CAD Import Module

User's Guide

COMSOL

CAD Import Module User's Guide

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Introduction

Welcome to the CAD Import Module! This *User's Guide* details features and functionality available with the product. For example, you can read about how to import 3D CAD files, and how to use the automatic or interactive tools for repair and defeaturing to prepare your geometry for meshing.

About this Guide

This documentation covers the CAD Import Module and the add-on for file import of CATIA® V5 files.

About this Product

The CAD Import Module enables importing 3D designs from CAD files into your simulations.

The import capabilities cover the most common 3D CAD file formats: *Parasolid*®, *ACIS*®, *AutoCAD*®, *IGES*, *STEP*, *SOLIDWORKS*®, *Inventor*®, *NX*®, *PTC*® *Creo*® *Parametric*™, and *PTC*® *Pro/ENGINEER*®. 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® or Parasolid® file formats.

Finally, the product provides a wide range of tools for you to prepare an imported 3D design for meshing and analysis. You can interactively search for and remove small features, for example, fillets, slivers, faces, and 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.

Where Do I Access the Documentation and Application Libraries?

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

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

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes all core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has

instructions about how to use COMSOL Multiphysics and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

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

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

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

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

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

THE APPLICATION LIBRARIES WINDOW

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

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

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

Opening the Application Libraries Window
To open the **Application Libraries** window (▥):

Win	• From the **Home** toolbar, **Windows** menu, click (▥) **Applications Libraries**. • From the **File** menu select **Application Libraries**. To include the latest versions of model examples, from the **File>Help** menu, select (▥) **Update COMSOL Application Library**.

	Select **Application Libraries** from the main **File>** or **Windows>** menus.
Mac	
Linux	To include the latest versions of model examples, from the **Help** menu select (▦) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

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

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

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CAD Import, Repair and Defeaturing

This section describes the tools and geometry operations available for importing and modifying 3D CAD data in the CAD Import Module.

In this section:

- Functionality Overview
- Defeaturing Tools
- Geometry Operations

Functionality Overview

This section includes these topics:

- Available Geometry Operations
- Importing Files
- Defeaturing and Repair
- Creating CAD Objects
- Converting Objects to COMSOL Geometry Kernel Representation
- Converting Objects to Parasolid® Geometry Kernel Representation
- Exporting Objects to CAD Formats

Available Geometry Operations

The CAD Import Module provides a number of tools for importing objects from 3D CAD files, repairing them, and removing unnecessary details in them. There is also the possibility to draw objects represented using the Parasolid® geometry kernel, which is utilized by the Design Module, CAD Import Module, and all LiveLink™ products for CAD packages.

The geometry operations can only be applied on *CAD objects* — that is, geometry objects represented using the Parasolid geometry kernel. However, you can convert objects from the COMSOL kernel representation to CAD objects, for example by applying a **Union** or **Convert to Solid** operation on the object.

TABLE 2-1: AVAILABLE GEOMETRY OPERATIONS

NAME	DESCRIPTION
Cap Faces	Add faces to fill holes or partition objects
Convert to COMSOL	Convert to the COMSOL geometry kernel representation
Delete Faces	Delete and replace faces
Delete Fillets	Delete fillets
Delete Short Edges	Delete short edges
Delete Sliver Faces	Delete sliver faces
Delete Small Faces	Delete small faces
Delete Spikes	Delete spikes from faces

TABLE 2-1: AVAILABLE GEOMETRY OPERATIONS

NAME	DESCRIPTION
Detach Faces	Detach faces and form a new object from them
Import	Import 3D CAD file
Knit to Solid	Knit surface object to form solid or surface object
Repair	Repair and removal of small details

To access these geometry operations, you need to use the CAD kernel for the geometry representation, which you can specify in the **Preferences** dialog box and in the **Settings** window for the **Geometry** node.

Importing Files

Use the **Import** feature to import 3D CAD files. By default, small details and defects are automatically removed and the software tries to knit surface objects to form solids.

Defeaturing and Repair

To find and delete small details in the imported objects, use the defeaturing tools. These can also be used to delete or detach larger faces.

To more automatically remove small details and repair defects, use the **Repair** feature.

To knit surface objects to form solids, use the **Knit to Solid** feature.

Creating CAD Objects

By default, the software uses the geometry kernel of the CAD Import Module. This means that the geometry you draw is represented using the Parasolid kernel. If you would like to use the COMSOL kernel instead, you can click the **Geometry** node to open the **Settings** window. Change **Geometry representation** to **COMSOL kernel**.

The 3D operations and primitives listed in Table 2-2 do not support the geometry kernel of the CAD Import Module — they always use the COMSOL geometry kernel. However, an automatic conversion is performed for these primitives before they are

used in Boolean operations, see Converting Objects to Parasolid® Geometry Kernel Representation.

TABLE 2-2: 3D OPERATIONS AND PRIMITIVES THAT DO NOT SUPPORT THE PARASOLID GEOMETRY KERNEL

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

Converting Objects to COMSOL Geometry Kernel Representation

To convert CAD objects to objects represented using the COMSOL geometry kernel, use the **Convert to COMSOL** feature node.

Converting Objects to Parasolid® Geometry Kernel Representation

If the current geometry representation for the geometry sequence is **CAD kernel**, an automatic conversion of COMSOL objects to CAD objects takes place before using the objects in Boolean operations and before using the objects in the **Convert to Solid, Convert to Surface, Convert to Curve**, and **Convert to Point** operations. This ensures that the CAD kernel is used in the abovementioned operations.

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

If the automatic conversion cannot be performed, the geometry operation is performed by the COMSOL geometry kernel. The conversion cannot be performed if there is an edge adjacent to three or more isolated faces, or if a face is bounded by an edge loop that intersects itself.

The automatic conversion to CAD 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.

Exporting Objects to CAD Formats

The CAD Import Module, Design Module, and LiveLink™ products for CAD packages add ACIS and Parasolid to the list of supported formats for geometry export.

To export 3D geometry objects to these formats:

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

In the **File type** list, select **Parasolid binary file**, **Parasolid text file**, **ACIS binary file**, or **ACIS text file**. Use the **Browse** button to choose the filename, then click **Save**. Finally, 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 Parasolid® Geometry Kernel Representation.

Defeaturing Tools

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

To access these features, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Fillets**, **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 a defeaturing page, 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.

In this section:

- Finding and Deleting Small Details
- Delete Faces
- Detach Faces

Finding and Deleting Small Details

You can use any of the **Delete Fillets** ( ), **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 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 Operations

In this section:

Cap Faces

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

CAP FACES

Select edges that form loops around the faces you want to create. The edges display in the **Bounding edges** list.

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

SELECTIONS OF RESULTING ENTITIES

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

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

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

Convert to COMSOL

To convert geometry objects represented using the Parasolid® geometry kernel, used by the CAD Import Module, to COMSOL geometry kernel representation, from the **Geometry** toolbar, **Conversions** menu, select **Convert to COMSOL** ( ).

CONVERT TO COMSOL
Select the geometry objects that you want to convert in the **Graphics** window. The objects display 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*).

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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*).

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Delete Fillets

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

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

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

The Delete Fillets tool cannot delete fillets for which the adjacent faces cannot be extended to cover the gap. The figure below shows an example of such fillets. Applying the fillets on the highlighted edges deletes the annular face from the geometry, which cannot be recreated if the fillets are to be deleted.
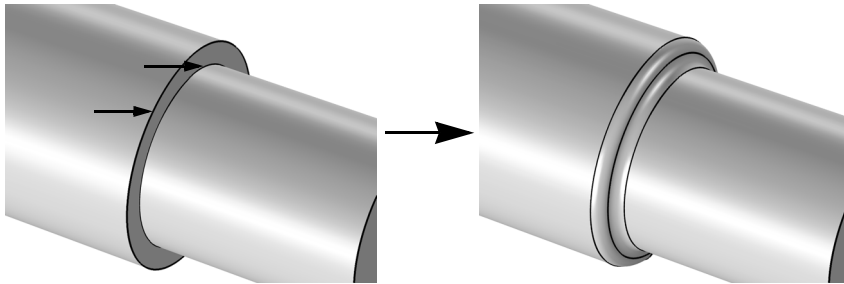
DELETE FILLETS

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

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

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

Delete Short Edges

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

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

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

DELETE SHORT EDGES

In the **Input objects** list, select the objects you want to delete short edges from. In the field **Maximum edge length**, enter the maximum length of the edges you want to delete. When you click the **Find Short Edges** button, a list of edges with length smaller that the

given value is shown in the **Short edge selection** list. If **Deletion type** is **All short edges**, all such edges are deleted. You can delete a subset of these edges by clicking in the **Short edge selection** list, and choosing **Selected short edges** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

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

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Delete Sliver Faces

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

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

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

DELETE SLIVER FACES

In the **Input objects** list, select the objects you want to delete sliver faces from. In the field **Maximum face width**, enter the maximum width of the faces you want to delete. When you click the **Find Sliver Faces** button, a list of faces with width smaller that the given value are shown in the **Sliver faces selection** list. If **Deletion type** is **All sliver faces**, all such faces are deleted. You can delete a subset of these faces by clicking in the **Sliver face selection** list, and choosing **Selected sliver faces** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

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

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Delete Small Faces

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

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

Note that this defeaturing tool cannot find small faces on nonmanifold objects. An example of a nonmanifold object is an object with several domains. Such an object can

for example result from a Union or a Partition operation. To avoid this situation defeature the geometry objects before applying Boolean operations that result in nonmanifold objects.

DELETE SMALL FACES

In the **Input objects** list, select the objects you want to delete small faces from. In the field **Maximum face size**, enter the maximum diameter of the faces you want to delete. When you click the **Find Small Faces** button, a list of faces with diameter smaller that the given value appears in the **Small faces selection** list. If **Deletion type** is **All small faces**, all such faces are deleted. You can delete a subset of these faces by clicking in the **Small face selection** list, and choosing **Selected small faces** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

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

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

Delete Spikes

A spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges. To delete spikes from an object, from the **Geometry** toolbar, **Defeaturing and Repair** menu, select **Delete Spikes** ( ). This opens the **Delete Spikes** window, see Finding and Deleting Small Details. When the deletion operation has been performed,

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

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

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

DELETE SPIKES

In the **Input objects** list, select the objects you want to delete spikes from. In the field **Maximum spike width**, enter the maximum width of the spikes you want to delete. When you click the **Find Spikes** button, a list of spikes with width smaller that the given value are shown in the **Spike selection** list. If **Deletion type** is **All spikes**, all such spikes are deleted. You can delete a subset of these spikes by clicking in the **Spike selection** list, and choosing **Selected spikes** in the **Deletion type** list.

SELECTIONS OF RESULTING ENTITIES

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

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

Detach Faces

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

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

DETACH FACES

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

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

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

SELECTIONS OF RESULTING ENTITIES

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

Select the **Resulting objects selection** check box to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the resulting objects consist of available as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, which is suitable for use with materials and physics defined in domains. For

use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence.

Import

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 in the Parasolid® geometry kernel, which is used by the CAD Import Module, Design Module, and LiveLink products interfacing CAD packages.

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

SUPPORTED FORMATS

The CAD import supports the following 3D CAD formats:

TABLE 2-3: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
ACIS®	I,	.sat, .sab	up to 2018 1.0
AutoCAD®	I, 2	.dwg, .dxf	2.5-2019
CATIA® V5	2, 3	.CATPart, .CATProduct	R8 to R2018
PTC® Creo® Parametric™	I	.prt, .asm	1.0-5.0
IGES	I	.igs, .iges	up to 5.3
Inventor® assembly	I, 2	.iam	11, 2008-2017
Inventor® part	I, 2	.ipt	6 to 11, 2008-2019
NX®	I, 4	.prt	up to 12

TABLE 2-3: SUPPORTED 3D CAD FILE FORMATS

FILE FORMAT	NOTES	FILE EXTENSIONS	SUPPORTED VERSIONS
Parasolid®	1	.x_t, .x_b	up to V31.0
PTC® Pro/ENGINEER®	1	.prt, .asm	16 to Wildfire 5
SOLIDWORKS®	1, 2, 5	.sldprt, .sldasm	98-2018
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.

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.

If the **Curves and points** check box is selected, the **Unite curve objects** check box is selected by default to unite the imported curve objects, which speeds up the rendering of the geometry.

IMPORT OPTIONS

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

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

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

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

SELECTIONS OF RESULTING ENTITIES

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

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

Select the **Individual objects selections** check box to create predefined selections (for all levels — objects, domains, boundaries, edges, and points — that are applicable) in subsequent nodes in the geometry sequence for each individual object in the geometry file and for each relevant entity level. To also make all or one of the types of resulting entities (domains, boundaries, edges, and points) that the objects consist of available

as selections in all applicable selection lists (in physics and materials settings, for example), choose an option from the **Show in physics** list: **All levels**, **Domain selection**, **Boundary selection**, **Edge selection**, or **Point selection**. The default is **Domain selection**, if available, which is suitable for use with materials and physics defined in domains. For use with a boundary condition, for example, choose **Boundary selection**. These selections do not appear as separate selection nodes in the model tree. Select **Off** to not make any selection available outside of the geometry sequence

Knit to Solid

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

KNIT TO SOLID

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

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

SELECTIONS OF RESULTING ENTITIES

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

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

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.

3

Programming and Command Reference

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

In this section:

- Defeaturing Tools
- Summary of Commands
- Commands Grouped by Function

Defeaturing Tools

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

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

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

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

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

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

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

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

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

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

The found details appear in the selection

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

To get the number of found details, type

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

To get the names of the found details, type

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

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

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

To get the object that contains the nth detail, type

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

To delete all details found, type

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

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

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

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

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

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

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

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

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

DEFEATURING METHODS

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

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

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

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

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

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

DEFEATURING SELECTION METHODS

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

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

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

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

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

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

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

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

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

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

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

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

Defeaturing Tools — Delete Faces

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

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

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

Defeaturing Tools — Detach Faces

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

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

Summary of Commands

These commands are included in this section:

- CapFaces
- ConvertToCOMSOL
- DeleteFaces
- DeleteFillets
- DeleteShortEdges
- DeleteSliverFaces

- DeleteSmallFaces
- DeleteSpikes
- DetachFaces
- Import
- Knit
- 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
DeleteShortEdges	Find and delete short edges in CAD objects
DeleteSliverFaces	Find and delete sliver faces in CAD objects
DeleteSmallFaces	Find and delete small faces in CAD objects
DeleteSpikes	Find and delete spikes in CAD objects
DetachFaces	Detach faces from CAD objects to form a new solid

Commands for File Import, Conversion and Repair

FUNCTION	PURPOSE
ConvertToCOMSOL	Convert CAD Import Module geometry objects to COMSOL objects
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 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 3-1: 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 3-2: 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.

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

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

DESCRIPTION

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

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

TABLE 3-3: 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","defeaturing_demo_2.mphbin");
model.component("comp1").geom("geom1").run("imp1");
model.component("comp1").geom("geom1").feature().
      create("dfa1","DeleteFaces");
model.component("comp1").geom("geom1").feature("dfa1").
      selection("input").set("imp1",6,7,8,9,11,12,13);
model.component("comp1").geom("geom1").run();
```

COMPATIBILITY

The following property is no longer supported:

TABLE 3-4: OBSOLETE PROPERTIES

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

SEE ALSO

DeleteFillets, DeleteSliverFaces, DeleteSmallFaces, DetachFaces

DeleteFillets

Find and delete fillets in CAD objects.

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

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

DESCRIPTION

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

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

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

TABLE 3-5: 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 3-5: 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 3-6: 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

DeleteShortEdges

Find and delete short edges in CAD objects.

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

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

DESCRIPTION

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

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

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

TABLE 3-7: AVAILABLE PROPERTIES

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

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

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

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

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

COMPATIBILITY

The lengths of the edges are no longer returned.

The following property is no longer supported:

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>);
```

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

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

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

TABLE 3-9: AVAILABLE PROPERTIES

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

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

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

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

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 3-10: 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.

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

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

DESCRIPTION

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

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

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

TABLE 3-11: AVAILABLE PROPERTIES

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

TABLE 3-11: AVAILABLE PROPERTIES

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

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

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

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

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 3-12: 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_6.x_b, finds sliver faces narrower than 10^{-3}, and deletes the first of these.

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

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

SEE ALSO

DeleteFaces, DeleteSliverFaces

DeleteSpikes

Find and delete spikes in CAD objects.

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

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

DESCRIPTION

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

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

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

TABLE 3-13: AVAILABLE PROPERTIES

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

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

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

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

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

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

COMPATIBILITY

The width of each spike is no longer returned.

The following property is no longer supported:

TABLE 3-14: 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_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>);
```

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

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

TABLE 3-15: AVAILABLE PROPERTIES

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

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

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

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

parent faces surrounding the detached faces are grown or shrunk to form a solid together with the detached faces.

EXAMPLE

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 3-16: OBSOLETE PROPERTIES

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

SEE ALSO

DeleteFaces

Import

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

SYNTAX

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

DESCRIPTION

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

TABLE 3-17: AVAILABLE PROPERTIES

PROPERTY	VALUE	DEFAULT	DESCRIPTION
`check`	`on	off`		Check imported objects for errors.
`filename`	`String`		Filename.
`fillholes`	`on	off`	`off`	Attempt to generate new faces to replace missing geometry if the property knit is solid or surface
`importtol`	`double`	`1e-5`	Absolute repair tolerance.
`keepbnd`	`on	off`	`on`	Import surface objects.
`keepfree`	`on	off`	`off`	Import curve and point objects.
`keepsolid`	`on	off`	`on`	Import solid objects.
`knit`	`solid	surface	off`	`solid`	Knit together surface objects to form solids or surface objects.
`removeredundant`	`on	off`	`off`	Remove redundant edges and vertices.
`repair`	`on	off`	`on`	Repair imported objects.
`type`	`cad`		Type of import.
`unit`	`source	current`	`source`	Take length unit from file or from the current geometry unit.
`unitecurves`	`on	off`	`on`	Unite curve objects.
`selresult`	`on	off`	`off`	Create selections of all resulting objects.
`selresultshow`	`all	obj	dom	bnd	edg	pnt	off`	`dom`	Show selections of resulting objects in physics, materials, and so on, or in part instances. `obj` is not available in a component's geometry. dom, bnd, and edg are not available in all features.
`contributeto`	`String`	`none`	Tag of cumulative selection to contribute to.

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

TABLE 3-18: 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.

COMPATIBILITY

The following property is no longer supported:

TABLE 3-19: OBSOLETE PROPERTIES

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

Knit

Knit surface CAD objects to form solids or surface objects.

SYNTAX

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

DESCRIPTION

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

TABLE 3-20: AVAILABLE PROPERTIES

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

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

COMPATIBILITY

The following property is no longer supported:

TABLE 3-21: 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

Repair

Repair CAD objects.

SYNTAX

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

DESCRIPTION

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

TABLE 3-22: 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.

TABLE 3-22: 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.
simplify	on | off	off	Simplify the underlying curve and surface manifolds of geometric entities
contributeto	String	none	Tag of cumulative selection to contribute to.

The function tries to remove or repair the following defects:

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

COMPATIBILITY

The following property is no longer supported:

TABLE 3-23: 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

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