

LiveLink™ *for* PTC® Creo® Parametric™

User's Guide

LiveLink™ for PTC® Creo® Parametric™ User's Guide

© 2012–2019 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,208,270; 9,323,503; 9,372,673; 9,454,625; and 10,019,544. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement. Portions of this software are owned by Siemens Product Lifecycle Management Software Inc. © 1986–2019. All Rights Reserved. Portions of this software are owned by Spatial Corp. © 1989–2019. All Rights Reserved.

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

Version: COMSOL 5.5

Contact Information

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

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

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM023101

C o n t e n t s

Chapter 1: Introduction

About the Product	8
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

Synchronizing the Geometry	16
The LiveLink Node	16
The COMSOL Parameter Selection Window	21
The COMSOL Selections Window	22
 Connecting to COMSOL Server™ and Running Applications	 24
Overview.	24
Connecting to a COMSOL Server™	25
Running Applications with COMSOL Multiphysics®.	25

Chapter 3: Geometry Tools and Features

Geometry Representation	28
Working with the CAD Kernel	28
Converting Objects to COMSOL Kernel Representation	29
Converting Objects to CAD Kernel Representation	30
 Importing and Exporting CAD Files	 32
Importing 3D CAD Files	32
Exporting Objects to 3D CAD Formats	37

Using the Defeaturing Tools	39
Finding and Deleting Small Details	39
Delete Faces	40
Detach Faces	40
 Geometry Features	 41
Cap Faces	41
Delete Faces	42
Delete Fillets	43
Delete Holes	45
Delete Short Edges	46
Delete Sliver Faces	47
Delete Small Faces	48
Delete Spikes	49
Detach Faces	50
Knit to Solid	51
Repair	52

Chapter 4: Programming and Command Reference

Defeaturing Tools	56
Defeaturing Tools — Finding and Deleting Small Details	56
Defeaturing Tools — Delete Faces	59
Defeaturing Tools — Detach Faces	60
 Summary of Commands	 61
 Commands Grouped by Function	 62
 Commands in Alphabetical Order	 64
CapFaces	64
ConvertToCOMSOL	65
DeleteFaces	65
DeleteFillets	67
DeleteHoles	70
DeleteShortEdges	72

DeleteSliverFaces. 74

DeleteSmallFaces. 76

DeleteSpikes 79

DetachFaces 81

Export, ExportFinal 83

Import. 85

Knit. 92

LiveLinkPTCCreoParametric 94

Repair 96

Introduction

Welcome to the LiveLink™ *for* PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ enables modeling using 3D designs synchronized from the PTC® Creo® Parametric™ CAD software. The included LiveLink™ interface builds on an associative transfer of the geometry from the CAD program to the COMSOL model. 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 PTC Creo Parametric. 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
Cap Faces		Generate faces from edges to fill gaps and create solid objects, or to partition solids
Convert to COMSOL		Convert to the COMSOL kernel representation
Delete Faces		Delete and replace faces
Delete Fillets		Find and delete fillets

GEOMETRY FEATURE	ICON	DESCRIPTION
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
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 PTC Creo Parametric		Synchronize geometry between PTC Creo Parametric and COMSOL
Repair		Repair and removal of small details

Overview of the User's Guide

This documentation covers LiveLink™ for PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™*.

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

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

Opening the Documentation Window









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

Mac	<p>To open the Documentation window:</p> <ul style="list-style-type: none">• Press Ctrl+F1.• On the main toolbar, click the Documentation () button.• From the main menu, select Help>Documentation.
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 <i>COMSOL Multiphysics Reference Manual</i> .
<p><i>Opening the Application Libraries Window</i></p> <p>To open the Application Libraries window ():</p>	
	<ul style="list-style-type: none">From the Home toolbar, Windows menu, click () Applications Libraries.From the File menu select Application Libraries. <p>To include the latest versions of model examples, from the File>Help menu, select () Update COMSOL Application Library.</p>
 	<p>Select Application Libraries from the main File> or Windows> menus.</p> <p>To include the latest versions of model examples, from the Help menu select () Update COMSOL Application Library.</p>

CONTACTING COMSOL BY EMAIL

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

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

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

COMSOL WEBSITES

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

The LiveLink™ Interface

When running the COMSOL Multiphysics® software and the PTC® Creo® Parametric™ 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. After synchronization you can apply further geometry operations to prepare the synchronized geometry objects for analysis — for example, by partitioning to remove details.

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

This section includes the following topics:

- [Synchronizing the Geometry](#)
- [Connecting to COMSOL Server™ and Running Applications](#)

Synchronizing the Geometry

To initiate the geometry synchronization between PTC® Creo® Parametric™ and COMSOL Multiphysics® use the **LiveLink for PTC Creo Parametric** feature node.

Before synchronization consider to review and change the settings for the LiveLink node, and to configure the synchronization of parameters and selections in the PTC® Creo® Parametric™ file as described in this section.

This section includes the following topics:

- [The LiveLink Node](#), where you initiate a synchronization
- [The COMSOL Parameter Selection Window](#), where you can select parameters in PTC® Creo® Parametric™ to include in the synchronization
- [The COMSOL Selections Window](#), which you can use to define selections on the CAD geometry in PTC® Creo® Parametric™

The LiveLink Node

The **LiveLink for PTC Creo Parametric** feature, available from the **LiveLink** menu on the **Home** toolbar, synchronizes the geometry between PTC® Creo® Parametric™ and COMSOL Multiphysics®.

For geometry synchronization to take place both COMSOL Multiphysics and PTC® Creo® Parametric™ need to be running, and the CAD file needs to be open in the CAD software.



A list of compatible versions of PTC® Creo® Parametric™ can be found at: www.comsol.com/system-requirements/module.

ASSOCIATIVITY AND GEOMETRY OPERATIONS

During synchronization the LiveLink™ interface initiates a rebuild of the CAD geometry in PTC® Creo® Parametric™ based on the parameter values set in the COMSOL Multiphysics® 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.

Synchronized geometry objects are represented using the Parasolid® geometry kernel inside COMSOL Multiphysics. Thus, you can apply all the tools and features for defeaturing and geometry modification as included with this LiveLink™ 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 PTC® Creo® Parametric™. For the initial synchronization after adding a LiveLink™ 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 PTC® Creo® Parametric™. 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 PTC® Creo® Parametric™ until the solver completes.

The LiveLink interface also determines the **Instance** and **Simplified representation** used in the synchronized PTC® Creo® Parametric™ 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 instance and simplified representation. To be able to synchronize the CAD document in a different state, first make the switch to the desired instance or simplified representation in PTC® Creo® Parametric™, then from the **Synchronize with** list select **Active document**.

For information on how to use instances to manages configurations for parts and assemblies, and how to use simplified representations see the PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™, 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.

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 PTC® Creo® Parametric™ and transferred back to COMSOL Multiphysics®. 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 PTC® Creo® Parametric™ 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® from the PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™. Click the symbol in the **Sync** column to turn on or off the synchronization of a parameter.

Unless a unit is specified the updated are assumed to have units as defined in the PTC® Creo® Parametric™ file.

You can type in parameters in the tables, or use the **COMSOL Parameter Selection** window in PTC® Creo® Parametric™ to link parameters from the PTC® Creo® Parametric™ 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.

Synchronizing Parameters

To retrieve the linked parameters from the PTC® Creo® Parametric™ 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 PTC Creo Parametric** click the **Synchronize** button:

New parameters, which have been selected in the PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™, based on the parameters in the **Controllable parameters** table, and transferred to COMSOL.

- In the **Settings** window for **LiveLink for PTC Creo Parametric** click the **Update Parameters from CAD** button ()

New parameters, which have been selected in the PTC® Creo® Parametric™ 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 a relation inside the PTC Creo® Parametric™ file, using the symbolic format, for example d1. When the parameter refers to one of the components of a synchronized assembly the syntax also includes the name of the component, for example d1:part1.prt. 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 PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™.

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 PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ it is recommended to save the PTC® Creo® Parametric™ file in the state corresponding to the latest synchronization.

SELECTIONS

The LiveLink™ interface automatically synchronizes selections for materials from the PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ 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.

The COMSOL Parameter Selection Window

In PTC® Creo® Parametric™, right-click a feature in the model tree, then select **COMSOL Parameter Selection** to open the window with the same name. Parameters for the selection are displayed in the upper table, **Feature parameters**. Click the **Change Selection** button to select a different feature. Select the feature from the model tree,

then click **OK** in the **Select** window to return to the **COMSOL Parameter Selection** window. Select parameters from the **Feature parameters** table, then click the **Add** button to link them to COMSOL. Linked parameters are listed in the lower table, **Parameters to add to COMSOL model**. All types of parameters can be selected, but only parameters that are not driven by a relation are possible to control from a COMSOL model. Linking parameters that are relation driven enables using their values in COMSOL model definitions.

Parameters listed in the **Parameters to add to COMSOL model** table will be saved in the PTC® Creo® Parametric™ file next time you save the file. To remove saved parameters from the file clear the table first and then save the file. Click the **Revert to Saved** button to re-load the parameters from the saved PTC® Creo® Parametric™ file. Confirm your changes by clicking the **OK** button.

For additional information about parameters in CAD files see the PTC® Creo® Parametric™ documentation.

The COMSOL Selections Window

In the PTC® Creo® Parametric™ 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® model.


Alternatively, to create a user-defined selection right-click a feature or component in the PTC® Creo® Parametric™ model tree, 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 PTC® Creo® Parametric™. 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** edit field.

The following is a list of geometric entities or model elements that can be selected on a PTC® Creo® Parametric™ 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** () button. A selection can only contain entities of the same type, for example, only faces or only points. The first item added to a selection determines the type for the selection. Change the selection mode in PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ file the next time you save the file. To re-load the selections saved in the PTC® Creo® Parametric™ file click the **Revert to Saved** button.

To confirm changes and close the **COMSOL Selections** window click **OK**. This step is necessary before synchronizing between PTC® Creo® Parametric™ and COMSOL to transfer the selections.

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 a PTC® Creo® Parametric™ model, applications can also use the LiveLink™ interface for PTC® Creo® Parametric™.

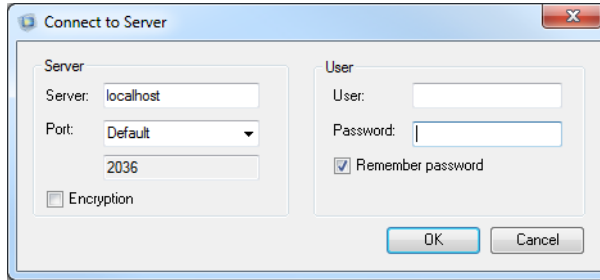
To create such applications, use the Application Builder, which is included in the Windows® version of COMSOL Multiphysics. For more information on how to build applications, refer to the book *Introduction to Application Builder*.

With a COMSOL Server™ license together with a license for LiveLink™ for PTC® Creo® Parametric™, 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. The software components installed with the COMSOL Client allow you to connect to a COMSOL Server right from the user interface of PTC® Creo® Parametric™, 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, 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 PTC® Creo® Parametric™ click the **COMSOL Server** () button. After you enter a valid




username and password the COMSOL Server interface is displayed in a separate window. 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 PTC® Creo® Parametric™, 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 PTC® Creo® Parametric™ 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 PTC® Creo® Parametric™ user interface by clicking the **Run Application** () button on the **COMSOL Multiphysics** tab in PTC® Creo® Parametric™. 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 PTC® Creo® Parametric™ before using the app.

Note that the **Run Application** button is disabled if you have a COMSOL Client installation of LiveLink™ *for* PTC® Creo® Parametric™. 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* PTC® Creo® Parametric™.

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 PTC® Creo® Parametric™ 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 models**.

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

Converting Objects to COMSOL Kernel Representation

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



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

CONVERT TO COMSOL

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

SELECTIONS OF RESULTING ENTITIES

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

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

Converting Objects to CAD Kernel Representation

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

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


If the automatic conversion cannot be performed, the geometry operation is performed by the COMSOL kernel. For example, geometry objects created from a mesh cannot be converted to CAD kernel representation. Other examples of geometry objects that cannot be converted to CAD representation include objects that have an

edge adjacent to three or more isolated faces, or objects that have a face bounded by an edge loop that intersects itself.

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

Importing and Exporting CAD Files

Importing 3D CAD Files

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

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

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

SUPPORTED FORMATS

The CAD import supports the following 3D CAD formats:

TABLE 3-2: SUPPORTED 3D CAD FILE FORMATS

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

TABLE 3-2: SUPPORTED 3D CAD FILE FORMATS

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

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

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

Note 3: This format requires, in addition to the CAD Import Module, or Design Module, or a LiveLink product for a CAD package, a license for the File Import for CATIA V5 module.

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

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

ASSOCIATIVITY

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

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

LENGTH UNIT

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

OBJECTS TO IMPORT

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

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

- Choose **Form solids** (the default) to knit together surface objects to form solids.
- Choose **Knit surfaces** to form surface objects by knitting.
- Choose **Do not knit** to not form any surface or solid objects from the imported surfaces.

For the **Form Solids** and **Knit surfaces** options select the **Fill holes** check box to generate new faces to replace missing geometry.

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

IMPORT OPTIONS

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

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

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

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

SELECTIONS OF RESULTING ENTITIES

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

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

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

SELECTIONS GENERATED BASED ON INFORMATION IN THE CAD FILE

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

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

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

- **Object Selections**
- **Boundary Selections**
- **Edge Selections**
- **Point Selections**

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

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



Click a row in a table to highlight the corresponding selection on the geometry in the Graphics window. To help with identifying the color selections, these are highlighted

with the colors defined in the imported CAD file. To always highlight on the geometry the color selections that you keep select from the **Graphics** toolbar **Colors > Show Selection Colors**.

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

Exporting Objects to 3D CAD Formats

With a license for LiveLink™ for PTC® Creo® Parametric™ you can export 3D geometry objects to the ACIS®, IGES, Parasolid®, and STEP formats. To do this:

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

Then, in the **File type** list, select **Parasolid binary file**, **Parasolid text file**, **ACIS binary file**, **ACIS text file**, **IGES file**, or **STEP file**. Use the **Browse** button to choose the filename, then click **Save** to close the Export Geometry window.

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



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

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

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

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

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



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

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



For details on which objects can be converted to CAD objects see [Converting Objects to CAD Kernel Representation](#).









Using the Defeaturing Tools

This section describes the defeaturing tools for removing details from imported 3D CAD geometry. With the defeaturing tools you can search for and delete both small details, such as short edges, small faces, sliver faces, and spikes, and larger details, for example, fillets, chamfers, and cylindrical holes.


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

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


Finding and Deleting Small Details


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

In the field **Maximum fillet radius**, **Maximum hole radius**, **Maximum edge length**, **Maximum face width**, **Maximum face size**, or **Maximum spike width**, enter the maximum size of the details you want to delete. When you click the **Find** button, a list of details that are smaller than the given size are shown in the list below. To delete the found details, either click the **Delete All** button, or select a subset of the found details in the list and click **Delete Selected**. Then, the selected details are deleted from their objects, and a node corresponding to this operation is added to the geometry branch of the model tree.


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

Delete Faces

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


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

Detach Faces

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

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

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


When you click the **Detach Selected** button, the program detaches the selected faces and adds a node corresponding to this operation to the geometry branch of the model tree. If you want to modify the performed detach operation, select the added node in the geometry branch. Then edit the node's form that appears in the **Settings** window. Click the **Build Selected** button () to see the result of your edits.

Geometry Features

In this section:

- [Cap Faces](#)
- [Delete Faces](#)
- [Delete Fillets](#)
- [Delete Holes](#)
- [Delete Short Edges](#)
- [Delete Sliver Faces](#)
- [Delete Small Faces](#)
- [Delete Spikes](#)
- [Detach Faces](#)
- [Knit to Solid](#)
- [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.

Delete Faces

To delete and replace faces from an object, from the **Geometry** toolbar, **CAD Defeaturing** menu, select **Delete Faces** (). This opens the [Delete Faces](#) window. When the deletion operation has been performed, you can modify it by editing the corresponding Delete Faces node that appears in the geometry branch by clicking it.



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

DELETE FACES

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


SELECTIONS OF RESULTING ENTITIES

If you want to make the resulting entities contribute to a cumulative selection, select a cumulative selection from the **Contribute to** list (the default, **None**, gives no

contribution), or click the **New** button to create a new cumulative selection (see [Cumulative Selections](#) in the *COMSOL Multiphysics Reference Manual*).

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

Delete Fillets

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

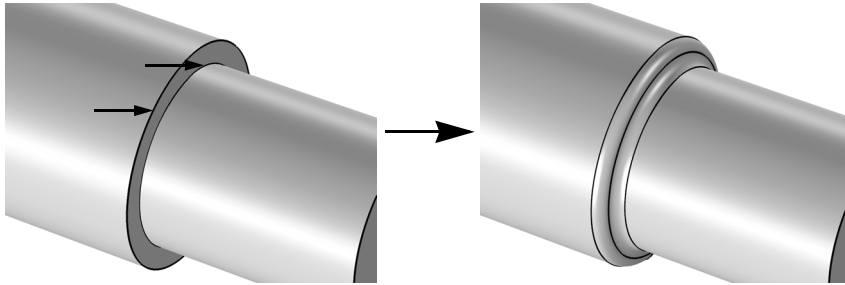


The Delete Fillets tool can only be applied to objects that are represented by the Parasolid[®] geometry kernel, also called CAD objects.

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

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

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



DELETE FILLETS


In the **Input objects** list, select the objects you want to delete fillets from. In the field **Maximum fillet radius**, enter the maximum size of the fillets you want to delete. When you click the **Find Fillets** button, a list of fillets with radius smaller 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® 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[®] 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® 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[®] 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[®] 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.

SELECTIONS OF RESULTING ENTITIES

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

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

Repair

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

REPAIR

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

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

- Entities with invalid sense
- Invalid edge and vertex tolerances
- Invalid manifolds
- Self-intersecting manifolds
- Non-G1 manifolds
- Missing edge or vertex manifolds
- Missing vertex

- Vertices not on curve of edge
- Edges and vertices not on surface of face
- Removal of surface self-intersections that lie outside the face
- Splitting at edge intersections which have no vertex
- Removal of discontinuities by either splitting or smoothing
- Remove small features (short edges, small faces, sliver faces, and spikes)

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

SELECTIONS OF RESULTING ENTITIES

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

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

Programming and Command Reference

In this section you find detailed COMSOL[®] API reference information for the geometry features in LiveLink[™] for PTC[®] Creo[®] Parametric[™].

Defeaturing Tools

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

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

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 — 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("Fillet").
    set("entsize",size);
```

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

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

The found details appear in the selection

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

To get the number of found details, type

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

To get the names of the found details, type

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

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

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

To get the object that contains the nth detail, type

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

To delete all details found, type

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

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

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

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

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

```
model.component(<ctag>).geom(<tag>).defeaturing("Fillet").
    detail().removeGroup(3);
```

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

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

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

DEFEATURING METHODS

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

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

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

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

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

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

DEFEATURING SELECTION METHODS

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



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

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

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

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

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

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

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

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

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

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

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

Defeaturing Tools — Delete Faces

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

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

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

Defeaturing Tools — Detach Faces

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

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

Summary of Commands

- [CapFaces](#)
- [ConvertToCOMSOL](#)
- [DeleteFaces](#)
- [DeleteFilletts](#)
- [DeleteHoles](#)
- [DeleteShortEdges](#)
- [DeleteSliverFaces](#)
- [DeleteSmallFaces](#)
- [DeleteSpikes](#)
- [DetachFaces](#)
- [Export, ExportFinal](#)
- [Import](#)
- [Knit](#)
- [LiveLinkPTCCreoParametric](#)
- [Repair](#)

Commands Grouped by Function

Commands for Defeaturing

FUNCTION	PURPOSE
DeleteFaces	Delete faces from CAD objects and heal the wounds
DeleteFillets	Find and delete fillets in CAD objects
DeleteHoles	Find and delete holes in CAD objects
DeleteShortEdges	Find and delete short edges in CAD objects
DeleteSliverFaces	Find and delete sliver faces in CAD objects
DeleteSmallFaces	Find and delete small faces in CAD objects
DeleteSpikes	Find and delete spikes in CAD objects
Export, ExportFinal	Detach faces from CAD objects to form a new solid

Commands for File Import, Export, Conversion and Repair

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

Commands for Geometry Creation and Modification

FUNCTION	PURPOSE
CapFaces	Add cap faces to fill holes in CAD geometries

Commands for Interfacing CAD Software

FUNCTION	PURPOSE
LiveLinkPTCCreoParametric	Synchronize geometry objects with a PTC Creo Parametric 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);
```

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.
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.

ConvertToCOMSOL

Convert CAD Import Module geometry objects to COMSOL objects.

SYNTAX

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

DESCRIPTION

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

creates a ConvertToCOMSOL feature.

TABLE 4-2: 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.

SEE ALSO

[Import](#)

DeleteFaces

Delete faces from CAD objects and heal the wounds.

SYNTAX

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

DESCRIPTION

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

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

TABLE 4-3: AVAILABLE PROPERTIES

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

The faces in the property input are deleted from their objects. The resulting object is healed so that a solid object is obtained. If heal is `fill`, a new face is formed based on

the surrounding edges of each wound. If heal is patch, the surrounding faces of each wound are grown or shrunk to heal the wound.

When you delete through holes, set the throughhole property to on to indicate that the two wounds from where the hole entered and exited the geometry are to be healed independently instead of as a single wound. If throughhole is off, the wound would be healed with a single new face that would just recreate the hole.

EXAMPLE

The following example imports the file defeaturing_demo_2.mphbin, and removes a hole from the geometry model.

```
Model model = ModelUtil.create("Model1");
model.component.create("comp1");
model.component("comp1").geom().create("geom1",3);
model.component("comp1").geom("geom1").feature().
    create("imp1","Import");
model.component("comp1").geom("geom1").feature("imp1").
    set("filename","defeating_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 4-4: 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.geom(<tag>).feature().create(<ftag>,"DeleteFillets");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property);
model.geom(<tag>).feature(<ftag>).find();
model.geom(<tag>).feature(<ftag>).detail();

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

DESCRIPTION

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

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

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

TABLE 4-5: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
delete	all selected	selected	Delete all fillets of given size, or a selection. Only available for the feature.
entsize	double	1e-3	Maximum fillet radius.
input	Selection		Names of input objects.
selresult	on off	off	Create selections of all resulting objects.

TABLE 4-5: AVAILABLE PROPERTIES

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

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

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

The `find` and `detail` methods of `model.geom(<tag>).defeaturing("Fillets")` has the corresponding functionality for the defeaturing tool.

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-6: OBSOLETE PROPERTIES

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

EXAMPLE

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

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
    "defeaturing_demo_3.mphbin");
model.geom("geom1").run("imp1");
model.geom("geom1").feature().create("dfi1","DeleteFillets");
model.geom("geom1").feature("dfi1").selection("input").
```

```

        set("imp1");
model.geom("geom1").feature("dfi1").set("entsize",4e-3);
model.geom("geom1").feature("dfi1").find();
model.geom("geom1").feature("dfi1").detail().setGroup(1);
model.geom("geom1").run();

```

SEE ALSO

[DeleteFaces](#)

DeleteHoles

Find and delete holes in CAD objects.

SYNTAX

```

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

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

```

DESCRIPTION

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

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

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

TABLE 4-7: AVAILABLE PROPERTIES

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

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

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

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-8: OBSOLETE PROPERTIES

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

EXAMPLE

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

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

SEE ALSO

[DeleteFaces](#)

DeleteShortEdges

Find and delete short edges in CAD objects.

SYNTAX

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

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

DESCRIPTION

`model.geom(<tag>).defeaturing("ShortEdges").delete(<ftag>)` creates a `DeleteShortEdges` feature tagged `<ftag>` with the specified properties. The

property `delete` is set to `selected`. If the feature can be built, it is inserted in the geometry sequence after the current feature; otherwise, the feature is discarded.

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

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

TABLE 4-9: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
<code>delete</code>	<code>all 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
<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.geom(<tag>).feature(<ftag>).find()` searches the input objects for edges of length less than `entsize`.

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

The `find` and `detail` methods of

`model.geom(<tag>).defeaturing("ShortEdges")` has the corresponding functionality for the defeaturing tool.

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

COMPATIBILITY

The lengths of the edges are no longer returned.

The following property is no longer supported:

TABLE 4-10: OBSOLETE PROPERTIES

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

EXAMPLE

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

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
    "defeaturing_demo_4.x_b");
model.geom("geom1").runAll();
model.geom("geom1").feature().
    create("dse1","DeleteShortEdges");
model.geom("geom1").feature("dse1").selection("input").
    set("imp1");
model.geom("geom1").feature("dse1").set("entsize",3e-3);
model.geom("geom1").feature("dse1").find();
model.geom("geom1").feature("dse1").detail().setGroup(1);
model.geom("geom1").runAll();
```

DeleteSliverFaces

Find and delete sliver faces in CAD objects.

SYNTAX

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

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

DESCRIPTION

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

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

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

TABLE 4-11: AVAILABLE PROPERTIES

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

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

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

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

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-12: OBSOLETE PROPERTIES

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

EXAMPLE

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

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

SEE ALSO

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

DeleteSmallFaces

Find and delete small faces in CAD objects.

SYNTAX

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

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

DESCRIPTION

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

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

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

TABLE 4-13: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
delete	all selected	selected	Delete all small faces of given size, or a selection. Only available for the feature.
entsize	double	1e-3	Maximum face size.
input	Selection		Names of input objects.
selresult	on off	off	Create selections of all resulting objects.

TABLE 4-13: AVAILABLE PROPERTIES

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

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

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

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

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-14: OBSOLETE PROPERTIES

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

EXAMPLE

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

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

```

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

```

SEE ALSO

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

DeleteSpikes

Find and delete spikes in CAD objects.

SYNTAX

```

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

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

```

DESCRIPTION

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

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

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

TABLE 4-15: 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.
<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 spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges.

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

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

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

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

COMPATIBILITY

The width of each spike is no longer returned.

The following property is no longer supported:

TABLE 4-16: OBSOLETE PROPERTIES

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

EXAMPLE

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

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

SEE ALSO

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

DetachFaces

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

SYNTAX

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

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

DESCRIPTION

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

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

TABLE 4-17: 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.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
    "defeating_demo_2.mphbin");
model.geom("geom1").runAll();
model.geom("geom1").feature().create("det1","DetachFaces");
model.geom("geom1").feature("det1").selection("input").
    set("imp1",6,7,8,9,11,12,13);
model.geom("geom1").runAll();
```

COMPATIBILITY

The following property is no longer supported:

TABLE 4-18: OBSOLETE PROPERTIES

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

SEE ALSO

[DeleteFaces](#)

Export, ExportFinal

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

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

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

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

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

To export the finalized geometry to a file, enter

```
model.component(<ctag>).geom(<tag>).exportFinal(<filename>);
```

where *<filename>* is a string.

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

TABLE 4-19: 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](#)

Import

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

SYNTAX

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

DESCRIPTION

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

TABLE 4-20: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
<code>check</code>	<code>on off</code>		Check imported objects for errors.
<code>filename</code>	String		Filename.
<code>fillholes</code>	<code>on off</code>	<code>off</code>	Attempt to generate new faces to replace missing geometry if the property <code>knit</code> is <code>solid</code> or <code>surface</code>
<code>importtol</code>	double	<code>1e-5</code>	Absolute repair tolerance.
<code>keepbnd</code>	<code>on off</code>	<code>on</code>	Import surface objects.
<code>keepfree</code>	<code>on off</code>	<code>off</code>	Import curve and point objects.
<code>keepsolid</code>	<code>on off</code>	<code>on</code>	Import solid objects.
<code>knit</code>	<code>solid surface off</code>	<code>solid</code>	Knit together surface objects to form solids or surface objects.
<code>removedundant</code>	<code>on off</code>	<code>off</code>	Remove redundant edges and vertices.
<code>repair</code>	<code>on off</code>	<code>on</code>	Repair imported objects.
<code>type</code>	<code>cad</code>		Type of import.
<code>unit</code>	<code>source current</code>	<code>source</code>	Take length unit from file or from the current geometry unit.
<code>unitecurves</code>	<code>on off</code>	<code>on</code>	Unite curve objects.
<code>selresult</code>	<code>on off</code>	<code>off</code>	Create selections of all resulting objects.

TABLE 4-20: AVAILABLE PROPERTIES

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

The file to import is specified by `filename`, which can have of any of the following formats:

TABLE 4-21: SUPPORTED 3D CAD FILE FORMATS

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

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

Note 2: This format requires, in addition to the CAD Import Module, Design Module, or a LiveLink product for a CAD package, a license for the File Import for CATIA V5 module.

Note 3: Only supported on Windows.

The imported geometry objects are represented using the Parasolid geometry kernel, which is the geometry kernel utilized by the CAD Import Module and the LiveLink products for CAD software.

The method

```
model.geom(gname).feature(<ftag>).importData()
```

imports the file again, even if the feature is built.

The import can generate object, boundary, edge, and point selections based on material, layer, and color assignments in the 3D CAD file. The following properties are available for working these selections:

TABLE 4-22: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadshownamesfromfileobj	boolean	false	Show the object selection names from the file in the GUI.
selcadnameobj	String[]	empty	Names of object selections in 3D CAD import.
selcadnameinfileobj	String[]	empty	Original names of object selections in 3D CAD import. Read-only.
selcadkeepobj	on off	empty	Keep object selections in 3D CAD import.
selcadshowobj	on off	empty	Show object selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
selcadcontributetoobj	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for object selections in 3D CAD import.
selcadtagobj	String[]	empty	Tags of object selections (read-only, hidden in GUI) in 3D CAD import.

TABLE 4-22: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadcolorobj	String[]	empty	Colors of object selections (read-only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be "none" (in which case it will be displayed in yellow).
selindividualintable	boolean	false	Show individual object selections and, for the knit case, individual original object selections in the CAD-tables.
selcadshownamesfromfilebnd	boolean	false	Show the boundary selection names from the file in the GUI.
selcadnamebnd	String[]	empty	Names of boundary selections in 3D CAD import.
selcadnameinfilebnd	String[]	empty	Original names of boundary selections in 3D CAD import. Read-only.
selcadkeepbnd	on off	empty	Keep boundary selections in 3D CAD import.
selcadshowbnd	on off	empty	Show boundary selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
selcadcontributetobnd	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for boundary selections in 3D CAD import.

TABLE 4-22: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadtagbnd	String[]	empty	Tags of boundary selections (read-only, hidden in GUI) in 3D CAD import.
selcadcolorbnd	String[]	empty	Colors of boundary selections (read-only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be "none" (in which case it will be displayed in yellow).
selcadshownamesfromfileedg	boolean	false	Show the edge selection names from the file in the GUI.
selcadnameedg	String[]	empty	Names of edge selections in 3D CAD import.
selcadnameinfileedg	String[]	empty	Original names of edge selections in 3D CAD import. Read-only.
selcadkeepedg	on off	empty	Keep edge selections in 3D CAD import.
selcadshowedg	on off	empty	Show edge selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
selcadcontributetoedg	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for edge selections in 3D CAD import.
selcadtagedg	String[]	empty	Tags of edge selections (read-only, hidden in GUI) in 3D CAD import.

TABLE 4-22: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
selcadcoloredg	String[]	empty	Colors of edge selections (read-only) in 3D CAD import. The color is stored as a comma-separated triple of numbers between 0 and 1. It can also be "none" (in which case it will be displayed in yellow).
selcadshownamesfromfilepnt	boolean	false	Show the point selection names from the file in the GUI.
selcadnamepnt	String[]	empty	Names of point selections in 3D CAD import.
selcadnameinfilepnt	String[]	empty	Original names of point selections in 3D CAD import. Read-only.
selcadkeeppnt	on off	empty	Keep point selections in 3D CAD import.
selcadshowpnt	on off	empty	Show point selections in 3D CAD import in physics, materials, and so on; in part instances; or in 3D from a plane geometry.
selcadcontributetopnt	String[]	empty	Tags of cumulative selection to contribute to (or none to not contribute), for point selections in 3D CAD import.

TABLE 4-22: AVAILABLE PROPERTIES

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-23: 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.geom(<tag>).feature().create(<ftag>,"Knit");
model.geom(<tag>).feature(<ftag>).selection(property);
model.geom(<tag>).feature(<ftag>).set(property,<value>);
model.geom(<tag>).feature(<ftag>).getType(property)
```

DESCRIPTION

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

TABLE 4-24: AVAILABLE PROPERTIES

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 4-25: OBSOLETE PROPERTIES

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

EXAMPLE

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

```
Model model = ModelUtil.create("Model1");
model.geom().create("geom1",3);
model.geom("geom1").feature().create("imp1","Import");
model.geom("geom1").feature("imp1").set("filename",
                                     "repair_demo_2.x_b");

model.geom("geom1").runAll();
model.geom("geom1").feature().create("knit1","Knit");
```

```
model.geom("geom1").feature("knit1").selection("input").
    set("imp1");
model.geom("geom1").feature("knit1").set("repairtol",1e-3);
model.geom("geom1").runAll();
```

SEE ALSO

[Repair](#)

LiveLinkPTCCreoParametric

Synchronize geometry objects with a PTC Creo Parametric document.

SYNTAX

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

DESCRIPTION

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

TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
check	on off		Check imported objects for errors.
document	String		The full path of the synchronized CAD document.
importtol	double	1e-5	Absolute repair tolerance.
instance	String		The family table instance of the synchronized CAD document.
keepbnd	on off	on	Import surface objects.
keepsolid	on off	on	Import solid objects.
param	String[]		Names of parameters to set in PTC Creo Parametric. Only parameters with sync set to on are sent.

TABLE 4-26: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
paramexpr	String[] double[]		Values of parameters to send to PTC Creo Parametric.
removedundant	on off	on	Remove redundant edges and vertices.
repair	on off	off	Repair imported objects
selname	String[]		Read only property that corresponds to the names of the resulting selections.
seltag	String[]		Read only property that corresponds to the tags of the resulting selections.
simplprep	String		The simplified representation of the synchronized CAD document.
sync	String[]		Enable/disable synchronization of parameters. Valid values are on and off.
synchronizewith	active specified	active	Synchronize the active CAD document in PTC Creo Parametric or the specified document.
unit	source current	source	Take length unit from PTC Creo Parametric, or from the current geometry unit.

The method

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

sets the parameters or dimensions named **param** in PTC Creo Parametric to the values **paramvalue**, rebuilds the geometry in PTC Creo Parametric, and sends back geometry objects to COMSOL.

The method

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

updates the properties **param** and **paramexpr** with data read from PTC Creo Parametric. If the **add** argument is true, all parameter names retrieved from PTC Creo Parametric that do not already exist in **param** are appended to **param**, corresponding COMSOL Multiphysics variable 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 PTC Creo Parametric. 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 imported geometry objects are represented using the Parasolid geometry kernel, which is the geometry kernel utilized by the CAD Import Module, the Design Module, and the LiveLink products for CAD software.

Repair

Repair CAD objects.

SYNTAX

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

DESCRIPTION

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

TABLE 4-27: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
input	Selection		Names of input surface objects
repairtol	double	1e-5	Absolute repair tolerance
selresult	on off	off	Create selections of all resulting objects.
selresultshow	all obj dom bnd edg pnt off	dom	Show selections of resulting objects in physics, materials, and so on, or in part instances. obj is not available in a component's geometry. dom, bnd, and edg are not available in all features.

TABLE 4-27: AVAILABLE PROPERTIES

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

The function tries to remove or repair the following defects:

- Entities with invalid sense
- Invalid edge and vertex tolerances
- Invalid manifolds
- Self-intersecting manifolds
- Non-G1 manifolds
- Missing edge or vertex manifolds
- Missing vertex
- Vertices not on curve of edge
- Edges and vertices not on surface of face
- Removal of surface self-intersections that lie outside the face
- Splitting at edge intersections which have no vertex
- Removal of discontinuities by either splitting or smoothing
- Remove small features (short edges, small faces, sliver faces, and spikes)

COMPATIBILITY

The following property is no longer supported:

TABLE 4-28: OBSOLETE PROPERTIES

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

EXAMPLE

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

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

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

SEE ALSO

[Knit](#)