Comsol Multiphysics

# CAD IMPORT MODULE

# VERSION 3.4



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### CAD Import Module User's Guide

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# Introduction

Welcome to the CAD Import Module. This optional package extends the COMSOL Multiphysics<sup>®</sup> modeling environment with the ability to import 3D CAD models. Besides drawing a geometric model with the CAD tools built into COMSOL Multiphysics, an alternative is to create it with a specialized CAD software application and save it to a file—one that the CAD Import Module can open and translate into a COMSOL Multiphysics geometry object. Then you continue the usual modeling procedure of meshing and simulating. In addition to the automatic repair of CAD models, the CAD Import Module also includes interactive repair and defeaturing tools that you can use to prepare the CAD model for meshing and finite-element analysis.

The CAD Import Module supports the most common 3D CAD file formats: *Parasolid*<sup>®</sup>, *SAT*<sup>®</sup>, *IGES*, and *STEP*. Moreover, it provides the ability to set up a live connection with *Solid Works*<sup>®</sup>, one of the most popular CAD programs. With this connection you can transfer geometric models directly from SolidWorks to COMSOL Multiphysics and further modify them during a modeling session.

In addition, *CATIA<sup>®</sup> V4*, *CATIA<sup>®</sup> V5*, *Autodesk Inventor<sup>®</sup>*, *Pro/ENGINEER<sup>®</sup>*, and *VDA-FS* are other widely used CAD formats, and for each of them COMSOL offers a separate add-on to the CAD Import Module.

This documentation covers the CAD Import Module as well as all these add-ons. For an overview of the different products and the file formats they support see Table 1-1 on page 5.

This chapter provides an overview and some background on the different file formats. It concludes on page 6 with some "quick start" instructions on using the CAD Import Module. Detailed descriptions on how to use various features of this module in an optimal way appear in Chapter 2, "Using the CAD Import Module."

## New Features in the CAD Import Module 3.4

- You can now export 3D CAD geometries to Parasolid text and binary file formats (\*.x\_t and \*.x\_b files). This can be useful for saving a geometry that you have repaired or modified using the defeaturing tools in the CAD Import Module. It also provides an opportunity to translate 3D CAD files from other formats into Parasolid files.
- An upgraded Parasolid engine provides greater robustness and faster import of 3D CAD geometries.

# Typographical Conventions

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.
- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see

the term "Draw" referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).

- The symbol > indicates a menu item or an item in a folder in the Model Navigator. For example, Physics>Equation System>Subdomain Settings is equivalent to: On the Physics menu, point to Equation System and then click Subdomain Settings.
   COMSOL Multiphysics>Heat Transfer>Conduction means: Open the COMSOL Multiphysics folder, open the Heat Transfer folder, and select Conduction.
- A Code (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as "Type 1.25 in the **Current density** edit field." The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

# Overview of the CAD Import Module

To get an overview of the module, first review some general background information about CAD file formats; then you can better understand the list of all file formats that various COMSOL products support. This section also includes some recommendations on which format to use together with a specific CAD program. For more details on the live connection with SolidWorks, see the section "SolidWorks Live Connection" on page 30.

## CAD Software, Geometry Kernels, and File Formats

Each CAD program uses a geometry kernel to create a mathematical description of the objects and to calculate the results of solid-modeling operations. Parasolid and ACIS<sup>®</sup> are the two most common kernels, and many CAD programs license these kernels. In addition, some programs use their own kernel (as does COMSOL Multiphysics). Each of these kernels has a native file format associated with it. For example, Parasolid's file format is simply called Parasolid, and the one from ACIS is called SAT.

The geometry kernel defines the type of internal representations used for 3D modeling, which can vary considerably among different kernels. That explains why the representations stored in the various file formats are also very different. The CAD Import Module can read all these different descriptions of objects and translate them into COMSOL Multiphysics geometries.

In addition to the file formats that are native to a geometry kernel, yet other formats are based on neutral standards that were defined to ease the exchange of geometric models among CAD software applications. STEP and IGES are the two most popular formats of this class.

Yet another class of files use surface-mesh geometry formats. They do not represent a model's exact 3D geometry but store only triangular meshes of the surfaces. The most common examples of these types of files formats are VRML and STL.

For 2D modeling two formats are used primarily: DXF<sup>®</sup> is the native format of the 2D modeling program AutoCAD<sup>®</sup>, whereas GDS is a neutral standard.

This discussion presents only a short overview of CAD terms. The large number of applications and formats can be rather confusing. See the Glossary on page 63 for a short description of some CAD software, kernels, and file formats.

# File Formats Supported by COMSOL Products

The various COMSOL products can import different sets of file formats. COMSOL Multiphysics itself comes with support for 2D formats and the surface-mesh geometry formats. The CAD Import Module covers the most common needs for translating 3D models; it reads the native formats of the two most common kernels and the most popular standards. For files from other CAD software we supply additional modules as add-ons to the CAD Import Module. See Table 1-1 for a summary and details on supported versions.

PRODUCT	FILE FORMAT (FILE EXTENSIONS)	SUPPORTED VERSIONS
COMSOL Multiphysics	STL (.stl)	
	VRML (.wrl, .vrml)	1.0
	DXF (.dxf)	up to RI4
	GDS (.gds)*	2
CAD Import Module	Parasolid (.x_t, .x_b)	up to VI8
	SAT (.sat, .sab)	up to RI7
	STEP (.step, .stp)	AP203, AP214
	IGES (.igs, .iges)	up to 5.3
CATIA V4 Import Module	CATIA V4 (.model)	4.1.9 to 4.2.4
CATIA V5 Import Module	CATIA V5 (.CATPart, .CATProduct)	R2 to R17
Inventor Import Module	Autodesk Inventor (.ipt) $^{**}$	6 to 11
Pro/E Import Module	Pro/ENGINEER (.prt, .asm)	16 to Wildfire 3
VDA-FS Import Module	VDA-FS (.vda)	I and 2

TABLE I-I: FILE FORMATS SUPPORTED BY COMSOL PRODUCTS

\* GDS Import requires COMSOL Script.

\*\* Only import of parts is supported.

Geometric models do not always pass flawlessly between different file formats due to the different representations they use. This implies that the quality of a translation from a CAD model to a COMSOL Multiphysics geometry depends heavily on the file format. The smoothest way is, of course, to use the native format of your CAD system. If this is not possible, we in general recommend you to use Parasolid or STEP.

# Quick Start

Importing CAD files into the COMSOL Multiphysics modeling environment is straightforward. Go to the **File** menu and open the **Import CAD Data From File** dialog box, where you:

🐠 Import CAD D	ata From File	<b>EX</b>
Look <u>i</u> n:	📳 Computer 🔹 🦻 📴	
Recent Items	Icopy Disk Drive (A:) Local Disk (C:) BVD/CD-RW Drive (D:)	
Desktop		
Documents		
Computer		
Network	File name:	Options Import Cancel

- I Select the type of file you want to import in the Files of type list.
- **2** Choose the file to import by clicking on it or specifying its name in the **File name** edit field.
- **3** Click **Import** to close the dialog box and import the geometry in the file. This action also opens the **Import CAD Data From File** progress window.

See Chapter 2, "Using the CAD Import Module," for import options and more details on how to handle various types of geometries. There you can also find information on the SolidWorks live connection.

# Using the CAD Import Module

This chapter covers the use of the CAD Import Module, which requires the COMSOL Multiphysics package. The module provides three main features:

- · Importing CAD models from CAD-formatted files
- · Interactive repair and defeaturing of imported geometries
- Controlling the live connection to the CAD program SolidWorks

See page 8 for details and examples on how to import a CAD model from a file. To read about repair and defeaturing functionality turn to page 16. Finally, a description of the SolidWorks live interface can be found on page 30.

This chapter focuses on using the CAD Import module from the COMSOL Multiphysics graphical user interface. For details on how to access the functionality described in this chapter from COMSOL Script<sup>™</sup> or MATLAB<sup>®</sup> see Chapter 3, "CAD Import Module Command Reference," on page 35.

# Importing CAD Models from a File

Usually a CAD model is represented by either a *part* or an *assembly*, which consists of multiple parts. Importing a part into COMSOL Multiphysics results in a single geometry object, while importing an assembly produces a collection of objects. Some CAD programs use the concept of multibody parts. The CAD Import Module treats these as assemblies. There are various ways to handle imported assemblies and to continue the modeling process, choices that are described further on page 11.

For both parts and assemblies, the import procedure is quite straightforward. Select **File>Import>CAD Data From File** to open the **Import CAD Data From File** dialog box. Here you choose a CAD file and merely click **Import** to continue with the procedure. If desired, there are a number of options to customize the import procedure. You can find a detailed description of these options on page 9.

The object that appears in the drawing area of the graphical user interface after import of a part is called a *CAD geometry object*. You can distinguish this from a COMSOL Multiphysics geometry object by the different coloring. Face rendering of CAD objects is done in a paler color than COMSOL objects, and edges for CAD objects are rendered gray as opposed to black for COMSOL objects.

You have the possibility to repair or defeature CAD objects. You can also carry out transformation operations like scaling, moving, rotation or mirroring. Any other geometry operation requires the CAD geometry object to be translated into a COMSOL Multiphysics geometry object.

Meshing and analysis can only be performed on COMSOL Multiphysics geometry objects. This means that conversion of all CAD objects automatically takes place when you leave the Draw mode. However, you can always access the CAD objects if you return to Draw mode.

CAD geometry objects can only be handled by the CAD Import Module. So to be certain that the model can be opened by COMSOL Multiphysics without the CAD Import Module, convert such objects to COMSOL objects before saving your model. Do this by selecting **Draw>CAD Repair and Defeaturing>Convert to COMSOL Object**.

### EXAMPLE: IMPORT OF A CAD MODEL FROM A FILE

- I Open the Model Navigator and select a new 3D model.
- 2 Select File>Import>CAD Data From File.

- 3 In the Files of type list, select STEP file.
- 4 Locate the COMSOL installation directory, go to the cad folder, then select the file demo2.stp.
- 5 Click Import to close the dialog box and import the geometry. This action also opens the Import CAD Data From File progress window.
- **6** Click the **Headlight** button on the Camera toolbar to get a more accentuated 3D impression of the geometry.



## 7 Select Draw>CAD Repair and Defeaturing>Convert to COMSOL Object.

The CAD object is translated into a COMSOL Multiphysics geometry object. You can now save the model and access it without requiring the CAD Import Module.

# CAD Import Options

The CAD import options give you the possibility to select which type of objects to import from a CAD file. Objects types can be solids, faces, or lines. When importing face objects, you can choose how to treat these during the import procedure. You can import each face object separately, knit them together, or even form a solid from the knitted objects. The CAD Import Module can also repair geometries during the import procedure with a tolerance you can specify, and lets you turn the repairing on or off.

To change the CAD import options click the **Options** button in the **Import CAD Data From File** dialog box. This opens the **CAD Import Options** dialog box.

CAD Import Options	3
General STL / VRML	
Entities to import	
Solid entities	
Face entities	
Try forming solids	
🔘 Knit faces	
💿 Do not knit	
Tree point/edge entities	
Absolute import tolerance: 1.0E-5	
Repair imported data	
Reset to Defaults OK Cancel Help	

- Select the Solid entities check box to import solid geometry objects.
- Select the **Face entities** check box to import face objects. To combine these entities you have three choices:
  - Select the Try forming solids button to combine individual faces to solid objects.
  - Select the Knit faces button to combine the individual faces to one face object.
  - Select the **Do not knit** button to create a separate geometry object for each imported face.
- Select the Free point/edge entities check box to import isolated edges and points.
- Specify the tolerance to use during import in the Absolute import tolerance edit field.
- Select the **Repair imported data** check box to remove small gaps and tiny edges from the geometry.

Many geometries, especially those designed using a CAD program without finite element analysis, contain small anomalies: short edges, holes, or gaps. These can lead to large meshes or even failure of mesh generation. To avoid these problems, make sure the repair feature is active, and that the import tolerance is appropriate.

Repairing a CAD model means that you remove short edges, overlapping edges, and gaps.

For documentation on the import options specific to the STL and VRML file formats see the chapter "Importing STL and VRML Files" on page 89 of the *COMSOL Multiphysics User's Guide*.

# Dealing with Assemblies

A CAD model represented as an assembly is a collection of parts. When you import such a model, each part becomes a CAD geometry object in COMSOL Multiphysics. With several geometry objects present, there are two ways to continue the modeling process:

- *Creating a composite object.* A union is made of all CAD geometry objects. As a result of this, the CAD objects are converted into a single COMSOL Multiphysics geometry object. This is usually the preferred way of modeling because it leads to the best convergence when solving, and the greatest accuracy. However, it is not always possible to create such a composite object from CAD assemblies due to tolerance issues in the geometry representations.
- *Creating an assembly.* The individual geometry objects become parts of a COMSOL Multiphysics assembly. These individual parts are coupled at their boundaries with *identity* or *contact pairs.* Creating an assembly involves an easier geometry operation than creating a composite object, and is thus more robust for dealing with tolerance issues. In addition, you have greater freedom in meshing each part, and you have the possibility to model structural contact between interior boundaries of the assembly.

### CREATING A COMPOSITE OBJECT

If you choose to leave Draw mode after the import of a CAD assembly, COMSOL Multiphysics automatically creates a converted and composite object, which the program uses for meshing and analysis. You can return to Draw mode and still have access to the CAD geometry objects.

Note that the interior boundaries are kept in the process of creating a composite object. This means that the CAD parts become subdomains in the composite object. You can still apply different physics and material properties on each subdomain.

The steps involved in creating a composite object do not always succeed due to small edges in the objects, or small gaps between the individual parts. Often you can avoid these errors by manually creating the composite object and setting a *repair tolerance*. Do this in the **Create Composite Object** dialog box, which you can open by clicking the **Create Composite Object** button on the Draw toolbar. You can find more details on page

32 of the *COMSOL Multiphysics User's Guide*. Keep in mind that, after this operation, you no longer have access to the CAD geometry objects in Draw mode.

## CREATING AN ASSEMBLY

In case you want to continue working with an assembly rather than a composite object, you need to create identity pairs between the CAD geometry objects. This is necessary for the physics application modes to be able to connect the defined physics between these parts. There are two possibilities for doing this automatically in COMSOL Multiphysics:

- *Identity Pairs*. After import, with all objects selected, click **Create Pairs** on the Draw toolbar.
- Identity Pairs and Imprints. After import, with all objects selected, click Create Pairs and Imprints on the Draw toolbar. In this case imprints are created on the boundaries of objects. Objects that are modified are then converted to COMSOL Multiphysics geometry objects.

You can also perform these operations from the **Create Pairs** dialog box, which you can access by selecting **Create Pairs** from the **Draw** menu.

If your license includes the Structural Mechanics Module or the MEMS Module, you can also create contact pairs using the **Create Pairs** dialog box. Contact pairs are used when modeling structural contact. You can read about this topic in the chapter "Contact Modeling" on page 118 of the *Structural Mechanics Module User's Guide* or on page 136 of the *MEMS Module User's Guide*.

After creating identity or contact pairs, notice that there is a check mark in front of the **Use Assembly** menu item in the **Draw** menu.

As an alternative approach to letting the software set up identity pairs, you can manually select and define couplings between object boundaries. To do this after import, select **Draw>Use Assembly** to treat the geometry model as an assembly. You can set up identity boundary pairs by selecting **Physics>Identity Pairs**, or contact pairs by selecting **Physics>Contact Pairs**.

You can find more details on creating pairs in the section "Creating Assemblies" on page 352 in the *COMSOL Multiphysics Modeling Guide*. In addition, the section "Meshing Assemblies" on page 365 in the same book illustrates the advantages of the assembly feature when meshing a geometry.

### EXAMPLE: CREATING A COMPOSITE OBJECT

The following example describes how to import a CAD assembly and create a composite object.

- I Open the Model Navigator and select a new 3D model.
- 2 Select File>Import>CAD Data From File.
- 3 In the Files of type list, choose SAT file.
- 4 Locate the COMSOL installation directory, go to the cad directory, and then select the file demo3.sat.
- 5 Click Import to close the dialog box and import the geometry. This action also opens the Import CAD Data From File progress window.
- **6** Click the **Headlight** button on the Camera toolbar for a more emphasized 3D impression of the geometry.



The imported assembly consists of four parts, which were converted into four individual geometry objects in COMSOL Multiphysics.

7 Leave Draw mode by clicking the **Boundary Mode** button on the Main toolbar.

Through this act, COMSOL Multiphysics converts and combines all the geometry objects into the one composite object, which the program uses for meshing and analysis.

- 8 Click the **Draw Mode** button on the Main toolbar to return to Draw mode.
- 9 Click the Create Composite Object button on the Draw toolbar to open the Create Composite Object dialog box. In the Object selection list you can see that you still have access to four CAD objects.

If you now choose to create a composite object by taking the union of all objects in the **Create Composite Object** dialog box, you no longer have access to the CAD objects.

### EXAMPLE: CREATING AN ASSEMBLY

Import the CAD assembly from the previous example and create an assembly in COMSOL Multiphysics.

- I Open the CAD assembly example file by repeating Steps 1–6 of the previous example.
- 2 Press Ctrl+A to select all geometry objects.
- **3** Click the **Create Pairs** button on the Draw toolbar to create identity pairs. The boundaries contained in these pairs turn yellow.

Now the individual parts are coupled through the identity pairs, and you can continue the modeling procedure as usual.

## Exporting CAD Data to File

You can save a CAD geometry to a file using four different formats:

- COMSOL Multiphysics binary files (\*.mphbin) and COMSOL Multiphysics text files (\*.mphtxt)
- Parasolid binary files (\*.x\_b, \*.xmt\_bin) and Parasolid text files (\*.x\_t,\*.xmt\_txt)

The COMSOL Multiphysics files are native COMSOL file formats and make it possible to import the geometry to COMSOL Multiphysics without the CAD Import Module. For more information about these file formats, see "The COMSOL Multiphysics Files" on page 535 in the *COMSOL Multiphysics Reference Guide*.

Using the Parasolid file formats, you can translate CAD files from other formats and save a geometry, which you have modified using the repair and defeaturing tools in the CAD Import Module, to a widely recognized CAD file format.

To save the geometry to file, do the following steps:

I From the File menu, choose Export>Geometry Objects To File.

- 2 Browse to the directory where you want to save the file, specify a file name in the **File name** edit field, and select one of the four file formats from the **Files of type** list.
- 3 Click OK.

# CAD Repair and Defeaturing

Sometimes you might find it necessary to repair and defeature the CAD geometry after the import is completed. This can typically be the case if the geometry contains inaccuracies that fall outside the import tolerance, which by default is set to  $10^{-5}$ . With the CAD Import Module's defeaturing tools, you can also remove details that are not important to the finite element analysis. Repair and defeaturing are only available for CAD geometry objects.

# CAD Repair

To access the Repair dialog box choose Draw>CAD Repair and Defeaturing>Repair.

Repair	X
Object selection	Repair object Absolute tolerance: 1e-5 Repair
	Knit faces and fill gaps Absolute tolerance: 1e-5 Knit
	Close Help

- Click **Repair** to remove small entities from the selected object. Enter the tolerance in the **Absolute tolerance** edit field. Only small entities that fall within the specified tolerance are removed. Small entities can be short edges, small faces, sliver faces, and spikes.
- Click Knit to sew together selected faces. Enter the tolerance in the Absolute tolerance edit field. Spikes and gaps that fall within the specified tolerance are removed during this operation. After knitting, a solid object is automatically made from the knitted face objects.

The repair tool might sometimes have undesired effects because it removes all features that fall within the tolerance interval. To avoid this and to delete only certain features, use the interactive defeaturing tools (see page 22).

### EXAMPLE I: REPAIR

In this example you use the repair tool to remove small faces from an imported CAD model of a wheel rim.

- I Open the Model Navigator and select a new 3D model.
- 2 Select File>Import>CAD Data From File.
- 3 In the Files of type list, select Parasolid file.
- 4 Locate the COMSOL installation directory, go to the cad folder, and then select the file repair\_demo\_1.x\_b.
- 5 Click Import to close the dialog box and import the geometry. This action also opens the Import CAD Data From File progress window.
- **6** Click the **Headlight** button on the Camera toolbar to highlight the 3D impression of the geometry in a better way.



The CAD geometry includes a number of small faces, that are larger than the default import tolerance, and were therefore not removed during import.



Next, mesh the geometry:

7 Select Mesh>Free Mesh Parameters to open the Free Mesh Parameters dialog box.

- 8 On the Global page, select the Custom mesh size option button.
- 9 Type 0.4 in the Resolution of narrow regions edit field.

When dealing with imported CAD models, you can often achieve a significant reduction in number of elements without sacrificing mesh quality by this slight decrease of the parameter for resolution of narrow regions.

- IO Click OK to close the Free Mesh Parameters dialog box.
- II Mesh the geometry by clicking the Initialize Mesh button on the Main toolbar.



Note that, as soon as the meshing is complete, the message log displays the number of mesh elements, which is approximately 78,000.

- 12 Click the Draw Mode button on the Main toolbar.
- I3 Select Draw>CAD Repair and Defeaturing>Repair to open the Repair dialog box.
- 14 Type 1e-3 in the Absolute tolerance edit field in the Repair object area.
- **I5** Click **Repair**.

When the geometry repair is completed, you can see that not only small faces but also sliver faces, which were located just next to them, are removed from the model.



16 Click Close to close the Repair dialog box.

**17** Mesh the geometry by clicking the **Initialize Mesh** button on the Main toolbar.



The message log displays now around 69,000 elements for the new mesh, which is significantly smaller than before repairing the geometry.

# EXAMPLE 2: KNIT FACES

In this example you use the knit faces tool to remove a gap from an imported CAD model of an engine piston rod.

- I Open the Model Navigator and select a new 3D model.
- 2 Select File>Import>CAD Data From File.
- 3 In the Files of type list select Parasolid file.

- 4 Locate the COMSOL installation directory, go to the cad folder, and then select the file repair\_demo\_2.x\_b.
- 5 Click Import to close the dialog box and import the geometry. This action also opens the Import CAD Data From File progress window.
- **6** Click the **Headlight** button on the Camera toolbar to stress the 3D features of the geometry better.

The imported geometry is a face object, labeled with F1, which contains a gap. A solid body could not be created during import as the gap is larger than the default import tolerance.



7 Select Draw>CAD Repair and Defeaturing>Repair to open the Repair dialog box.

8 Enter 1e-3 in the Absolute tolerance edit field in the Knit faces and fill gaps area.

Repair	X
Object selection	Repair object
F1 ^	Absolute tolerance: 1e-5 Repair
	Knit faces and fill gaps Absolute tolerance: 1e-3
	Knit
	Close Help

9 Click Knit.



After the operation is completed you can see that the gap is healed and a solid body is created. The object is listed now as a solid object, labeled CO1, in the **Object selection** list box.

IO Click Close to close the Repair dialog box.

II Mesh the geometry by clicking the Initialize Mesh button on the Main toolbar.



The mesh consists of around 31,000 tetrahedral elements. You can further reduce the mesh size by defeaturing; see "Example 1: Defeaturing" on page 24 and "Example 2: Advanced Defeaturing" on page 26.

# CAD Defeaturing

To access the Defeaturing dialog box choose Draw>CAD Repair and

### Defeaturing>Defeaturing.

Defeaturing	X
Fillets Small Faces Sliver Faces Spikes	Short Edges Advanced
Detail size	Sliver faces
Maximum face width:	🖃 CO1 🔺
Ie-3 Find	Face 1 [89] Face 2 [126] Face 3 [79] Face 4 [147] Face 5 [146] Face 5 [64] Face 7 [63] Face 9 [86] Face 9 [86] Face 10 [127] Face 10 [127]
	Delete
	Close Help

Here you are presented with a number of pages, one for each type of feature—fillets, small and sliver faces, spikes, and short edges—that you can search for and then delete. In addition to the automatic detection of features, there is an **Advanced** page where you can manually select details to remove or to detach to form a new solid body.

• The Fillets page. Specify the Maximum fillet radius and click Find. The Fillets list box is populated with fillets or fillet chains that have radii smaller than that specified. Select an item from the list and click **Delete** to delete the fillet.

Depending on the geometry, the software might not always recognize fillets. In most cases you can still use the advanced defeaturing tool to select and delete appropriate faces.

- The **Small Faces** page. Specify the **Maximum face size** and click **Find**. The **Small faces** list box is populated with faces that fit within a sphere of a radius equal to the maximum face size specified. Select an item from the list and click **Delete** to delete the face.
- The Sliver Faces page. Specify the Maximum face width and click Find. The Sliver Faces list box is populated with faces that fall within the width requirements. Select an item from the list and click **Delete** to delete the face.

If COMSOL Multiphysics notifies you in the message log that deleting fails, you can test using the find and remove short edges tool to delete one or both of the short edges of the sliver face. After this you can try again to find and remove the sliver face.

• The **Spikes** page. Specify the **Maximum spike width** and click **Find**. The **Spikes** list box is populated with edges that fall within the size requirements. Select an item from the list and click **Delete** to delete the spike.

Spikes might sometimes be difficult to remove from face objects. If you encounter a geometry where this is the case, you can also use the Knit Faces tool, which can both remove spikes and fill gaps. To do this, knit the faces from the **Repair** dialog box after changing the tolerance to match the size of the spikes. You might experience greater success if you choose not to knit the faces during import and instead use the knit faces tool afterward.

- The **Short Edges** page. Specify the **Maximum edge length** and click **Find**. The **Chains of short edges** list box is populated with edges or chains of edges shorter than the specified length. Select an item from the list and click **Delete** to delete the edge.
- The **Advanced** page.

Defeatu	ring				X
Fillets	Small Faces	Sliver Faces	Spikes	Short Edges	Advanced
Face	eselection		De	elete details	
<b>.</b> (	:01	*	He	al method:	Patch   Delete
			De	tach details	
			Ch	rent heal meth ild heal methor	d: Patch from parent v
					Detach
		-			
					Close Help

Select the faces that define the feature you would like to remove or detach from the geometry object. You can do this in both the **Face selection** list box and in the drawing area. Detach removes the selected faces from the solid objects and then forms a new solid object from them. If you are deleting a detail, select a heal method and click **Delete**. If you are detaching the detail, select a parent heal method and a child heal method and click **Detach**.

For both deleting and detaching faces there are two available heal methods for the wounds that these operations cause. If you select **Fill** a surface is calculated that covers the wound and includes all the edges of the wound. With the **Patch** heal method the adjacent surfaces to the wound are either shrunk or grown until the wound is covered.

Due to the internal representation of surfaces in the COMSOL Multiphysics geometry kernel, faces might be split into two during import of CAD files. This is usually the case if it is possible to create the face by a revolve operation. A few examples are entire or partial cylinders, spheres, and cones. While the split is required for faces that describe a full revolution, even faces with less than a full revolution can be split during translation. This might have the undesired effect of introducing sliver and small faces in the geometry. You can remove these faces using the Delete Small Faces or Delete Sliver Faces tools. You can also use the Delete Details tool. Do this by selecting the small face, then select **Fill** as the heal method.

### EXAMPLE I: DEFEATURING

In this example you use the defeaturing tools to remove some features from an imported CAD model of an engine piston rod.

- I Open the Model Navigator and select a new 3D model.
- 2 Select File>Import>CAD Data From File.
- 3 In the Files of type list, select Parasolid file.
- 4 Locate the COMSOL installation directory, go to the cad folder, and then select the file defeaturing\_demo\_1.x\_b.
- 5 Click Import to close the dialog box and import the geometry. This action also opens the Import CAD Data From File progress window.
- 6 Click the Headlight button on the Camera toolbar.
- 7 Select Draw>CAD Repair and Defeaturing>Defeaturing to open the Defeaturing dialog box.

Start by removing sliver faces that were introduced during the import process.

8 On the Sliver Faces page, enter 1e-3 in the Maximum face size edit field and click Find. The Sliver faces list displays 14 faces, which are also highlighted in the model. **9** Select Faces 1-3 and 8-10 by holding down the Ctrl key while left-clicking on the items in the list.



**IO** Click **Delete** to remove the selected sliver faces.



Continue by removing some fillets present in the model.

II On the Fillets page enter 4e-3 in the Maximum fillet radius edit field and click Find.

The Fillets list displays seven fillets that are within the specified size intervall.



12 With all fillets selected in the list, click Delete to remove them.



**I3** Click **Close** to close the **Defeaturing** dialog box.

14 Mesh the geometry by clicking the Initialize Mesh button on the Main toolbar.



The mesh consists of approximately 21,000 tetrahedral elements. Compare this number to 31,000 elements, which is the mesh size for the same geometry before defeaturing (see "Example 2: Knit Faces" on page 19).

### EXAMPLE 2: ADVANCED DEFEATURING

In this example you use the advanced defeaturing tools to manually remove details from an imported CAD model. The piston rod CAD model from the previous example has already been partly defeatured and saved as a COMSOL Multiphysics geometry file. You import this file and continue the defeaturing by removing the remaining details on the face of the end where the rod attaches to the cap.

I Open the Model Navigator and select a new 3D model.

- 2 Select File>Import>CAD Data From File.
- 3 In the Files of type list, select COMSOL Multiphysics file.
- 4 Locate the COMSOL installation directory, go to the cad folder, and then select the file defeaturing\_demo\_2.mphbin.
- **5** Click **Import** to close the dialog box and import the geometry. This action also opens the **Import CAD Data From File** progress window.
- **6** Click the **Headlight** button on the Camera toolbar to get a better 3D appearance of the geometry.



- 7 Choose Draw>CAD Repair and Defeaturing>Defeaturing to open the Defeaturing dialog box.
- 8 On the **Advanced** page click the object CO1 in the **Face selection** list to expand the list of faces that belong to the object.
- **9** Select Faces 6–9 and 11–13.

**IO** Select **Patch** from the **Heal method** list in the **Delete details** area and then click **Delete**.



With the hole removed continue with removing the slot that runs across the face.

II Select Faces 7–9, 16, 19, 22, 25, 35, and 38, and then repeat Step 10.



As the final step, remove the grooves that are still left on the face.

12 Select Faces 10–13, 21, 22, and then repeat Step 10.



**I3** Click **Close** to close the **Defeaturing** dialog box.

14 Finish by clicking the Initialize Mesh button on the Main toolbar to mesh the part.



The mesh size is approximately 5000 elements. This mesh is around five times smaller than the mesh for the same geometry before defeaturing. Compare to "Example 2: Knit Faces" on page 19 and "Example 1: Defeaturing" on page 24.

# SolidWorks Live Connection

With the CAD Import Module you can set up a live connection between COMSOL Multiphysics and the SolidWorks CAD program. In this way, the geometry model present in the COMSOL Multiphysics environment mirrors the geometry model present in SolidWorks. The connection is truly bidirectional. On the one hand, you can modify a model in SolidWorks to simultaneously update the COMSOL Multiphysics geometry. On the other hand, you can define constants in COMSOL Multiphysics, values that SolidWorks can then use as geometric parameters.

A typical design and modeling process includes some or all of the following steps:

- I Create a CAD model in SolidWorks.
- **2** Import the geometry model to COMSOL Multiphysics by initializing the connection from COMSOL Multiphysics to SolidWorks.
- 3 Define a physics model in COMSOL Multiphysics and perform initial analysis.
- **4** Modify the CAD model in SolidWorks.
- **5** Update the geometry in COMSOL Multiphysics from the modified CAD model. The program maps the boundary conditions from the old geometry model to the new one.
- 6 Continue modeling in COMSOL Multiphysics.
- **7** Define constants in COMSOL Multiphysics that serve as parameters for the geometric model.
- 8 Update the connection between the two programs. SolidWorks reads the defined constants and changes the CAD model accordingly, and the connection updates the COMSOL Multiphysics object.
- **9** Perform a parameter study by repeatedly changing the values of the constants and updating the connection.

See the model on page 537 of the *COMSOL Multiphysics Model Library* for an example on how to effectively use the SolidWorks live connection in a practical example.

Note that in COMSOL Multiphysics you should not manipulate the geometry object, which mirrors a CAD model in SolidWorks. However, you can draw additional objects in COMSOL Multiphysics. This means that your model can contain both geometry objects imported from SolidWorks and geometry objects drawn in COMSOL Multiphysics, and you can still update models in SolidWorks to affect only those imported objects in COMSOL Multiphysics.

The live connection transfers only the geometry from SolidWorks to COMSOL Multiphysics; it ignores everything else. Yet, the connection requires that both programs run on the same PC.

To specify options on how to translate a SolidWorks model to a COMSOL Multiphysics geometry object, select the menu item **File>SolidWorks Connection>Options**. The options are exactly the same as for importing CAD models from files and are discussed in the section "CAD Import Options" on page 9.

**Note:** To implement the live connection, SolidWorks needs an add-in called the *COMSOL Multiphysics Interface*, which the COMSOL Installer adds to SolidWorks. However, if you install SolidWorks after installing COMSOL the connection could fail. For details on how to set up the connection, see the chapter "Adding the SolidWorks Connection to the CAD Import Module" on page 18 of the *COMSOL Installation Guide*.

# Example of Using the Solid Works Live Connection

Start the SolidWorks program on your computer and open the example geometry by locating the COMSOL installation directory, going to the cad folder, and then opening the file demo4.SLDPRT.



Set up the live connection to COMSOL Multiphysics using the following steps:

- I Open the Model Navigator and select a new 3D model.
- 2 From the File menu select SolidWorks Connection>Initialize.
- **3** A progress window opens, and the connection translates the SolidWorks model to a COMSOL Multiphysics geometry.

Now modify the SolidWorks model, which is mirrored in the COMSOL Multiphysics geometry.

I In the feature tree on the left side of the SolidWorks window, locate the symbol called Equations. The constant fillet\_radius is currently set to 5. This value determines the size of the fillet that is slightly darker than the rest of the drawing.


- 2 Select Edit Equation by right-clicking on the text fillet\_radius.
- 3 Change the value to 1, and the fillet in the model becomes very small.
- 4 Return to the COMSOL Multiphysics window. Select the menu item
   File>SolidWorks Connection>Update to adjust the geometry to the changes you just made in SolidWorks.



As another exercise, assume that this radius is too small for your modeling needs, but you want to control its size from within COMSOL Multiphysics. This is possible and illustrates the connection's bidirectional nature.

I In COMSOL Multiphysics select the menu item **Options>Constants**.

2 Define a constant with the name fillet\_radius and in the corresponding **Expression** edit field and enter 4. Click **OK**.

Name	Expression	Value	Description	
fillet_radius	4	4		

- 3 Select the menu item File>SolidWorks Connection>Update.
- **4** This action updates the fillet in the COMSOL Multiphysics geometry to the new value. In addition, the connection updates the value of **fillet\_radius** and the drawing in SolidWorks.



# CAD Import Module Command Reference

### Summary of Commands

cad3part on page 38 cadconvert on page 39 caddetachfac on page 40 cadfindfillets on page 41 cadfindshortedg on page 42 cadfindsliverfac on page 43 cadfindsmallfac on page 44 cadfindspikes on page 45 cadknit on page 46 cadrepair on page 47 cadrmfac on page 49 cadrmfillets on page 50 cadrmshortedg on page 51 cadrmsliverfac on page 52 cadrmsmallfac on page 53 cadrmspikes on page 54 geomexport on page 56 geomimport on page 57 geomupdate on page 60 Swb0dy on page 61 Swinit on page 62

## Commands Grouped by Function

FUNCTION	PURPOSE
cadconvert	Convert CAD object to COMSOL object
caddetachfac	Detach faces from CAD object to form a new solid
cadfindfillets	Find fillets in CAD object
cadfindshortedg	Find short edges in CAD object
cadfindsliverfac	Find sliver faces in CAD object
cadfindsmallfac	Find small faces in CAD object
cadfindspikes	Find spikes in CAD object
cadknit	Knit faces and form solid CAD object
cadrepair	Repair CAD object
cadrmfac	Remove faces from CAD object
cadrmfillets	Remove fillets from CAD object
cadrmshortedg	Remove short edges from CAD object
cadrmsliverfac	Remove sliver faces from CAD object
cadrmsmallfac	Remove small faces from CAD object
cadrmspikes	Remove spikes from CAD object

#### CAD Repair and Defeaturing Functions

#### Geometry Functions

FUNCTION	PURPOSE
geomimport	Import geometry objects from file
geomupdate	Update geometry data in FEM structure
swinit	Initialize connection with SolidWorks

#### Geometry Objects

FUNCTION	PURPOSE
cad3part	CAD geometry object
swbody	SolidWorks imported body

Purpose	Create a CAD geometry object.
Syntax	g = cad3part
Description	g = cad3part creates a CAD geometry object.
See Also	geomimport

Purpose	Convert CAD object to COMSOL object.
Syntax	g1 = cadconvert(g)
Description	g1 = cadconvert(g) converts a CAD geometry object to a COMSOL geometry object.
See Also	cad3part, geomimport

Purpose	Detach faces from CAD object to form a new solid.					
Syntax	[g1,] = caddetachfac(g,faces,)					
<b>Description</b> g1 = caddetachfac(g,faces) detaches the set of faces in the vector f the CAD object g (parent) and forms a new solid (child) with the detach The returned cell array of CAD objects g1 includes the modified parent the new child solid object.						
	This function supports the following properties:					
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION		
	Healparent	fill patch	patch	Healing method used on the parent body. For details see cadrmfac		
	Healchild	fill  patchchild  patchparent	patchparent	Healing method used on the child body. For details see below		
	Out	stx ftx  ctx ptx	none	Output variables. For more information see the entry for geomcsg		
Examples	With patchchild, the function will try to heal the wound on the child body by growing or shrinking the faces surrounding the wound on the child body. The patchparent property will do the same but with the surfaces on the parent body. The following example imports the COMSOL Multiphysics geometry file					
	<pre>defeaturing_demo_2.mphbin and detaches a hole defined by a set of faces: filepath = which('defeaturing_demo_2.mphbin'); g = geomimport(filepath); geomplot(g{1}); faces=[6 7 8 9 11 12 13]; g1 = caddetachfac(g{1},faces); figure geomplot(g1{1}) figure geomplot(g1{2})</pre>			e defined by a set of faces: ohbin');		
See Also	cadrmfac					

Purpose	Find fillets in CAD object.				
Syntax	<pre>facesets = cadfindfillets(g,)</pre>				
Description	<pre>facesets = cadfindfillets(g) finds fillets in the CAD object g. Fillets are defined as a set of connected faces and are returned in a cell array with vector elements of face indices. All fillets and chains of fillets with a radius smaller than the property Entsize are returned. This function supports the following properties: TABLE 3-2: VALID PROPERTY/VALUE PAIRS</pre>				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Entsize	positive real scalar	see below	Maximum fillet radius	
Examples	The default for Entsize is calculated as the size of the object multiplied with $1 \cdot 10^{-3}$ . See the example under the function cadrmfillets.				
See Also	cadrmfillets,	cadrmfac			

Purpose	Find short edges in	n CAD object.		
Syntax	edgesets = cadfindshortedg(g,)			
Description	edgesets = cadf	indshortedg(g) find	s short edges	s in the CAD object g.
	The returned cell array edgesets contains chains of short edges as elements. Each element is a matrix with the edge indices in the first row and the corresponding edge lengths in the second row. This function supports the following properties: TABLE 3-3: VALID PROPERTY/VALUE PAIRS			
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
	Entsize	positive real scalar	see below	Maximum edge length
	The default for $En^{-3}$ .	tsize is calculated as t	he size of the	e object multiplied with
Examples	See the example un	nder the function cadr	mshortedg.	
See Also	cadrmshortedg			

Purpose	Find sliver faces in G	CAD object.			
Syntax	<pre>faces = cadfindsliverfac(g,)</pre>				
Description	faces = cadfinds	liverfac(g) finds and a	returns sliver	faces in the CAD object g.	
	The returned vector faces contains the face indices.				
	Sliver faces are narrow but long faces with large aspect ratio, which usually give rise to extremely fine local meshes in their vicinity This function supports the following properties:				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Entsize	positive real scalar	see below	Maximum face width	
Examples	The default for Entsize is calculated as the size of the object multiplied with $1 \cdot 10^{-3}$ . See the example under the function cadrmsliverfac.				
See Also	cadrmsliverfac, d	cadrmfac			

Purpose	Find small faces in	CAD object.			
Syntax	<pre>faces = cadfindsmallfac(g,)</pre>				
Description	faces = cadfindsmallfac(g) finds small faces in the CAD object g.				
	A small face is a face that fits within a sphere of specified radius, given in the pro Entsize. The returned vector contains the face indices. This command supports the following properties:				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Entsize positive real scalar see below Maximum face size				
	The default for Ent $1.10^{-3}$ .	tsize is calculated as the	e size of the	object multiplied with	
Examples	See the example un	nder the function cadrms	smallfac.		
See Also	cadrmsmallfac,c	adrmfac			

Purpose	Find spikes in CA	D object.			
Syntax	<pre>spikes = cadfindspikes(g,)</pre>				
<b>Description</b> spikes = cadfindspikes(g) finds spikes in the CAD object g.				) object g.	
	The output spike element:	s is a cell array of struct	ture elements.	There are two fields in each	
	<ul> <li>the edges field, which is a vector with the edge indices of edges that define a spike</li> <li>the width field, which contains the width of the spike</li> </ul>				
	A spike is a long and narrow protrusion on an edge or corner of a face defined by two or three edges.				
	This function supports the following properties:				
	TABLE 3-6: VALID PROPERTY/VALUE PAIRS				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Entsize	positive real scalar	see below	Maximum spike width	
	The default for Entsize is calculated as the size of the object multiplied with $1 \cdot 10^{-3}$ .				
Examples	See the example u	nder the function cadr	mspikes.		
See Also	cadrmspikes				

Purpose	Knit face CAD objects to solid.				
Syntax	[g1,] = cadknit(faces,)				
Description	<ul> <li>g1 = cadknit(faces) knits the faces in the cell array faces and tries to form a solid CAD object. The new face and/or solid objects are returned in the cell arrag1.</li> <li>This function also removes gaps and spikes that are within the absolute tolerance specified in the property Repairtol.</li> </ul>				
	This function supports the following properties:				
	TABLE 3-7: VALID PRO	PERTY/VALUE PAIRS			
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Repairtol	positive real scalar	1e-4	Absolute repair tolerance	
	Out	stx ftx ctx  ptx	none	Output variables. For more information see the entry for geomcsg.	
Examples	The following example imports the file repair_demo_2.x_b, and knits the object into a solid. A gap is also removed during the operation.				
	/hich('repair_de ort(filepath); (g,'Repairtol',	mo_2.x_b'); 1e-3);			
See Also	cadrepair				

Repair CAD object.

Syntax

Purpose

```
[g1,...] = cadrepair(g,...)
```

Description

g1 = cadrepair(g) repairs the CAD object g.

This function supports the following properties:

TABLE 3-8:	VALID	PROPERT	Y/VALUE	PAIRS

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
Repairtol	positive real scalar	1e-4	Absolute repair tolerance
Out	stx ftx  ctx ptx	none	Output variables. For more information see the entry for geomcsg.

The function tries to remove or repair the following defects:

- Topology with invalid sense
- Invalid edge and vertex tolerances
- Invalid manifold
- Self-intersecting manifold
- Non-G1 manifold
- Missing edge or vertex manifold
- Missing vertex
- Vertices not on curve of edge
- Edges and vertices not on surface of face
- Removal of small edges
- 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
- Recalculates incorrect or missing curves for edges
- Remove small features (edges, faces, edge pinches and spikes)

The function also performs the following simplifications:

- Simplification of B-splines to analytic representations
- · Simplification and merging of intersection curves

- Simplification of swept and spun surfaces to analytic
- Reducing degree of high order B-splines

See Also

cadknit

Purpose	Remove faces fro	Remove faces from CAD object.				
Syntax	[g1,] = ca	[g1,] = cadrmfac(g,faces,)				
Description	g1 = cadrmfac vector faces fro	g1 = cadrmfac(g,faces) removes the faces specified with face indices in the vector faces from the CAD object g and returns a CAD object g1.				
	Only fillets that o	can be removed	without inval	idating the CAD object are removed.		
	This function su	pports the follow	ving propertie	es:		
	TABLE 3-9: VALID PRO	OPERTY/VALUE PAIRS				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION		
	Heal	fill patch	patch	Healing method used on the parent body		
	Out	stx ftx  ctx ptx	none	Output variables. For more information see the entry for geomcsg.		
	With the Heal put the removal of fa property is set to based on the sur	With the Heal property set to patch the function tries to heal the wound caused by the removal of faces by shrinking and/or growing the surrounding faces. If the Heal property is set to fill the function tries to calculate a surface to cover the wound based on the surrounding edges.				
Examples	<pre>The following example imports the file defeaturing_demo_2.mphbin, an removes a hole from the geometry model:     filepath = which('defeaturing_demo_2.mphbin');     g = geomimport(filepath);     geomplot(g{1}, 'Facemode', 'off');     faces=[6 7 8 9 11 12 13];     g1 = cadrmfac(g{1}, faces);     figure     geomplot(g1, 'Facemode', 'off')</pre>					

See Also caddetachfac

Purpose	Remove fillets from CAD object.				
Syntax	<pre>[g1,] = cadrmfillets(g,facesets,)</pre>				
Description	<pre>g1 = cadrmfillets(g,facesets) removes fillets specified in facesets from CAD object g.</pre>				
	The cell array facesets contains vector elements with face indices of the fillets or fillet chains. This is typically the output from a call to the cadfindfillets command.				
	Only faces that ca	n be removed w	vithout invalidat	ing the object are removed.	
	This command su	pports the follo	wing properties	:	
	TABLE 3-10: VALID PRO	OPERTY/VALUE PAIRS			
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Out	stx ftx  ctx ptx  status	none	Output variables. For more information see the entry for geomcsg.	
	With the Out property set to status the function also returns a vector whose elements indicate if the operation succeeded for the corresponding elements in facesets. Ones and zeros indicate success and failure, respectively.				
Examples	<pre>mples The following example imports the CAD object in the COMSOL Multiphys geometry file defeaturing_demo_3.mphbin and finds and removes all fillets a radius less than 4.10<sup>-3</sup>:     filepath = which('defeaturing_demo_3.mphbin');     g = geomimport(filepath);     geomplot(g{1});     facesets=cadfindfillets(g{1},'Entsize',4e-3);     g1 = cadrmfillets(g{1},facesets);     figure     geomplot(g1)</pre>				
See Also	cadfindfillets, cadrmfac				

Purpose	Remove short edges from CAD object.
---------	-------------------------------------

Syntax

Description

g1 = cadrmshortedg(g,edgesets) removes short edges, that are specified in edgesets, from the CAD object g.

g = cadrmshortedg(g,edgesets,'Entsize',length) removes edges, which are specified in edgesets, and that have a length shorter than length, from the CAD object g.

The elements of the cell array edgesets are matrices with the indices of the edges in the first row and the corresponding length in the second row. This is typically the output from a call to the cadfindshortedg function. For a single edge chain, edgesets can also be a matrix with two rows.

Only edges that can be removed without invalidating the geometry are removed.

This function supports the following properties:

[g1,...] = cadrmshortedg(g,edgesets,...)

TABLE 3-11: VALID PROPERTY/VALUE PAIRS

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
Entsize	positive real scalar	<pre>max(edgesets{:,1} (2,:))*1.0001</pre>	Length of short edge
Out	stx ftx  ctx ptx  status	none	Output variables. For more information see the entry for geomcsg.

With the Out property set to status the function also returns a vector whose elements indicate if the operation succeeded for the corresponding elements in edgesets. Ones and zeros indicate success and failure, respectively.

```
Examples The following example imports the geometry model from the file
defeaturing_demo_4.x_b, finds edges shorter than 3.10<sup>-3</sup>, and finally deletes
those edges that are shorter than 3.10<sup>-4</sup>:
    filepath = which('defeaturing_demo_4.x_b');
    g = geomimport(filepath);
    geomplot(g{1}, 'Edgelabel', 'On');
    edgesets=cadfindshortedg(g{1}, 'Entsize', 3e-3);
    g1 = cadrmshortedg(g{1}, edgesets, 'Entsize', 3e-4);
    figure
    geomplot(g1, 'Edgelabel', 'On')See Also cadfindshortedg
```

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Purpose	Remove sliver faces from CAD object.					
Syntax	<pre>[g1,] = cadrmsliverfac(g,faces,'Entsize',width,)</pre>					
Description	<pre>g1 = cadrmsliverfac(g,faces,'Entsize',width) removes sliver faces, which are specified in faces, and that are narrower than width, from the CAD object g. The vector faces contains face indices, and is typically the output from a call to the function findsliverfac.</pre>					
	Only faces that removed.	Only faces that can be removed without invalidating the geometry object are removed.				
	This function su	pports the following	properties:			
	TABLE 3-12: VALID P	ROPERTY/VALUE PAIRS				
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION		
	Entsize	positive real scalar	-	Width of sliver face		
	Out	stx ftx ctx  ptx status	none	Output variables. For more information see the entry for geomcsg.		
	With the Out property set to status the function also returns a vector whose elements indicate if the operation succeeded for the corresponding elements in faces. Ones and zeros indicate success and failure, respectively.					
Examples	The following e	xample imports the g	eometry mod	el from the file		
·	defeaturing_c	1emo_5.x_b, finds sliv	ver faces narro	ower than $2 \cdot 10^{-3}$ , and deletes		
	those edges that	t are narrower than $2 \cdot$	$10^{-5}$ :			
	<pre>filepath = which('defeaturing_demo_5.x_b'); g = geomimport(filepath); geomplot(g{1},'Facelabel','On','Facemode','Off'); faces = cadfindsliverfac(g{1},'Entsize',2e-3); [g1 relg1tog] = cadrmsliverfac(g{1},faces, 'Entsize',2e-5,'Out','ftx'); figure geomplot(g1,'Facelabel','On','Facemode','Off') Connectivity information between faces of the modified and original body is returned in the variable relg1tog. This variable is a sparse matrix with rows and</pre>					
	columns corresp	oonding to face indices	s in the bodie	s g1 and g{1}, respectively. The		
	filled positions i	ndicate a connection	between the c	corresponding faces.		
See Also	cadfindsliverfac, cadrmfac					

Purpose	Remove small faces from CAD object.			
Syntax	<pre>[g1,] = cadrmsmallfac(g,faces,'Entsize',size,)</pre>			
Description	g1 = cadrmsmallfac(g,faces) removes small faces, that are specified in faces, from the CAD object g.			
	<pre>g1 = cadrmsmallfac(g,faces,'Entsize',size) removes small faces, that are specified in faces, and that are smaller than size, from the CAD object g.</pre>			

The vector faces contains face indices, and is typically the output of a call to the function cadfindsmallfac.

Only faces that can be removed without invalidating the CAD object are removed.

This function supports the following properties:

TABLE 3-13: VALID PROPERTY/VALUE PAIRS

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
Entsize	positive real scalar	-	Size of small face
Out	stx ftx  ctx ptx  status	none	Output variables. For more information see the entry for geomcsg.

With the Out property set to status the function also returns a vector whose elements indicate if the operation succeeded for the corresponding elements in faces. Ones and zeros indicate success and failure, respectively.

Examples	The following example imports the geometry model from the file defeaturing_demo_6.x_b, finds faces smaller than $10^{-3}$ , and deletes those faces that are smaller than $10^{-4}$ :
	<pre>filepath = which('defeaturing_demo_6.x_b'); g = geomimport(filepath); geomplot(g{1},'Facelabel','On','Facemode','Off'); faces = cadfindsmallfac(g{1},'Entsize',1e-3); g1 = cadrmsmallfac(g{1},faces,'Entsize',1e-4); figure geomplot(g1,'Facelabel','On','Facemode','Off')</pre>
See Also	cadfindsmallfac, cadrmfac

Purpose	Remove spikes from CAD object.				
Syntax	[g1,] = cadrmspikes(g,spikes,)				
Description	<ul> <li>g1 = cadrmspikes(g, spikes) removes specified spikes from the CAD object g.</li> <li>g1 = cadrmspikes(g, spikes, 'Entsize', width) removes specified spikes, that have a width smaller than or equal to width, from the CAD object g.</li> <li>The cell array spikes is typically the output from a call to the function cadfindspikes and has structure elements. There are two fields in each element:</li> </ul>				
	• the edges fiel	d, which is a vector	with the edge in	ndices of edges that define a spike	
	• the width fiel	d, which contains	the width of the	e spike.	
	Only spikes that	can be removed wi	thout invalidati	ng the CAD object are removed.	
	This function su	pports the followin	ng properties:		
	TABLE 3-14: VALID PF	OPERTY/VALUE PAIRS			
	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Entsize	positive real scalar	max(spikes {:}.width) *1.00001	Width of spike	
	Out	stx ftx ctx  ptx status	none	Output variables. For more information see the entry for geomcsg.	
Examples	<pre>With the Out property set to status the function also returns a vector whose elements indicate if the operation succeeded for the corresponding elements in spikes. Ones and zeros indicate success and failure, respectively. The following example imports the geometry model from the file defeaturing_demo_7.x_b, finds and deletes spikes narrower than 10<sup>-4</sup>: filepath = which('defeaturing_demo_7.x_b'); g = geomimport(filepath); meshg = meshinit(g{1}); meshplot(meshg); spikes = cadfindspikes(g{1}, 'Entsize', 1e-4); g1 = cadrmspikes(g{1}, spikes); meshg1 = meshinit(g1); figure meshplot(meshg1) Visualize the spike by meshing the object g{1}: an extremely fine mesh is generated</pre>				

See Also cadfindspikes

Purpose	Export geometry objects to file.
Syntax	geomexport(filename, geoms,)
Description	<pre>geomexport(filename, geoms) exports the geometry data in the cell array geoms of geometry objects to a file.</pre>

filename can be any of the following formats:

TABLE 3-15: VALID FILE FORMATS

FILE FORMAT	NOTE	FILE EXTENSIONS
COMSOL Multiphysics Binary		.mphbin
COMSOL Multiphysics Text		.mphtxt
Parasolid Binary	1	.x_b
Parasolid Text	1	.x_t
DXF		.dxf

Note 1: This format requires a license for the COMSOL CAD Import Module. Only geometries imported using the CAD Import Module (cad3part objects) can be exported to Parasolid format.

The following properties are supported

TABLE 3-16: VALID PROPERTY/VALUE PAIRS

	PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION	
	Report	on off	on	Display a progress window	
	Report determin	Report determines if a progress window appears during the call.			
Examples	The following ex	xample import	s the DXF file	demo1.dxf as a curve2 object:	
	<pre>filepath = which('demo1.dxf');</pre>				

g	=	geomimport(filepath);	
ge	eon	<pre>nplot(g{1})</pre>	

**Diagnostics** geomexport replaces the functionality of the 3.1 function dxfwrite.

See Also geom0, geom1, geom2, geom3, meshimport, geomexport, cad3part, cadrepair

Purpose Import

Syntax

Import geometry objects from a file.

gl = geomimport(filename,...)

Description

gl = geomimport(filename,...) reads the geometry file filename and translates the geometry data using the specified properties into a cell array of CAD geometry objects gl.

filename can be any of the following formats:

FILE FORMAT	NOTE	FILE EXTENSIONS	OBJECT TYPE IN COMSOL MULTIPHYSICS
Autodesk Inventor	2	.ipt	CAD object
CATIA V4	2	.model	CAD object
CATIA V5	2	.CATPart, .CATProduct	CAD object
COMSOL Multiphysics Binary		.mphbin	CAD and/or COMSOL objects
COMSOL Multiphysics Text		.mphtxt	CAD and/or COMSOL objects
DXF		.dxf	COMSOL object
GDS	3	.gds	COMSOL object
IGES	I	.igs,.iges	CAD object
Parasolid	I	.x_t,.x_b	CAD object
Pro/ENGINEER	2	.prt,.asm	CAD object
SAT	I	.sat,.sab	CAD object
STEP	I	.step,.stp	CAD object
STL		.stl	COMSOL object
VDA-FS	2	.vda	CAD object
VRML		.wrl,.vrml	COMSOL object

TABLE 3-17: SUPPORTED FILE FORMATS

Note 1: This format requires a license for the COMSOL CAD Import Module.

Note 2: This format requires a license for a format-specific module from COMSOL.

Note 3: This format requires a license for COMSOL Script.

This function supports the following properties:

PROPERTY NAME	PROPERTY VALUE	DEFAULT	DESCRIPTION
Coercion	solid face  curve point  off	solid	Coerce the imported geometry
Keepbnd	on off	on	Keep boundary entities
Keepfree	on off	off	Keep free edge/point entities
Keepsolid	on off	on	Keep solid entities
Repair	on off	on	Repair imported data
Repairtol	positive scalar	1e-4	Repair tolerance
Importtol	positive scalar	1e-5	Absolute repair tolerance used when importing CAD Import Module formats.
Report	on   off	on	Display a progress window

TABLE 3-18: VALID PROPERTY/VALUE PAIRS

When importing COMSOL Multiphysics files and those in the GDS format, the function ignores all properties, except for Report.

For DXF import, the default for Coercion is curve.

For STL and VRML imports, the command supports all properties in the function meshenrich.

Coercion can force the import process to knit boundary segments together and possibly try to form solid entities.

Keepbnd, Keepfree, and Keepsolid indicate which type of entities the module should consider in the imported data.

**Repair** determines if the module should process the imported data to improve the quality. See the cadrepair function for details on the type of repair that is done to geometric data.

**Repairtol** is a relative tolerance. It indicates the size of entities to remove, which points to snap together, and similar features.

Importtol is an absolute tolerance. It indicates the size of entities to remove, which points to snap together, and similar features. It is used when importing a file using the CAD Import module, replacing Repairtol.

Report determines if a progress window should appear during the call.

Examples	The following example imports the DXF file demo1.dxf as a curve2 object:
	<pre>filepath = which('demo1.dxf'); g = geomimport(filepath); geomplot(g{1})</pre>
Diagnostics	geomimport replaces the functionality of the COMSOL Multiphysics 3.0 functions dxfread, igesread, stlread, and vrmlread. We no longer support those functions and their properties and thus do not document them.
See Also	geomO, geom1, geom2, geom3,meshimport,geomexport,cad3part, cadrepair

Purpose	Update geometry data contained in an FEM structure.			
Syntax	fem = geomupdate(fem,)			
Description	fem = geomupdate(fem,) updates geometry objects contained in the FEM structure fem.			
	If fem contains geometry objects of type swbody and there is an existing SolidWorks connection, the SolidWorks geometry is updated according to variables the geomupdate function passes, and these bodies are synchronized with SolidWorks.			
	Use geomanalyze to update the rest of the FEM structure after updating the geometry objects.			
	This function supports the following properties:			
	TABLE 3-19: VALID PROPERTY/VALUE PAIRS			
	PROPERTY NAME PROPERTY DEFAULT DESCRIPTION			
	Const	cell array		Definition of constants
	Geomnum	integer	1	Number of the FEM structure in an extended FEM structure

This function also supports all properties of geomimport.

See Also geomanalyze, geomimport

Purpose	Create a SolidWorks synchronized geometry object.
Syntax	s3 = swbody
Description	s3 = swbody creates a SolidWorks synchronized geometry object.
	A swbody is a geom3 object that is generated from a SolidWorks connection and that can be updated from SolidWorks by calling geomupdate.
	If there is no active SolidWorks connection, the swbody object acts as an ordinary geom3 object.
See Also	geomO, geom1, geom2, geom3,geomupdate

Purpose	Initialize a connection with SolidWorks.
Syntax	<pre>gl = swinit(doc,)</pre>
Description	gl = swinit(doc,) returns a cell array of SolidWorks bodies obtained from the open SolidWorks document doc. If doc is an empty string, the active open document is used.
See Also	geomanalyze, geomupdate

## Glossary

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This glossary contains CAD-specific terms that describe file formats and software packages in a general context. For geometric terms specific to COMSOL Multiphysics, see the glossary in the *COMSOL Multiphysics User's Guide*. To find references that point to more information about a term, see the index.

### Glossary of Terms

ACIS A geometry kernel from Spatial Corp. with SAT as its native file format.

AutoCAD A 2D CAD program from Autodesk, Inc.

**Autodesk Inventor** Name of both a CAD program and its native file format (.ipt). from Autodesk, Inc. The kernel of this CAD program is based on *ACIS*.

**CAD** Computer-aided design.

**CATIA V4** Name of both a CAD program and its native file format (.model) from Dassault Systèmes S.A. It is fundamentally different to *CATIA V5*.

**CATIA V5** Name of both a CAD program and its native file format (.CATPart, .CATProduct) from Dassault Systèmes S.A. It is fundamentally different to *CATIA V4*.

**defeaturing** The process of removing details in the CAD geometry in order to reduce the size of the finite element model.

**DXF** The 2D file format (.dxf) native to *AutoCAD* from Autodesk, Inc.

**GDS** A 2D file format (.gds) based on a neutral standard.

**IGES** A file format (.igs, .iges) based on a neutral standard not native to any geometry kernel.

**NX** A CAD program (formerly known as Unigraphics) from UGS Corp. using the *Parasolid* kernel.

**Parasolid** Name of both a geometry kernel and its native file format (.x\_t, .x\_b) from UGS Corp.

**Pro/ENGINEER** Name of both a CAD program and its native file format (.prt, .asm) from Parametric Technology Corp.

**SAT** The native file format (.sat, .sab) of the *ACIS* geometry kernel from Spatial Corp.

Solid Edge A CAD program from UGS Corp. using the *Parasolid* kernel.

SolidWorks A CAD program from SolidWorks Corp. using the *Parasolid* kernel.

**STEP** A file format (.step, .stp) based on a neutral standard not native to any geometry kernel.

**STL** A file format (.stl) based on a neutral standard using surface-mesh representations.

**VDA-FS** A file format (.vda) based on a neutral standard not native to any geometry kernel.

**VRML** A file format (.wrl, .vrml) based on a neutral standard using surface mesh representations.

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