventor is synchronized to the design in an Inventor file. At the time of opening a COMSOL model you need to specify an Inventor file to be linked with the model.

- I** Select one of the following options for the linked Inventor file:
  - Select **Embedded file** to open the Inventor file that has been saved inside the COMSOL model file. With this option you will be asked to save the embedded Inventor file.
  - Select **Linked file** to open the linked Inventor file from the location saved in the COMSOL model file.
  - Select **Open files**, then select one of the listed files, to link the COMSOL model to one of the designs already open in Inventor.
  - Select **Find file** to locate an Inventor file and link it to the COMSOL model.

- 2 Click **OK** to continue opening the file or **Cancel** to cancel.

### MODELS SAVED FROM THE COMSOL DESKTOP

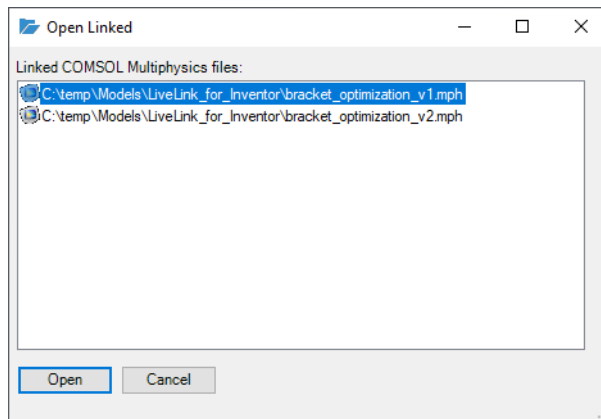
After you select a model .mph file and click **Open** in the **Open** dialog box in Inventor the geometry sequence in the COMSOL model that you are opening is replaced by a LiveLink feature node and at the same time the COMSOL geometry is transferred to an Inventor file. When the **Save As** dialog box appears select a location and name for this Inventor file, then save it. This Inventor file is now linked to the COMSOL model.

To keep the COMSOL model containing the original geometry sequence you can save the model with a different file name after opening it in Inventor. Select for example **Save As** from the Inventor **Application** menu, then follow the steps in the section [Saving a Model](#).

### OPENING LINKED MODELS

An Inventor file can contain references to one or several COMSOL model files associated with it, see [Saving a Model](#).

- 1 Open an Inventor file that has been linked to a COMSOL model.
- 2 Click the **Open Linked** button on the **COMSOL Multiphysics** tab.



The **Open Linked** dialog box appears. It lists those COMSOL model files that can be found at the location saved in the Inventor file.

- 3 Select a COMSOL model file from the list and click **Open**. Or click **Cancel** to cancel the operation.

# Connecting to COMSOL Server™ and Running Applications

## *Overview*

---

A COMSOL® runnable application is a COMSOL Multiphysics® MPH-file that, in addition to the model part, includes a custom user interface that you can run as a separate application or in a web client. For simulations with geometry that comes from an Inventor® model, applications can also use the LiveLink™ interface for Inventor®.


To create such applications, use the Application Builder, which is included in the Windows® version of COMSOL Multiphysics. For more information on how to build applications, refer to the book *Introduction to Application Builder*.

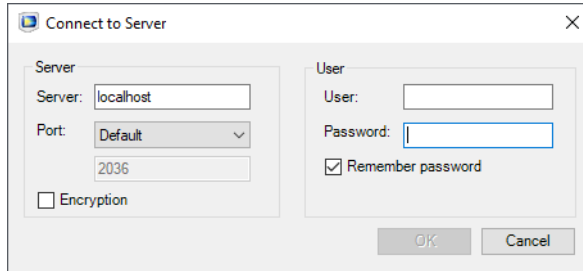
With a COMSOL Server™ license together with a license for LiveLink™ for Inventor®, a COMSOL application that uses the LiveLink interface can be run by connecting to a COMSOL Server with an easy-to-install COMSOL Client, available for download from [www.comsol.com](http://www.comsol.com). The software components installed with the COMSOL Client allow you to connect to a COMSOL Server right from the user interface of Inventor®, and to browse and launch apps using the COMSOL Server interface. COMSOL Server or COMSOL Client does not include the Application Builder, Physics Builder, and Model Builder tools that come with the COMSOL Desktop® environment, and that are required for building applications.

For more information on the installation and administration of the COMSOL Server, and on how to run applications, refer to the book *COMSOL Server Manual*.

## Connecting to a COMSOL Server™

---

To log in to a COMSOL Server™ interface, on the **COMSOL Multiphysics** tab in Inventor® click the **COMSOL Server** (  ) button. For accessing a remote server use the



computer name and domain, or the IP address, instead of localhost. If the port number 2036 is taken, then COMSOL Server will use the next available port number: 2136, 2236, and so on. Log in to COMSOL Server with your username and password. After you click **OK** the COMSOL Server interface is displayed embedded in the Inventor® user interface. Here you can view the apps available in the Application Library.

To launch an app click the **Launch** button below its icon. The app is launched in a separate COMSOL Client window. The first time you start the COMSOL Client you will need to log in to the COMSOL Server.

As an alternative to connecting to the COMSOL Server interface from Inventor®, you can also start the COMSOL Client from the Windows® Desktop or **Start** menu, and log in to the COMSOL Server to launch apps.


---

**Note:** Applications that use LiveLink™ for Inventor® require a COMSOL Client installation as they are not supported to be run from a web browser.

---

## Running Applications with COMSOL Multiphysics®

---

If you have a COMSOL Multiphysics® installation you can still launch apps from the Inventor® user interface by clicking the **Run Application** (  ) button on the **COMSOL Multiphysics** tab in Inventor®. In the **Open** dialog box browse to the application, then click **Open**. This will bring up the app interface in a separate window. If the application utilizes the LiveLink™ interface make sure that the CAD document is open in Inventor® before using the app.

Note that the **Run Application** button is disabled if you have a COMSOL Client installation of LiveLink™ *for* Inventor®. In this case run the app with the COMSOL Client, for example by first logging in to a COMSOL Server interface as described in the section [Connecting to a COMSOL Server™](#).

# Geometry Tools and Features

This chapter describes the tools and features available for importing and modifying geometry with LiveLink™ *for* Inventor®.

# 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 LiveLink™ *for Inventor*® 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.

The 3D operations and primitives listed in [Table 3-1](#) 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 3-1: 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 geometries**.

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

### *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.

## ASSIGNED ATTRIBUTES

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

## *Converting Objects to CAD Kernel Representation*

---

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

An automatic conversion to CAD objects is also performed before exporting geometry in the ACIS<sup>®</sup>, Parasolid<sup>®</sup>, 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 3-2: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
ACIS®	1	.sat, .sab	up to 2021 1.0
AutoCAD®	1, 2	.dwg, .dxf	2.5-2022
CATIA® V5	2, 3	.CATPart, .CATProduct	R8 to R2022
IGES	1	.igs, .iges	up to 5.3
Inventor® assembly	1, 2	.iam	11-2022
Inventor® part	1, 2	.ipt	6-2022
NX®	1, 4	.prt	up to 1980
Parasolid®	1	.x_t, .x_b	up to V34.0
PTC® Creo® Parametric™	1	.prt, .asm	1.0-8.0

TABLE 3-2: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
PTC <sup>®</sup> Pro/ENGINEER <sup>®</sup>	1	.prt, .asm	16 to Wildfire 5
SOLIDWORKS <sup>®</sup>	1, 2, 5	.sldprt, .sldasm	98-2022
STEP	1	.step, .stp	AP203E1, AP214, AP242

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<sup>®</sup> 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. The input surface objects must have manifold topology, and the operation can only form solids with manifold topology. An example of a solid object with nonmanifold topology is a solid that has an interior surface that separates two domains. A surface object that contains an edge that is adjacent to more than one boundary is an example of a surface object with nonmanifold topology.
- Choose **Knit 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 than this tolerance.

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

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



#### ASSIGNED ATTRIBUTES

Select the **Construction geometry** check box to make the resulting objects available only in the feature's geometry sequence. For more information see [Construction Geometry](#).

#### *Exporting Objects to 3D CAD Formats*

---

With a license for LiveLink™ for Inventor® you can export 3D geometry objects to the ACIS® (version 2016 1.0), IGES (version 5.3), Parasolid® (version 34.0), and STEP (version AP203) 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 finding and removing details from imported 3D CAD geometry, and locating overlaps and gaps in imported CAD assemblies. 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. Here you also find the **Detect Interferences** tool to analyze the interference of geometry objects. 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.

## Detect Interferences

---

Use the **Detect Interferences** () page to find overlaps and gaps between objects.

In the **Input objects** list, select the objects you want to analyze. In the **Tolerance** text field, enter the absolute tolerance, which has the default value of 0.1 mm, to be used for the search. When you click the **Find Interferences** button, a list of interferences is displayed in the **Interfering faces** list.

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

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

By default the **Interfering faces** list displays the detected interferences in a tree with the interferences as the top level nodes sorted by the type of interference. To sort the list by the objects select the **Group by object** check box. In this case the objects involved in an interference are listed as the top level nodes in the tree. Expand a top level node to see the list of objects that the object on the top level is interfering with. Expand sub node to see the list of detected interferences for the objects. You can expand the nodes for the detected interferences to reveal the two interfering objects, and then expand the nodes for the objects to look at the interfering faces displayed in sub-nodes according to the following:

- For two intersecting solid objects the nodes for the objects have two sub-nodes each, *Inside* and *Outside*. Click the *Inside* node to highlight the parts of the interfering faces that are inside the other object. Click the *Outside* node to highlight the parts of the interfering faces that are outside the other object.

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

#### **VISUALIZATION OF DETECTED INTERFERENCES**

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

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

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

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

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


# Geometry Features

In this section:

- Cap Faces
- Check
- Delete Faces
- Delete Fillets
- Delete Holes
- Delete Short Edges
- Delete Sliver Faces
- Delete Small Faces
- Delete Spikes
- Detach Faces
- Knit to Solid
- Projection
- Repair

## *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.

## ASSIGNED ATTRIBUTES

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

## *Check*

---

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

Select the objects that you want to check in the **Graphics** window. These then appear in the **Input objects** list. If the geometry sequence includes user-defined selections above the **Check** 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.


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

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

- **Tolerance issues:** The warning messages *vertex not on edge*, *vertex not on face*, *edge not on face* belong to this category. These type of faults indicate that the topology (structure) of the object is not correct locally, for example that an edge is not located on the boundary where it is expected to be. You can often repair tolerance issues in an object by applying the Repair operation with a tolerance that is larger than the tolerance used for the import. Repair adjusts the tolerance of the entities where required, and, if this fixes the issues, warning nodes will not be displayed after the Repair node. In some cases it may also help to use the option **Simplify curves and surfaces** for the Repair operation. Note that increasing the repair tolerance too much may lead to removing important details from the geometry. As an alternative to the Repair operation you can also re-import the CAD file using a larger tolerance, however the option to simplify curves and surfaces is not available in this case. For more details see [Repair](#), and [Import Options](#).
- **Invalid entities:** The warning messages *invalid curve or surface*, *self-intersecting face*, *face-to-face inconsistency*, *self-intersecting curve or surface* signify invalid entities. If you encounter any of these faults after importing a geometry from a CAD file you may need to repair the geometry to avoid problems with meshing. For example:
  - For face-to-face inconsistency faults on solid objects, try to repair the object using the **Repair face-to-face inconsistencies in solids** option for the [Repair](#) feature.
  - Try to replace the faulty face. Use the Delete operation to delete the face, then use the Cap Faces operation to generate a new face in its place. Another solution may be to use Delete Faces to delete and patch in one operation several connected faces that have a fault.

### *Delete Faces*

---

To delete and replace faces from an object, from the **Geometry** toolbar, **Defeaturing and Repair** 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<sup>®</sup> 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

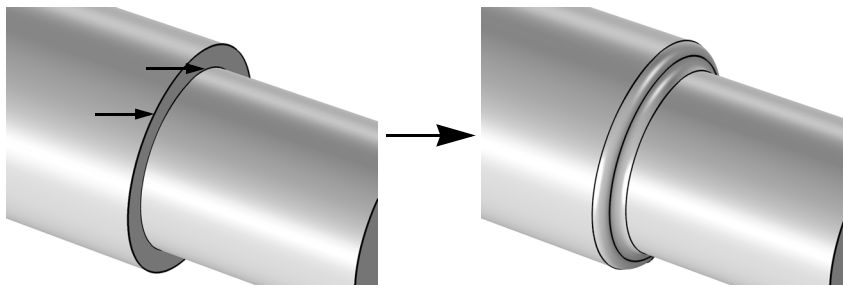
modify it by editing the corresponding Delete Fillets node that appears in the geometry branch by clicking it.



The Delete Fillets tool can only be applied to objects that are represented by the Parasolid<sup>®</sup> 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 than 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 than 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<sup>®</sup> 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 than 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<sup>®</sup> 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 than 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<sup>®</sup> 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 than 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<sup>®</sup> 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 than 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<sup>®</sup> 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.

### *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 than 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.

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

#### **SELECTIONS OF RESULTING ENTITIES**

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

Select the **Resulting objects selection** 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.



### ASSIGNED ATTRIBUTES

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

### *Projection*

---

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

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

### PROJECTION

From the **Project** list, choose **All objects** (the default) to project all 3D geometry objects to the work plane, or choose **Selected objects**, **Selected domains**, **Selected boundaries**, **Selected edges**, or **Selected vertices** to project only the objects or entities that you add to the **Entities to project** list that appears.

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

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

Projecting the outline for surface and solid objects results in the edges that form the boundary of the shadow of the object's faces. In this case the edges are projected only if they coincide with the outline. If you project the outline of a mixed object that

contains isolated edges and vertices, only the outline of the faces is generated, the isolated edges and vertices are not projected.

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

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

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

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

### SELECTIONS OF RESULTING ENTITIES

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


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

## ASSIGNED ATTRIBUTES

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

## *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 within the **Absolute import tolerance** the underlying curve and surface manifolds of the geometric entities.

Repairing objects with this option may improve both the performance and reliability of geometric operations on some imported geometry, for example it may help in some cases when Boolean operations on the imported objects fail. Simplification means that

the manifolds are converted where possible to analytical form: linear, circular, and elliptical curves; and planar, spherical, cylindrical, conical, and toroidal surfaces. Manifolds that are converted are B-spline curves and surfaces, or certain surfaces generated by operations such as sweeping, revolving, and filleting.

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

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

### SELECTIONS OF RESULTING ENTITIES

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

Select the **Resulting objects selection** 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.

### ASSIGNED ATTRIBUTES

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



# Programming and Command Reference

In this section you find detailed COMSOL<sup>®</sup> API reference information for the geometry features in LiveLink<sup>™</sup> *for* Inventor<sup>®</sup>.

# Defeaturing Tools

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

```
model.component(<ctag>).geom(<tag>).defeaturing("Filletts");
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");
model.component(<ctag>).geom(<tag>).
    defeaturing("DetectInterferences");
```

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

The features corresponding to the defeaturing tools are `DeleteFilletts`, `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](#)
- [Defeaturing Tools — Delete Faces](#)
- [Defeaturing Tools — Detach Faces](#)
- [Defeaturing Tools — Detect Interferences](#)

## *Defeaturing Tools — Finding and Deleting Small Details*

---

The defeaturing tools `Filletts`, `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("Filletts").
    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 *n*th detail, type

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

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

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

To delete all details found, type

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

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

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

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

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

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

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

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

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

### DEFEATURING METHODS

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

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

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

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

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

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

## DEFEATURING SELECTION METHODS

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



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

---

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

You can select groups either by explicitly referring to group numbers, or by selecting geometric entities. In the latter case, any group that has 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>);
```

### *Defeaturing Tools — Detect Interferences*

---

Access the `DetectInterferences` tool by the command

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

To access the input objects selection, use

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

To set the tolerance, use

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

To find interferences, type

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

To access the resulting interferences, use

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

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

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

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

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

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

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

to exit the local state.

See also [DetectInterferences](#).

# Summary of Commands

- `CapFaces`
- `Check`
- `ConvertToCOMSOL`
- `DeleteFaces`
- `DeleteFilletts`
- `DeleteHoles`
- `DeleteShortEdges`
- `DeleteSliverFaces`
- `DeleteSmallFaces`
- `DeleteSpikes`
- `DetachFaces`
- `DetectInterferences`
- `Export, ExportFinal`
- `Import 3D CAD`
- `Knit`
- `LiveLinkInventor`
- `Projection`
- `Repair`



# Commands Grouped by Function

## *Commands for Creating and Modifying Geometry in 2D*

<b>FUNCTION</b>	<b>PURPOSE</b>
<a href="#">Projection</a>	Project 3D objects and entities to a 2D work plane

## *Commands for Defeaturing*

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

## *Commands for File Import, Export, Conversion and Repair*

<b>FUNCTION</b>	<b>PURPOSE</b>
<a href="#">Check</a>	Check the validity of CAD objects
<a href="#">ConvertToCOMSOL</a>	Convert CAD Import Module geometry objects to COMSOL objects
<a href="#">Export, ExportFinal</a>	Export geometry objects to a 3D CAD file
<a href="#">Import 3D CAD</a>	Import geometry objects from a 3D CAD file
<a href="#">Knit</a>	Knit surface CAD objects to form solids or surface objects
<a href="#">Repair</a>	Repair CAD objects

### *Commands for Creating and Modifying Geometry in 3D*

<b>FUNCTION</b>	<b>PURPOSE</b>
<a href="#">CapFaces</a>	Add cap faces to fill holes in CAD geometries

### *Commands for Interfacing CAD Software*

<b>FUNCTION</b>	<b>PURPOSE</b>
<a href="#">LiveLinkInventor</a>	Synchronize geometry objects with an Inventor document

# Commands in Alphabetical Order

## *CapFaces*

---

### **PURPOSE**

Add cap faces to objects.

### **SYNTAX**

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

### **DESCRIPTION**

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

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

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

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

TABLE 4-1: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		The input edges.
selresult	on   off	off	Create selections of all resulting objects.

TABLE 4-1: 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 following attributes are available:

TABLE 4-2: VALID ATTRIBUTES

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

## Check

Check the validity of CAD objects.

### SYNTAX

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

### DESCRIPTION

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

TABLE 4-3: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		Names of input objects

## SEE ALSO

[Repair](#)

## *ConvertToCOMSOL*

---

Convert CAD objects to COMSOL objects.

### SYNTAX

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

### DESCRIPTION

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

creates a ConvertToCOMSOL feature.

TABLE 4-4: AVAILABLE PROPERTIES

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

The following attributes are available:

TABLE 4-5: VALID ATTRIBUTES

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

## SEE ALSO

[Import 3D CAD](#)

## *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").
    delete(<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 4-6: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
<code>input</code>	Selection		Faces to delete.
<code>heal</code>	<code>fill</code>   <code>patch</code>	<code>patch</code>	Healing method.
<code>throughhole</code>	<code>on</code>   <code>off</code>	<code>off</code>	Heal as if the removed faces are a through hole.
<code>selresult</code>	<code>on</code>   <code>off</code>	<code>off</code>	Create selections of all resulting objects.
<code>selresultshow</code>	<code>all</code>   <code>obj</code>   <code>dom</code>   <code>bnd</code>   <code>edg</code>   <code>pnt</code>   <code>off</code>	<code>dom</code>	Show selections of resulting objects in physics, materials, and so on, or in part instances. <code>obj</code> is not available in a component's geometry. <code>dom</code> , <code>bnd</code> , and <code>edg</code> are not available in all features.
<code>contributeto</code>	String	<code>none</code>	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 4-7: OBSOLETE PROPERTIES

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

## SEE ALSO

[DeleteFilletts](#), [DeleteSliverFaces](#), [DeleteSmallFaces](#), [Export](#), [ExportFinal](#)

### *DeleteFilletts*

Find and delete fillets in CAD objects.

## SYNTAX

```

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

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

```

## DESCRIPTION

```

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

```



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

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

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

TABLE 4-8: AVAILABLE PROPERTIES

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

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

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

The `find` and `detail` methods of

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

have the corresponding functionality for the defeaturing tool.

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

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-9: OBSOLETE PROPERTIES

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

## EXAMPLE

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

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
    create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
    set("filename", "defeaturing_demo_3.mphbin");
model.component("comp1").geom("geom1").run("imp1");
model.component("comp1").geom("geom1").feature().
    create("dfi1","DeleteFillets");
model.component("comp1").geom("geom1").feature("dfi1").
    selection("input").
    set("imp1");
model.component("comp1").geom("geom1").feature("dfi1").
    set("entsize",4e-3);
model.component("comp1").geom("geom1").feature("dfi1").find();
model.component("comp1").geom("geom1").feature("dfi1").detail().
    setGroup(1);
model.component("comp1").geom("geom1").run();
```

## SEE ALSO

[DeleteFaces](#)

*DeleteHoles*

---

Find and delete holes in CAD objects.

## SYNTAX

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

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

## DESCRIPTION

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

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

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

TABLE 4-10: AVAILABLE PROPERTIES

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

TABLE 4-10: AVAILABLE PROPERTIES

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

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

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

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

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

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-11: OBSOLETE PROPERTIES

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

## EXAMPLE

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

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

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

## SEE ALSO

[DeleteFaces](#)

### *DeleteShortEdges*

---

Find and delete short edges in CAD objects.

## SYNTAX

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

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

## DESCRIPTION

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

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

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

TABLE 4-12: AVAILABLE PROPERTIES

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

TABLE 4-12: AVAILABLE PROPERTIES

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

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

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

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

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

## COMPATIBILITY

The lengths of the edges are no longer returned.

The following property is no longer supported:

TABLE 4-13: OBSOLETE PROPERTIES

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

## EXAMPLE

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

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

```

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

```

### *DeleteSliverFaces*

---

Find and delete sliver faces in CAD objects.

#### **SYNTAX**

```

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

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

```



## DESCRIPTION

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

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

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

TABLE 4-14: AVAILABLE PROPERTIES

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

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

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

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

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

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

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-15: OBSOLETE PROPERTIES

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

## EXAMPLE

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

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

## SEE ALSO

[DeleteFaces](#), [DeleteSmallFaces](#)

## *DeleteSmallFaces*

---

Find and delete small faces in CAD objects.

### **SYNTAX**

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

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

### **DESCRIPTION**

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

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

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

TABLE 4-16: AVAILABLE PROPERTIES

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

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

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

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

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

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

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-17: OBSOLETE PROPERTIES

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

## EXAMPLE

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

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

## SEE ALSO

[DeleteFaces](#), [DeleteSliverFaces](#)

### *DeleteSpikes*

---

Find and delete spikes in CAD objects.

## SYNTAX

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

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

## DESCRIPTION

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

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

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

TABLE 4-18: AVAILABLE PROPERTIES

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

TABLE 4-18: AVAILABLE PROPERTIES

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

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

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

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

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

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

## COMPATIBILITY

The width of each spike is no longer returned.

The following property is no longer supported:

TABLE 4-19: OBSOLETE PROPERTIES

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

## EXAMPLE

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

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

## SEE ALSO

[DeleteShortEdges](#), [DeleteSliverFaces](#)

### *DetachFaces*

---

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



## SYNTAX

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

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

## DESCRIPTION

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

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

TABLE 4-20: 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. <code>obj</code> is not available in a component's geometry. <code>dom</code> , <code>bnd</code> , and <code>edg</code> are not available in all features.
contributeto	String	none	Tag of cumulative selection to contribute to.

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

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

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

### EXAMPLE

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

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

### COMPATIBILITY

The following property is no longer supported:

TABLE 4-21: OBSOLETE PROPERTIES

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

### SEE ALSO

[DeleteFaces](#)

## DetectInterferences

---

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

### SYNTAX

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

### DESCRIPTION

See [Defeating Tools — Detect Interferences](#).

Available properties:

TABLE 4-22: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		Input objects
abstol	double	0.1[mm]	Absolute tolerance
showingraphics	interferingonly   selected   other   both   all	selected	Objects to show in graphics
groupbyobject	boolean	false	Group interferences by object in GUI

## Export, ExportFinal

---

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

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

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

where *<objnames>* is a string array of object names. 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 4-23: SUPPORTED FILE FORMATS

FILE FORMAT	FILE EXTENSION
Parasolid Binary (3D)	.x_b, .xmt_bin
Parasolid Text (3D)	.x_t, .xmt_txt
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(<version>);
```

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(<unit>);
```

where *<unit>* is either `fromgeom` (default) to disable unit conversion or a supported length unit: `uin`, `um`, `mil`, `mm`, `cm`, `in`, `ft`, `m`, `km`, `mi`. To get the current value of the export length unit type:

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

#### **EXPORTING TO A STEP FILE**

When exporting to the STEP format, a unit conversion can optionally be performed during export. Use the following method to select the export length unit:

```
model.component(<ctag>).geom(<tag>).export().setLengthUnitSTEP(<unit>);
```

where *<unit>* is either `fromgeom` (default) to disable unit conversion or a supported length unit: `nm`, `uin`, `um`, `mil`, `mm`, `cm`, `in`, `dm`, `ft`, `m`, `km`, `mi`. To get the current value of the export length unit type:

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

#### **SEE ALSO**

[Import 3D CAD](#)

### *Import 3D CAD*

---

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

## SYNTAX

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

## DESCRIPTION

`model.component(<ctag>).geom(<tag>).feature().create(<ftag>,"Import")` creates an import feature. When the property filename is set to a filename recognized as a 3D CAD file, the property type is set to cad. The following properties are available.

TABLE 4-24: 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.
removedundant	on   off	off	Remove redundant edges and vertices.
repair	on   off	on	Repair imported objects.
type	cad		Type of import.

TABLE 4-24: 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 4-25: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTE	FILE EXTENSION
ACIS	1	.sat, .sab
AutoCAD	1, 3	.dwg, .dxf
CATIA V5	2, 3	.CATPart, .CATProduct
IGES	1	.igs, .iges
Inventor	1, 3	.ipt, .iam
NX	1, 4	.prt
Parasolid	1	.x_t, .x_b
PTC Creo Parametric	1	.prt, .asm
PTC Pro/ENGINEER	1	.prt, .asm
SOLIDWORKS	1, 3, 5	.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.

Note 4: Only supported on Linux, or Windows.

Note 5: Embedded parts in assemblies are not supported. To import such an assembly, first convert the embedded parts to external parts.

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 4-26: 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.



TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadcontributetoobj	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for object selections in 3D CAD import.
selcadtagobj	String[]	empty	Tags of object selections (read-only, hidden in GUI) in 3D CAD import.
selcadcolorobj	String[]	empty	Colors of object selections (read-only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be "none" (in which case it will be displayed in yellow).
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.

TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadshowbnd	on   off	empty	Show boundary selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
selcadcontributetobnd	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for boundary selections in 3D CAD import.
selcadtagbnd	String[]	empty	Tags of boundary selections (read-only, hidden in GUI) in 3D CAD import.
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.

TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
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.
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.

TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
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.
selcadtagpnt	String[]	empty	Tags of point selections (read-only, hidden in GUI) in 3D CAD import.
selcadcolorpnt	String[]	empty	Colors of point selections (read-only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be "none" (in which case it will be displayed in yellow).

The following attributes are available:

TABLE 4-27: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
construction	on   off	off	Designate the resulting objects as construction geometry.

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-28: OBSOLETE PROPERTIES

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

## SEE ALSO

[Export](#), [ExportFinal](#)

## *Knit*

---

Knit surface CAD objects to form solids or surface objects.

### SYNTAX

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

### DESCRIPTION

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

TABLE 4-29: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
fillholes	on   off	off	Attempt to generate new faces to replace missing geometry
input	Selection		Names of input surface objects.
repairtol	double	1e-5	Absolute repair tolerance.
selresult	on   off	off	Create selections of all resulting objects.
selresultshow	all   obj   dom   bnd   edg   pnt   off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
contributeto	String	none	Tag of cumulative selection to contribute to.

This function also removes gaps and spikes that are within the absolute tolerance specified in the property `repairtol`.

The following attributes are available:

TABLE 4-30: VALID ATTRIBUTES

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

### COMPATIBILITY

The following property is no longer supported:

TABLE 4-31: OBSOLETE PROPERTIES

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

### EXAMPLE

The following example imports the file `repair_demo_2.x_b`, and knits the surface objects into a solid. A gap is also removed during the operation.

```

Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
    create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
    set("filename","repair_demo_2.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
    create("knit1","Knit");
model.component("comp1").geom("geom1").feature("knit1").
    selection("input").set("imp1");
model.component("comp1").geom("geom1").feature("knit1").
    set("repairtol",1e-3);
model.component("comp1").geom("geom1").runAll();

```

### SEE ALSO

[Repair](#)

Synchronize geometry objects with an Inventor document.

**SYNTAX**

```
model.component(<ctag>).geom(<tag>).feature().  
    create(<ftag>,"LiveLinkInventor");  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    set(property,<value>);  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    getType(property);  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    updateCadParamTable(add,repl);  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    importData();  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    setAttribute(attribute,<value>);  
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
    getAttribute(attribute);
```

**DESCRIPTION**

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

The method

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).importData();
```

sends the parameters with names in `param` and the values in `paramexpr` to Inventor, then rebuilds the geometry in Inventor using the parameters, and sends back the geometry objects to COMSOL.

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.component(<ctag>).geom(<tag>).feature(<ftag>).  
    updateCadParamTable(add,repl);
```

updates the properties `param` and `paramexpr` with data read from Inventor. If the `add` argument is true, all parameter names retrieved from Inventor that do not already exist in `param` are appended to `param`, corresponding COMSOL Multiphysics parameter names are appended to `paramexpr` and the corresponding values are added to the COMSOL Multiphysics global parameters table. If the `repl` argument is true, all

parameters already present in `param` that also have `sync` set to `on` have their corresponding values replaced by the values retrieved from Inventor. If `paramexpr` has a value equal to a COMSOL Multiphysics parameter, the value of that parameter is replaced. Otherwise, if `paramexpr` is a numerical value, possibly with unit, the value of `paramexpr` is replaced.

The following properties are available.

TABLE 4-32: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
<code>check</code>	<code>on</code>   <code>off</code>		Check imported objects for errors.
<code>document</code>	String		The full path of the synchronized CAD document
<code>importtol</code>	double	1e-5	Absolute repair tolerance.
<code>keepbnd</code>	<code>on</code>   <code>off</code>	<code>on</code>	Import surface objects.
<code>keepsolid</code>	<code>on</code>   <code>off</code>	<code>on</code>	Import solid objects.
<code>levelofdetail</code>	String		The level of detail representation of the synchronized CAD document.
<code>param</code>	String[]		Names of parameters to set in Inventor. Only parameters with <code>sync</code> set to <code>on</code> are sent.
<code>paramexpr</code>	String[]   double[]		Values of parameters to send to Inventor.
<code>position</code>	String		The positional representation of the synchronized CAD document.
<code>removedundant</code>	<code>on</code>   <code>off</code>	<code>on</code>	Remove redundant edges and vertices.
<code>repair</code>	<code>on</code>   <code>off</code>	<code>off</code>	Repair imported objects.
<code>selname</code>	String[]		Read only property that corresponds to the names of the resulting selections.
<code>seltag</code>	String[]		Read only property that corresponds to the tags of the resulting selections.
<code>sync</code>	String[]		Enable/disable synchronization of parameters. Valid values are <code>on</code> and <code>off</code> .



TABLE 4-32: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
synchronizewith	active   specified	active	Synchronize the active CAD document in Inventor or the specified CAD document.
tablerecord	String		The table record of the synchronized CAD document.
unit	source   current	source	Take length unit from Inventor, or from the current geometry unit.
view	String		The view representation of the synchronized CAD document.

The following properties are available for offline synchronization.

TABLE 4-33: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
offline	on   off		Activate offline synchronization.
selectsweep	studyTag/studyStep Tag   none	none	Parametric sweep to request (for example stdl/param).
receivecount	integer	0	Number of synchronizations left in received synchronization file. Read only.
receivefilename	String	Empty String	Received synchronization file. Read only.
requestfilename	String	Empty String	Pending request file. Read only.
sweepcount	integer	0	Number of parameter tuples in received synchronization file with parametric sweep. Read only.
sweepfilename	String	Empty String	Received synchronization file with parametric sweep. Read only.

The method

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).
    receiveData(filename);
```

loads the synchronization file filename.

The method

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
  receiveDataNext();
```

loads the next synchronization from the previously received synchronization file that contains multiple synchronizations.

The method

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
  requestData(filename);
```

saves the request file `filename` for offline synchronization.

The method

```
model.component(<ctag>).geom(<tag>).feature(<ftag>).  
  requestDataAppend();
```

appends a request for synchronization to the previously saved request file.

The following attributes are available:

TABLE 4-34: VALID ATTRIBUTES

NAME	VALUE	DEFAULT	DESCRIPTION
construction	on   off	off	Designate the resulting objects as construction geometry.

---

## *Projection*

---

### **PURPOSE**

Project 3D objects and entities to a 2D work plane.

## SYNTAX

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

## DESCRIPTION

Use

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

to create a projection feature tagged *<ftag>* in the 2D sequence of the work plane feature *<wptag>*. It can compute the projection of 3D objects and entities to the work plane.

By default, you get the projection for all 3D objects that were generated by the features preceding the work plane feature. To select a subset of these objects or to select entities, set the `project` property to the appropriate entity level, and use the property `input` to select the 3D objects or entities.

The following properties are available.

TABLE 4-35: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
<code>absrepairtool</code>		<code>...geom(&lt;tag&gt;). absRepairTol()</code>	Absolute repair tolerance
<code>input</code>	Selection		Entities to project. Used when project is not all
<code>project</code>	<code>all   obj   dom   bnd   edg   vtx</code>	<code>all</code>	Project all objects or selected objects or entities
<code>projectiontype</code>	<code>edgvtx   outline   all</code>	<code>outline</code>	Projection type. Used when project is all, dom, or bnd.

TABLE 4-35: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
repairtol	double	...geom(<tag>). repairTol()	Relative repair tolerance, relative to size of each input object
repairtoltype	auto   relative   absolute	...geom(<tag>). repairTolType()	Repair tolerance type: automatic, relative, or absolute
workplane	String		Work plane to project onto
selresult	on   off	off	Create selections of all resulting objects.
selresultshow	all   obj   bnd   pnt   off	bnd	Show selections, if selresult is on, in physics, materials, and so on; or in 3D from a plane geometry. obj is not available in a component's geometry
contributeto	String	none	Tag of cumulative selection to contribute to.

The following attributes are available:

TABLE 4-36: VALID ATTRIBUTES

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

## SEE ALSO

[CrossSection](#), [WorkPlane](#)

## *Repair*

Repair CAD objects.

## SYNTAX

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

## DESCRIPTION

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

creates a repair feature tagged <ftag>. The following properties are available.

TABLE 4-37: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		Names of input objects
check	on   off	on	Check the input objects for errors.
repairtol	double	1e-5	Absolute repair tolerance
selresult	on   off	off	Create selections of all resulting objects.
selresultshow	all   obj   dom   bnd   edg   pnt   off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.
simplify	on   off	off	Simplify the underlying curve and surface manifolds of geometric entities
repairfacetoface	on   off	off	Repair face-to-face inconsistencies in solid objects
contributeto	String	none	Tag of cumulative selection to contribute to.

The function tries to remove or repair the following 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)

The following attributes are available:

TABLE 4-38: VALID ATTRIBUTES

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

## COMPATIBILITY

The following property is no longer supported:

TABLE 4-39: OBSOLETE PROPERTIES

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

## EXAMPLE

The following example imports the file `repair_demo_2.x_b`, and repairs the resulting objects.

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

```
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
  create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
  set("filename","repair_demo_2.x_b");
model.component("comp1").geom("geom1").runAll();
model.component("comp1").geom("geom1").feature().
  create("rep1","Repair");
model.component("comp1").geom("geom1").feature("rep1").
  selection("input").set("imp1");
model.component("comp1").geom("geom1").feature("rep1").
  set("repairtol",1e-3);
model.component("comp1").geom("geom1").runAll();
```

### SEE ALSO

[Check, Knit](#)

