INTRODUCTION TO

LiveLink™ *for* Revit®

5.5

COMSOL

Introduction to LiveLink™ *for* Revit®

Version: COMSOL 5.5

Contact Information

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

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

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

Part number: CM024502

Contents

Introduction 5
Synchronizing the Geometry 6
About CAD File Formats 21
Importing and Repairing a 3D CAD File 23
Working with Defeaturing Tools 30
Applying Virtual Geometry Operations 35
Creating a Fluid Domain Around a Solid Structure 44

Introduction

This guide introduces you to LiveLink™ *for* Revit®, which provides two possibilities for applying multiphysics analysis in the development of your designs. To begin with, you can synchronize geometries between Revit® Architecture and the COMSOL Desktop® when running them side-by-side. In addition, the product adds support for importing several 3D CAD file formats into your COMSOL models.

Regardless of the way you bring CAD designs into COMSOL, with LiveLink™ you have a robust platform, including repair and defeaturing tools, to prepare the geometry for multiphysics modeling. The detailed tutorials that follow start you off with becoming efficient in using the provided functionality.

Synchronizing the Geometry

The synchronization of geometry between Revit® Architecture and COMSOL is associative, and can be configured to generate selections in the COMSOL model for elements in the Revit project. The selections are available for setting up the analysis, for example for physics interface or mesh settings. Working with selections also retains the model settings should you change the design in Revit and re-synchronize.

This tutorial gives a quick introduction to the workflow when synchronizing the geometry of a room, and preparing it for analysis. Although setting up the physics interface is not part of the tutorial, the geometry that is generated may be used for example for acoustics simulation of the room.

The following steps are included in the exercise:

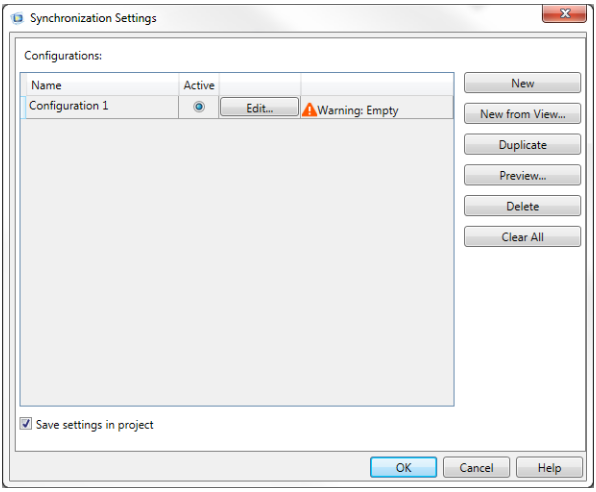
- Configuring the synchronization of the Revit project
- Creating a COMSOL model containing a LiveLink node
- Synchronizing the geometry
- Preparing the synchronized geometry for analysis

Configuring the Synchronization

1. In Revit Architecture open the file `house.rvt`, which is found in your COMSOL installation directory, under the folder `applications/LiveLink_for_Revit/Tutorials`.

   The example house project contains several rooms. In the following you set up the synchronization of the living room together with the elements within the room.
2. In the Revit Project Browser double click the Floor Plans > Level 1 view to switch to that view.

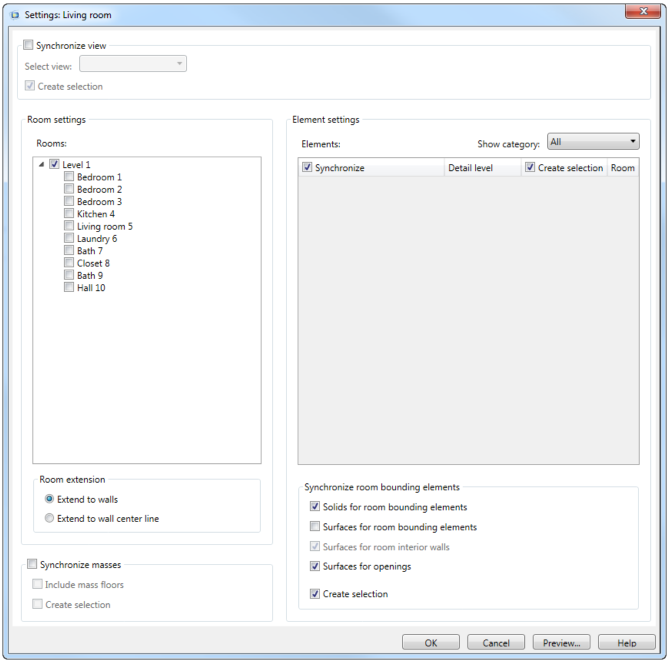
3 On the COMSOL Multiphysics tab click Synchronization Settings.



When you open the Synchronization Settings dialog box for the first time in a project an empty synchronization configuration is automatically added to the Configurations table. To enable synchronization of the project there needs to be at least one configuration with content.

4 Double click Configuration 1, in the Name column, then edit the text to change the name to Living room.

5 Click the Edit button to open the Settings dialog box for the Living room configuration.

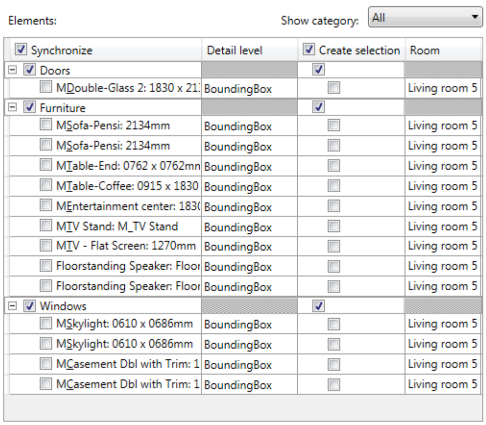


The Rooms list to the left contains the rooms defined in the project. To include the geometry of a room in the synchronization select its check box.

6 Select the Living room 5 check box.

According to the default setting in the Room extension section, the solid object generated for the room volume extends to the walls, and from the floor to the roof or ceiling.

The elements belonging to selected rooms are automatically displayed in the Elements table.



Elements: Show category: All

Synchronize	Detail level	Create selection	Room
Doors		☑	
MDouble-Glass 2: 1830 x 21	BoundingBox	☐	Living room 5
Furniture		☑	
MSofa-Pensi: 2134mm	BoundingBox	☐	Living room 5
MSofa-Pensi: 2134mm	BoundingBox	☐	Living room 5
MTable-End: 0762 x 0762mn	BoundingBox	☐	Living room 5
MTable-Coffee: 0915 x 1830	BoundingBox	☐	Living room 5
MEntertainment center: 183(	BoundingBox	☐	Living room 5
MTV Stand: M_TV Stand	BoundingBox	☐	Living room 5
MTV - Flat Screen: 1270mm	BoundingBox	☐	Living room 5
Floorstanding Speaker: Floor	BoundingBox	☐	Living room 5
Floorstanding Speaker: Floor	BoundingBox	☐	Living room 5
Windows		☑	
MSkylight: 0610 x 0686mm	BoundingBox	☐	Living room 5
MSkylight: 0610 x 0686mm	BoundingBox	☐	Living room 5
MCasement Dbl with Trim: 1	BoundingBox	☐	Living room 5
MCasement Dbl with Trim: 1	BoundingBox	☐	Living room 5

Here you select the elements to include in the synchronization, and their geometrical representation (Detail level), for which two choices are available:

- Bounding Box: The enclosing box of the element is generated for transfer during synchronization. Using this option allows you to create, using the drawing tools in COMSOL, a new, simplified, geometry for elements that are not suitable for simulation.
- Original: The geometry of the element is transferred as is during synchronization. Using this option may require defeaturing of the geometry using defeaturing tools in COMSOL.

There is also a choice in the table for creating a selection for the element in the COMSOL model during synchronization.

7 To synchronize all elements belonging to the Furniture category right-click somewhere in the Elements table, then from the context menu select Synchronize>Select all>Category>Furniture. The check boxes for the individual furniture elements become selected.

8 Right-click again in the table, then from the menu select Create selection>Select all>Category>Furniture. The check boxes for the individual furniture elements in the Create selection column become selected.

Finally, change the element representation to the original geometry for all furniture elements.

9 Right-click in the table, then from the menu select Detail level>Original>Select all>Category>Furniture.

The synchronization of solids for the room bounding elements, such as the walls, roofs, and floors is turned on by default in the Synchronize room bounding elements section. Here you also see that surfaces are going to be generated for openings such as doors and windows. These elements are useful due to the associated selections that can be used in the model set up after synchronization.

10 Click OK to confirm the settings and close the Settings dialog box for the Living room.

In the Synchronization Settings dialog box the Living room configuration is no longer empty, and can be used for synchronization.

11 Click OK to close the Synchronization Settings dialog box.

Starting a COMSOL Model

1 Switch to the COMSOL Desktop, and start a new model.

Note that the project in Revit needs to remain open for the synchronization to work.

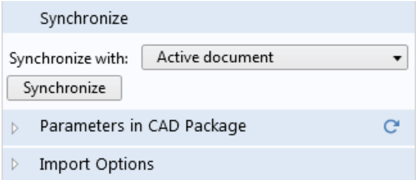
2 In the New window click Blank Model to skip the steps of selecting physics interfaces and study type.

3 On the Home toolbar, click Add Component and choose 3D.

Adding a LiveLink Node to the Geometry

1 On the Home toolbar, click LiveLink then select LiveLink for Revit .

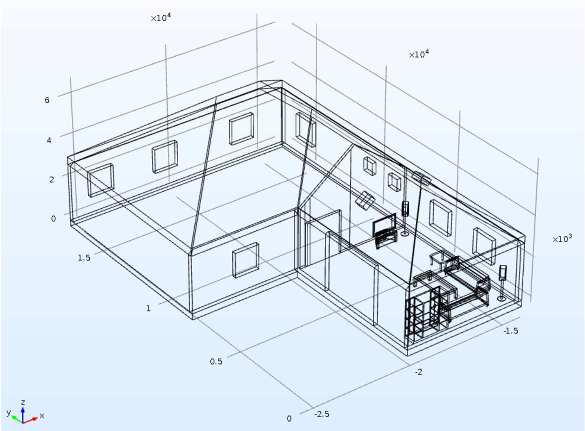
2 In the Settings window click Synchronize.



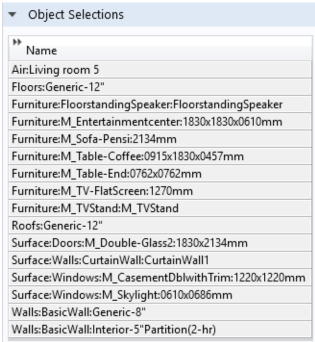
If the Revit window is hidden behind other windows on your desktop the Start Synchronization dialog box may appear. If this happens click OK to bring Revit to the front and start the synchronization. Since Revit Architecture only allows for the computations of the room geometry to start if a floor plan or section view is active, you may also have to switch to the Level 1 floor plan view in case the synchronization does not start.

During synchronization the geometry for the living room and the selected elements is generated in Revit and transferred to the COMSOL model.

3 On the Graphics toolbar click Wireframe Rendering.



4 In the Settings window, expand the Object Selections section.



These selections are on the object level, and clicking on a selection in the table highlights the corresponding objects in the Graphics window. The selections are available for input to operations and settings during all stages of the model set-up. Read more about how to work with selections in the *COMSOL Multiphysics Reference Manual*.

5 Click on the selections to see the corresponding objects highlighted in the Graphics window.

Working with Selections

Start with creating union selections to be used as inputs to the geometry operations that will create the final geometry for the simulation.

Selection for the Room Bounding Elements

1 From the Geometry toolbar click Selections and choose Union Selection .

2 In the Settings window for Union Selection enter `Room Bounding Solids` in the Label text field.

3 From the Level list select Object.

4 Click the Add button under the Selections to add table.

5 From the Add dialog box select the selections shown in the table below:

SELECTIONS TO ADD
`Floors:Generic-12”`
`Roofs:Generic-12”`
`Walls:BasicWall:Generic-8”`
`Walls:BasicWall:Interior-5”Partition(2-hr)`

To select several selections you can hold down the Ctrl button while clicking on the selections.

6 Click OK to close the Add dialog box.

Selection for the Room

1 From the Geometry toolbar click Selections and choose Union Selection.

2 In the Settings window for Union Selection enter `Room` in the Label text field.

3 From the Level list select Object.

4 Click the Add button under the Selections to add table.

5 From the Add dialog box select the selections shown in the table below:

SELECTIONS TO ADD
`Air:Living room 5`
`Surface:Doors:M_Double-Glass2:1830x2134mm`
`Surface:Walls:CurtainWall:CurtainWall1`
`Surface:Windows:M_CasementDblwithTrim:1220x1220mm`
`Surface:Windows:M_Skylight:0610x0686mm`
`Room Bounding Solids`

6 Click OK to close the Add dialog box.

Selection for the Furniture

1 From the Geometry toolbar click Selections and choose Union Selection.

2 In the Settings window for Union Selection enter `Furniture` in the Label text field.

3 From the Level list select Object.

4 Click the Add button under the Selections to add table.

5 From the Add dialog box select all the selections for the furniture as shown below:

SELECTIONS TO ADD
Furniture:FloorstandingSpeaker:FloorstandingSpeaker
Furniture:M_Entertainmentcenter:1830x1830x0610mm
Furniture:M_Sofa-Pensi:2134mm
Furniture:M_Table-Coffee:0915x1830x0457mm
Furniture:M_Table-End:0762x0762mm
Furniture:M_TV-FlatScreen:1270mm
Furniture:M_TVStand:M_TVStand

6 Click OK to close the Add dialog box.

Creating the Computational Domain

Converting to a Single Solid

To be able to use the objects in the defined selections in the model set-up, they need to become part of the final geometry. To incorporate all objects into a single solid object use the Convert to Solid geometry operation.

1 From the Geometry toolbar click Conversions and choose Convert to Solid.

2 In the Settings window for Convert to Solid, from the Input objects list select Room.

3 From the Repair tolerance list select Relative, and check that the Relative repair tolerance is set to 1E-6.

4 Click Build All Objects.

Deleting Entities

Since only the room volume is needed at the end, delete the domains for the walls, roof and floor.

1 On the Geometry toolbar click Delete.

2 In the Settings window for Delete Entities, from the Geometric entity level list select Domain.

3 From the Selection list select Room Bounding Solids.

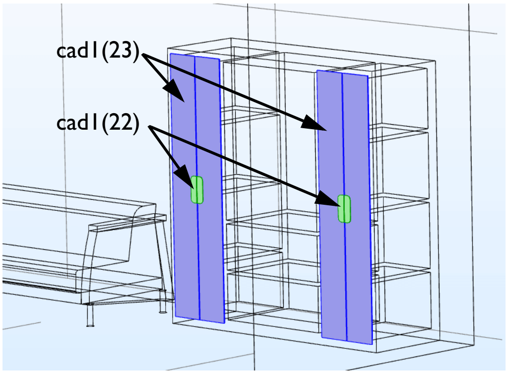
4 Click Build All Objects.

The next step is to delete the solid objects for the handles and the doors for the bookshelf in the room. The small details on these are not needed for the simulation. You will generate a new surface in place of the doors further ahead.

5 On the Geometry toolbar click Delete.

6 In the Settings window for Delete Entities, from the Geometric entity level select Object.

7 In the Graphics window select the two objects highlighted in the figure below.



8 Click Build All Objects.

Computing the Difference of the Room and Furniture

The last step before obtaining the computational volume is to subtract the objects for the furniture from the object for the room.

1 On the Geometry toolbar click Booleans and Partitions and choose Difference.

2 In the Settings window for Difference, from the Objects to add list select Room.

3 Click the Activate button under Objects to subtract.

4 From the Objects to subtract list select Furniture.

5 Click Build All Objects.

The solid for the room volume is now ready. Before meshing however, remove small features that remain in the geometry by using the defeaturing tools.

Defeaturing the Geometry

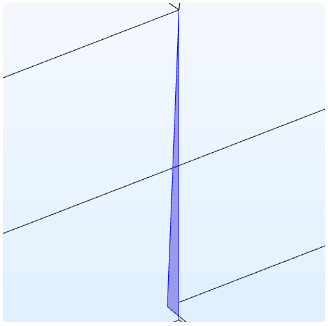
1 On the Geometry toolbar click Defeaturing and Repair and choose Delete Sliver Faces .

2 In the Maximum face width text field enter `5[mm]`.

3 Click Find Sliver Faces.

The 18 sliver faces that are detected are displayed in the Sliver face selection list.

4 Select for example `Sliver face 8` from the list, then click Zoom Selected next to the list to find the face in the Graphics window.



A sliver face is a face with a high aspect ratio. If it is not removed from the geometry it may cause problems during meshing. Remaining sliver faces in the list are located on the bezel of the flat screen TV, and some other on the two couches.

5 To remove the sliver faces click the Delete All button below the list.

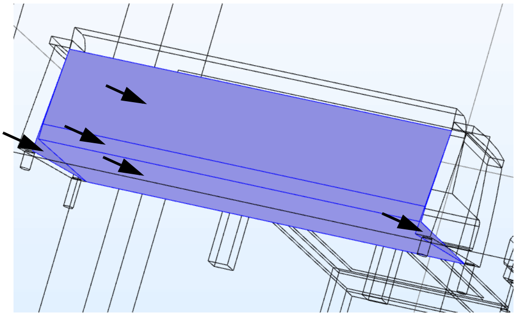
Continue with deleting some larger faces to further simplify the geometry, and also to create new faces on the front of the bookshelf.

6 From the toolbar in the Settings window for Delete Sliver Faces click the Delete Faces button.

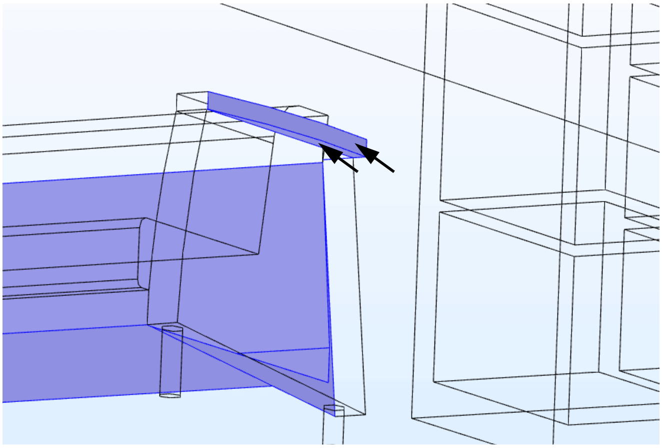
7 Click Activate next to the Faces to delete list.

The Delete Faces tool can delete faces from an object, and cover the resulting wound by growing (or shrinking) adjacent faces.

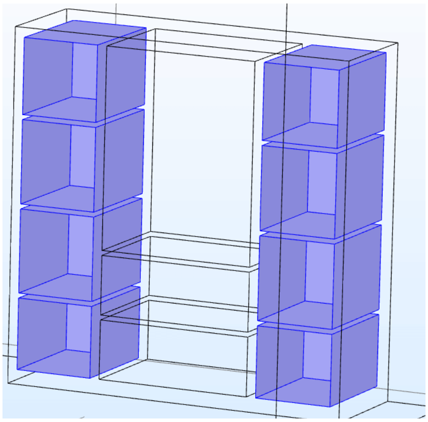
8 First, select, on both couches, the faces that are highlighted in the figure below. There are 5 faces located on the back and bottom of each couch.



9 Continue with selecting the highlighted faces on the sides of each armrest on the couches. There are 4 faces on each couch.



10 Finally select the highlighted faces on both sides of the bookshelf. There are 5 faces for each shelf.



11 Click Delete Selected.

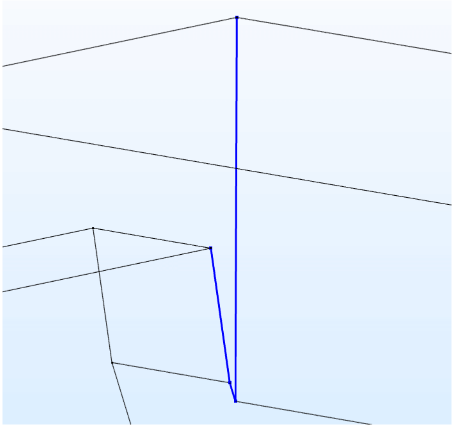
The last step before meshing is to find and delete spikes that still remain in the geometry. Similarly to a sliver face, a spike is a region in a face that has a high aspect ratio.

12 From the toolbar in the Settings window for Delete Faces click the Delete Spikes button.

13 In the Maximum spike width text field enter `10[mm]`.

14 Click the Find Spikes button.

15 The tool detected 4 spikes. Select `Spike 4` and click Zoom Selected to zoom in on the edges that comprise the spike.



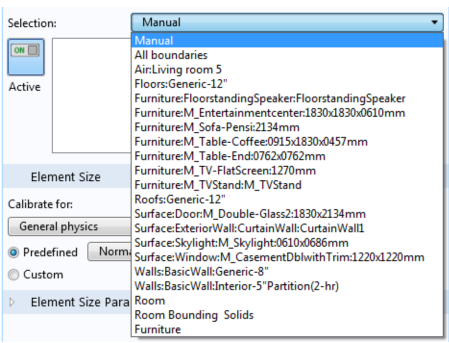
16 Click Delete All to remove all spikes.

17 To restore the default view click Go to Default 3D View on the Graphics toolbar.

Meshing the Geometry

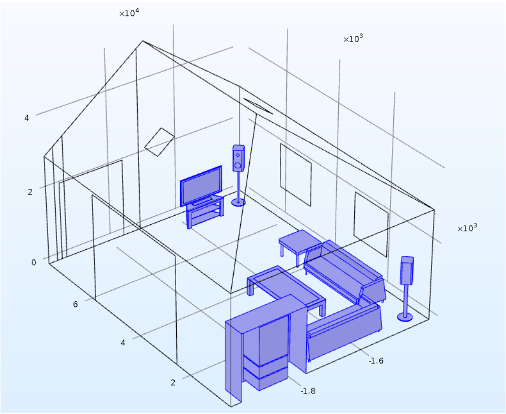
The following steps show how to create an unstructured tetrahedral mesh for the geometry using custom mesh size parameters on the faces for the furniture in the room. A mesh suitable for a simulation may be different from the one generated by following the steps below.

1 On the Mesh toolbar click Free Tetrahedral .
2 On the Mesh toolbar click Normal to add a Size attribute to the Free Tetrahedral 1 node.
3 In the Settings window for Size select Boundary from the Geometric entity level list.
4 Expand the Selection list.



The list of available selections includes the boundaries for the room bounding and furniture elements that were synchronized from the Revit project. The union selections created earlier in model are also available. In this case assume that to resolve the smaller boundaries of the furniture the Size 1 attribute node should apply to the Furniture selection.

5 From the Selection list select Furniture.



6 Click the Custom radio button.

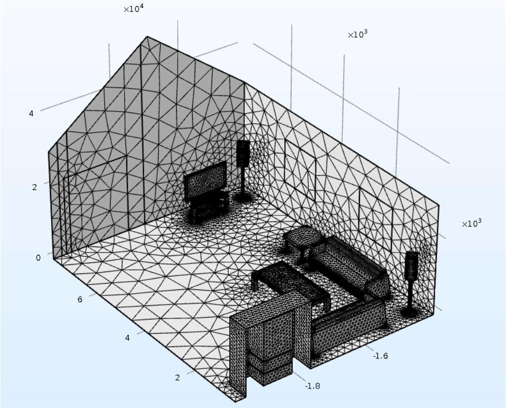
This allows the individual tuning of the mesh parameters. Change the maximum and minimum element size to more suitable values.

7 In the Element Size Parameters section select the Maximum element size check box and enter `100[mm]`.

8 Select the Minimum element size check box and enter 20[mm].

9 Click Build All.

10 To get the view below click the Click and Hide button on the Graphics toolbar, then click on the boundaries that obstruct the view to hide them. When done, click Hide and Select again to activate selection, instead of hiding, when clicking on the geometric entities.



About CAD File Formats

To better understand the file import related functionality of Revit®, first review some general background information about CAD file formats.

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® are the two most common kernels, and many CAD programs license these kernels. In addition, some programs use their own kernel (as does COMSOL). Each of these kernels has a native file format associated with it. For example, the Parasolid file format is simply called Parasolid, and the one from ACIS is called ACIS or 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. Revit® can read several of these different descriptions of objects and translate them into a format that COMSOL can work with.

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 such formats.

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 formats are VRML and STL.

Translating 3D CAD Files Between Formats

Geometric models do not always pass flawlessly between different file formats due to the fact that they are represented differently. This means that the quality of a translation when importing a file to COMSOL depends on the file format. The smoothest way is to use the native format of your CAD system. If this is not an option, we in general recommend that you use Parasolid, STEP, or ACIS.

Importing 3D CAD files into COMSOL is straightforward. Since the settings of the import operation have been tuned to suit the most common cases, the majority of files import simply with the click of a button. During import the geometry is checked for errors and automatically repaired. The repair operation also removes small features that fall within the import tolerance.

Importing and Repairing a 3D CAD File

In this example, the Parasolid® file of a wheel rim contains a few small faces and slivers, which are not removed during import, since they fall outside the default import tolerance. The step-by-step instructions below demonstrate one way to locate and remove these features. The general workflow is:

- Import the file
- Create a mesh for quick examination of the geometry
- Measure the size of the features you would like to remove
- Repair the object
- Create a new mesh for comparison

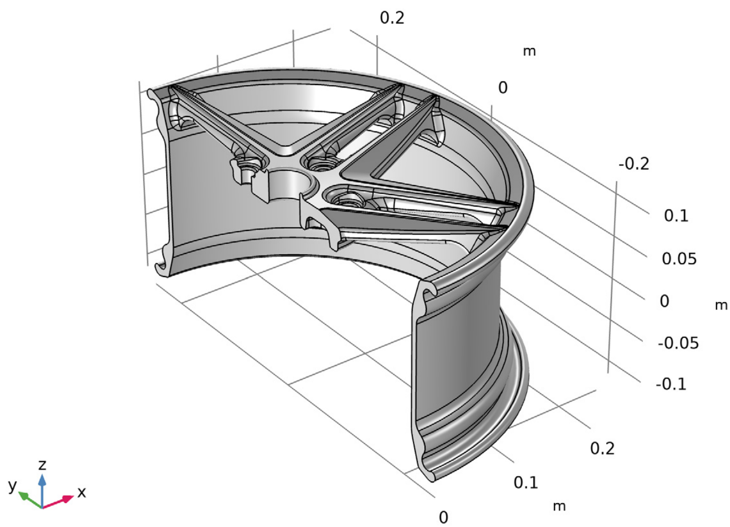
Model Wizard

1. Start COMSOL Multiphysics.
2. Select Blank Model to skip the steps of selecting physics interfaces and study type.
3. On the Home toolbar, click Add Component and select 3D.

Importing the Geometry

1. On the Home toolbar click Import .
2. In the Settings window for Import click the Browse button.
3. In your COMSOL installation directory navigate to the folder `applications/LiveLink_for_Revit/Tutorial_Examples` and double click the file `repair_demo_1.x_b`.
4. Click Import.

As soon as the import is done the geometry appears in the Graphics window.



Creating a Surface Mesh

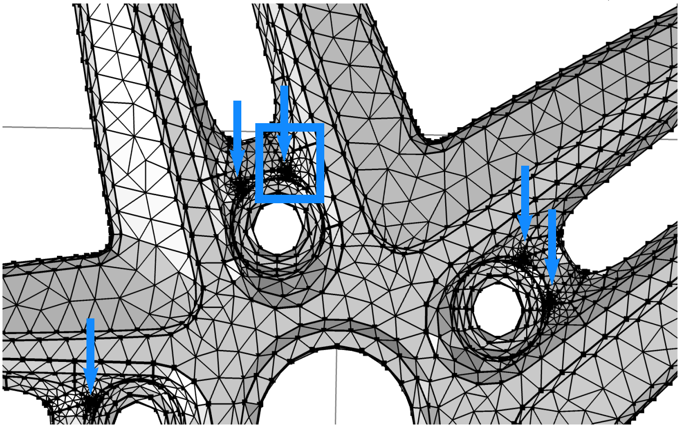
Creating a surface mesh for an imported solid is often the fastest way to assess the quality of the geometry and to identify regions needing repair or defeaturing.

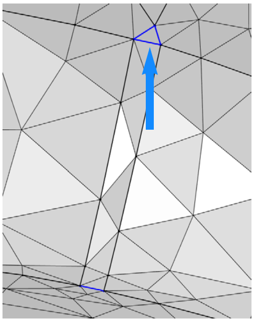
1. On the Mesh toolbar click Boundary and choose Free Triangular.
2. Go to the Settings window for Free Triangular and from the Selection list box select All boundaries.
3. Click the Build All button to create the mesh.

   As soon as the mesh is ready the Messages window displays the number of mesh elements, which is about 16,000. In addition, two warnings appear in the Messages window. These warnings indicate that the geometry contains edges that are much shorter than the minimum element size, and that there are faces which are smaller than the minimum element size.

   Two warning nodes, one for each type of warning, also appear under the Free Triangular 1 feature node in the mesh sequence. These nodes contain a list of entities, in this case short edges and small faces, causing problems. These entities are also highlighted in the geometry, and are usually surrounded by a denser mesh that indicates faces or edges significantly smaller in comparison to the size of the geometry.

4 Click the Warning 1 node in the meshing sequence, then in the Graphics window zoom in on the area around the bolt holes, shown below.





The areas of dense mesh, indicated by the arrows in the figure, are due to slivers and small faces. Zooming in even closer inside the blue rectangle reveals a small triangular face sitting adjacent to a sliver face. Two of these can be found around each bolt hole location. The edges that are highlighted in blue are listed in the Selection list of the Warning window.

To get a representative size for these faces measure the length of one of its edges, number 646 in the list, which is marked with an arrow in the figure to the right.

5 Scroll down in the Selection list inside the Warning window, then click edge 646.

6 Click Measure in the Evaluate section of the Mesh toolbar.

The Messages window displays the length of the edge, which is 2.556e-4 m — that is, $2.556 \cdot 10^{-4}$ m or about 0.01 inch.

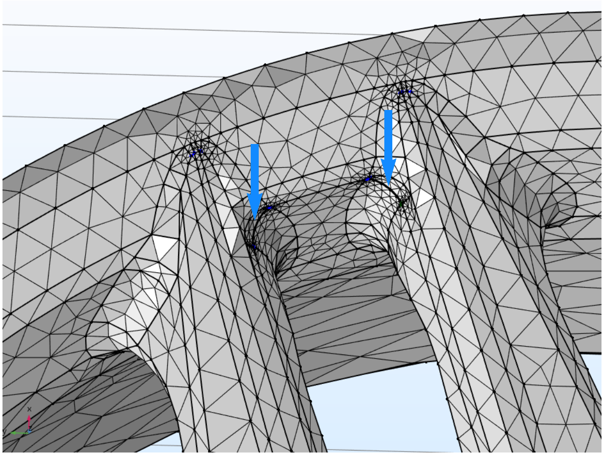
Now take a closer look at some of the other short edges listed in the Warning window.

7 Scroll down to the end of the Selection list inside the Warning window, then click edge 958.

8 Click the Zoom selected button next to the list.

The Graphics window centers and zooms in on the highlighted edge.

9 Using the mouse zoom out and pan to find where the edge is located on the wheel rim. It forms one side of a sliver face located in the region where two adjacent spokes connect to the rim.



Each spoke contains a similar sliver indicated by the arrows in the figure.

10 To get the width of the sliver face, click Measure on the Mesh tab, while the edge is still highlighted in the list.

The length for edge 958 is $3.126 \cdot 10^{-4}$ m (about 0.012 inch).

Repairing the Geometry

Now that you know the size of the faces to be removed you can repair the geometry.

1 On the Geometry toolbar click Defeaturing and Repair and choose Repair.

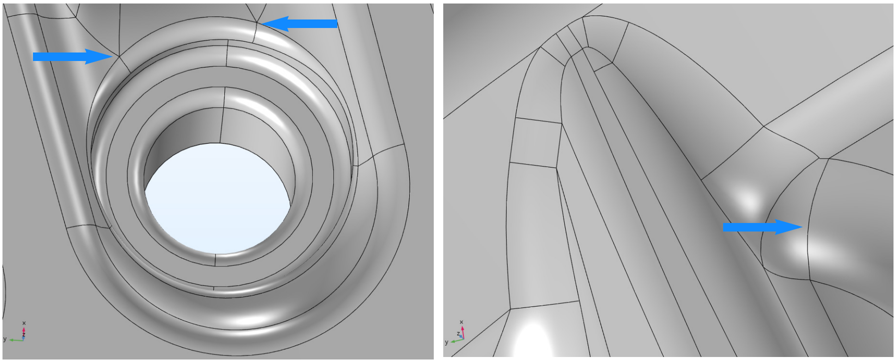
2 In the Graphics window select the wheel rim to add it to the Input objects list.

3 In the Absolute repair tolerance text field enter `3.2e-4`.

By keeping the repair tolerance close to the size of the features to be removed you can avoid removing anything else and breaking the geometry.

4 Click Build All Objects to perform the operation.

5 Examine the geometry. Pan and zoom to take a look at the areas that contained the slivers and small faces, which are now no longer present in the geometry.

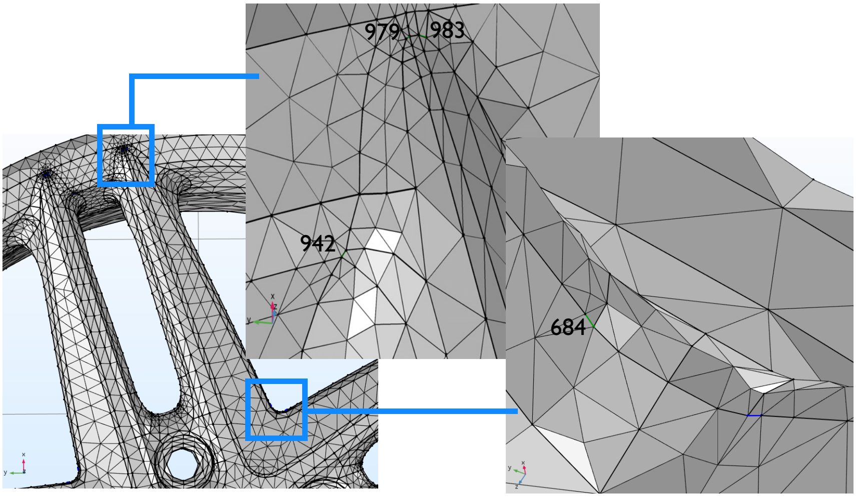


Updating the Mesh and Continuing with the Repair

1 Right-click the Mesh 1 node and select Build All .

This time the mesh contains about 1700 surface elements less than before the repair. The warning node informs that some edges are still shorter than the minimum element size.

2 Click the Warning 1 node, then use the Selection list and the Zoom selected button next to the list to find edges 684, 942, 979, and 983.



Three of these edges are located close to where the spokes connect to the rim, and one is found close to the center of the wheel. Similar edges occur on each spoke.

3 To find an appropriate repair tolerance measure the length of the edges using the Measure toolbar button. The Messages window reports the following

EDGE	LENGTH
684	8.91e-4
942	6.61e-4
979	4.77e-4
983	8.33e-4

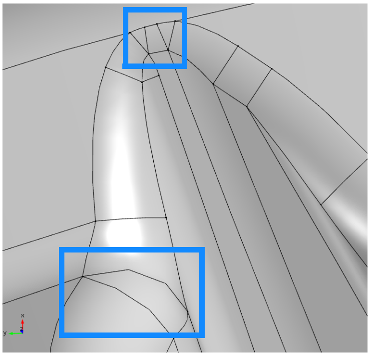
4 On the Geometry toolbar click Defeaturing and Repair and select Repair to continue with the repair of the geometry.

5 Select the wheel rim for the Input objects list.

6 Enter 9e-4 in the Absolute repair tolerance text field.

7 Click the Build All Objects button.

8 After the repair operation completes click the Warning 1 node below the Free Triangular 1 feature, without rebuilding the mesh, and find that no edges remain in the list.



The associativity algorithm in the program ensures that deleted edges are automatically removed from the list in the warning node.

As a result of deleting the edges, the adjacent faces have been modified by the repair algorithm. Most likely this also results in a change of surface curvature in the vicinity of the deleted edges. The longer the deleted edge, the larger the difference from the original geometry that we can expect. For this reason it is recommended to use a small tolerance together with the repair operation. If preserving the surface curvature is important for the analysis, virtual geometry operations, which work by hiding geometry features from the mesher, are available as an alternative. How you can do this is described further ahead.

Generating a Free Tetrahedral Mesh

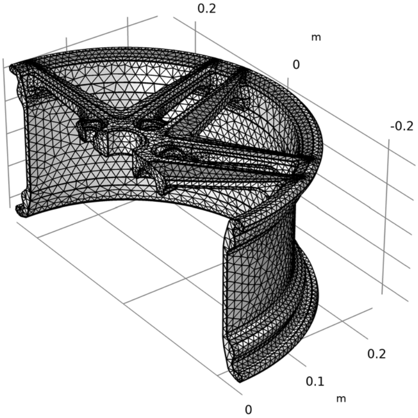
Now that we are done with the defeaturing, let's create a volume mesh for the wheel rim. The fastest way to do this is to reset the meshing sequence.

1 Right-click the Mesh 1 node and choose Reset to the Physics-Induced Sequence.

2 Click Yes in the Confirm Operation dialog box that appears.

   The meshing sequence is reset to contain only a Size and a Free Tetrahedral node.

3 Right-click the Mesh 1 node, then choose Build All.



The mesh builds without warnings this time, and it contains approximately 34000 tetrahedral elements.

Working with Defeaturing Tools

As an alternative to the repair operation described in the previous example you can also apply defeaturing tools to remove small features from the geometry. Using these tools you can first search the geometry for features that fall within a set tolerance, then, after examining the search results, you can decide which ones to delete. While the repair operation has the advantage that it quickly removes every feature it can within a specified tolerance, the defeaturing tools gives you more control with selective removal of features.

To search for and remove small features from a geometry using the defeaturing tools follow this general workflow:

- Import the file
- Search delete small faces
- Search and delete sliver faces
- Search and delete short edges

For the initial search for a feature it is good practice to use a tolerance slightly higher than the default import tolerance, 10^{-5} m. Thus, in a first attempt, search for small faces with a maximum size of 10^{-4} m. Continue by deleting all or some of the returned small faces, then search again with an even higher tolerance, for example $5 \cdot 10^{-4}$ m.

Meshing the geometry can also serve as a diagnostic tool for locating small features, and can be used in combination with the defeaturing tools. After meshing, you can measure some of the small edges and faces reported by the mesher to find a good starting point for a tolerance setting for the defeaturing tools.

The step-by-step instructions below guide you through how to defeature the geometry of the wheel rim that appeared in the previous example.

Model Wizard

1 Start COMSOL Multiphysics.
2 Select Blank Model to skip the steps of selecting physics interfaces and study type.
3 On the Home toolbar, click Add Component and select 3D.

Importing the Geometry

1. On the Home toolbar click Import .
2. In the Settings window for Import click the Browse button.
3. In your COMSOL installation directory navigate to the folder `applications/LiveLink_for_Revit/Tutorial_Examples` and double click the file `repair_demo_1.x_b`.
4. Click Import.

Finding and Deleting Small Faces

1. On the Geometry toolbar click Defeaturing and Repair and select Delete Small Faces .

   In the Tools window for Delete Small Faces, the wheel rim, `imp1`, already appears in the Input objects list.

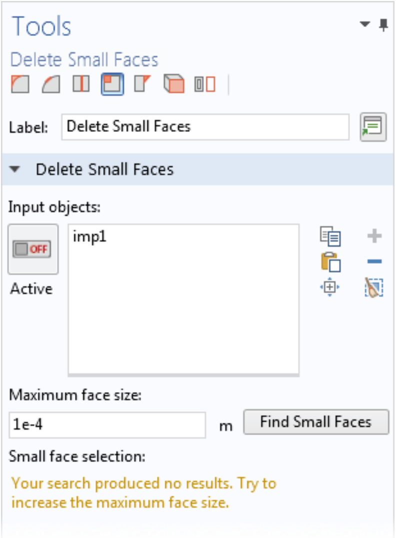
2. In the Maximum face size text field enter `1e-4`.

   Since the default import tolerance is 10^{-5} m it is good practice to start the search with 10^{-4} m, unless the imported CAD design is of a much larger scale.

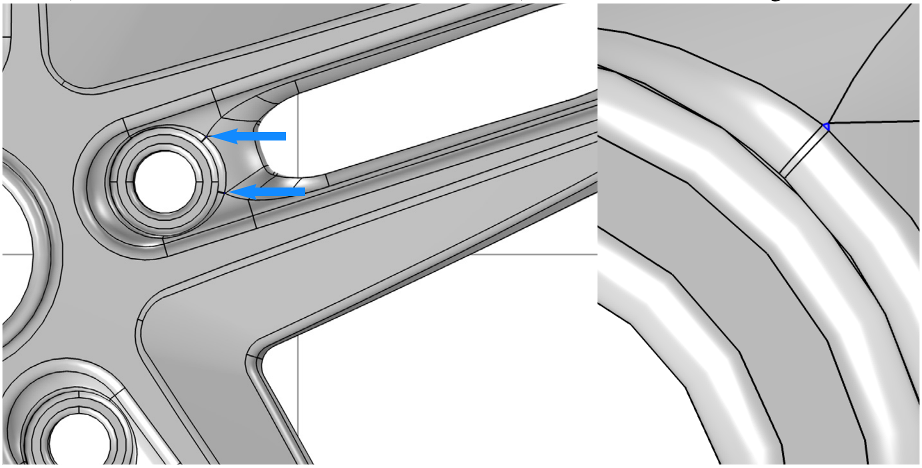
3. Click the Find Small Faces button.
4. Since no faces are found increase the Maximum face size to `4e-4`, then click the Find Small Faces button again.

   This time five faces are listed in the Small face selection list.

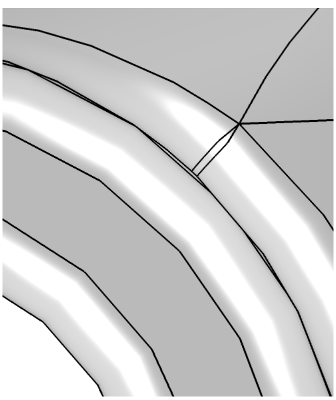
5. Use the Zoom to Selected button next to the list to find the faces on the



rim, which are found around the bolt holes, as illustrated in the figure below.



6 To delete all faces in the list click the Delete All button.



The tool removes small faces by collapsing them into a vertex(point). Therefore it is not recommended to delete larger faces this way as it might result in unexpected changes to the geometry.

Note that as the operation is done the Delete Small Faces 1 (dsf1) node is added to the geometry sequence in the Model Builder tree. The node allows you to go back and edit the delete operation.

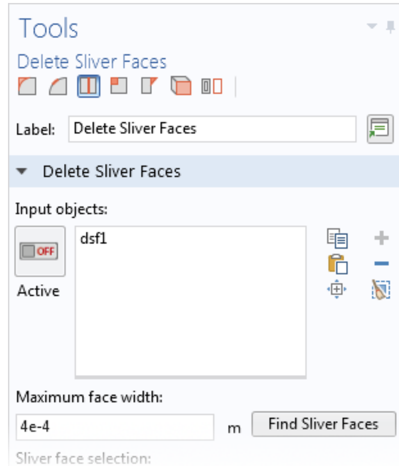
The Tools window for Delete Small Faces continues to be displayed so that you can continue defeaturing using this or any of the other defeaturing tools.

Finding and Removing Sliver Faces

Slivers are faces with high aspect ratio, just like the ones next to those small faces you have just deleted.

1. From the toolbar in the upper left corner of the Tools window for Delete Small Faces click the Delete Sliver Faces button.
2. Enter `4e-4` for the Maximum face width, then click Find Sliver Faces.

   A total of ten faces are found. In addition to the five slivers around the bolt holes, there are five more on the spokes. Use the Zoom to Selected button to find their location on the rim.
3. Click the Delete All button.

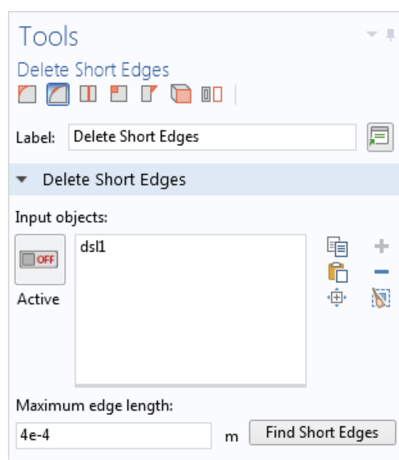


The tool removes sliver faces by collapsing them into an edge, and in this process it uses the tolerance specified for the search. For best results the tolerance needs to be close to the actual width of the face that is deleted. If it happens that a sliver cannot be deleted you can edit the settings for the operation to set a tolerance that is just slightly larger than the width of the face.

Finding and Removing Short Edges

1. From the toolbar in the upper left corner of the Tools window for Delete Sliver Faces click Delete Short Edges.
2. If not already selected, add the wheel rim to the Input objects list.
3. In the Maximum edge length text field enter `4e-4`.
4. Click the Find Short Edges button.

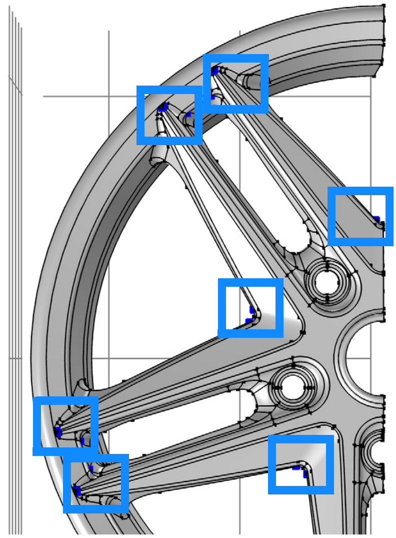
   It seems that the previous operations have removed all edges that were shorter than this value.



5 Increase the Maximum edge length to `9e-4`, then click the Find Short Edges button again.

Take some time to find the edges in the list on the geometry and measure their length. They reoccur in similar places on each spoke. Some of the locations are indicated in the figure to the right.

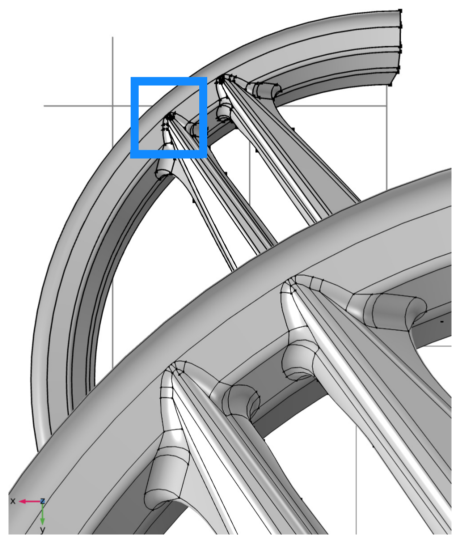
6 Click the Delete All button.



The resulting geometry is similar to the one after the last repair operation in the previous tutorial. The difference is that using the defeaturing tools you can have more control over which entities to delete and in which order.

Just as for geometry repair, it is recommended to use these tools with small tolerances to avoid large modifications to the geometry.

The next tutorial describes how to use virtual geometry operations to avoid small features when meshing, without changing the surface curvature.



Applying Virtual Geometry Operations

The repair and defeaturing tools that find and delete small geometry features can operate only within the limits of what is allowed by the topology of the geometry. To handle more complex cases, where defeaturing fails, you can use virtual geometry operations. With these tools you can set geometric entities, such as vertices, edges, or faces, to be ignored by the mesher. Since selected elements are "hidden" from the mesher, meshing takes place on a virtual geometry, hence the name virtual operations.

Other benefits of using virtual operations is that they work on the finalized geometry, and that they keep the curvature of the geometry. The latter is especially important when removing larger faces, or for certain physics applications when altering the curvature of the geometry can for example give rise to stress concentrations.

Working with virtual geometry operations usually involves the first step of finding small features in the geometry. The general workflow is:

- Import the file
- Find small features by doing one, or both, of the following
  - Search using the defeaturing tools
  - Create a surface mesh or a volume mesh and study the report returned by the mesher
- Use the appropriate virtual geometry tool to hide features.

Using the very same rim geometry as in the two previous examples in this guide, the step-by-step instructions below guide you through how to apply virtual geometry operations on various types of small features.

Model Wizard

1 Start COMSOL Multiphysics.
2 Select Blank Model to skip the steps of selecting physics interfaces and study type.
3 On the Home toolbar, click Add Component and select 3D.

Importing the Geometry

1. On the Home toolbar, click Import.
2. In the Settings window for Import, click Browse.
3. In your COMSOL installation directory navigate to the folder `applications/LiveLink_for_Revit/Tutorial_Examples` and double click the file `repair_demo_1.x_b`.
4. Click Import.

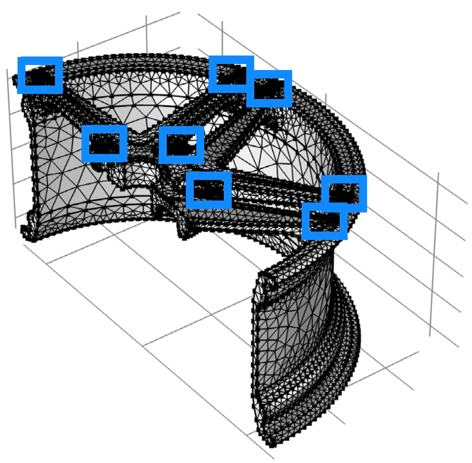
Creating a Surface Mesh

Creating a surface mesh for an imported geometry is often the fastest way to assess the quality of the geometry and to identify regions needing repair or defeaturing.

1. On the Mesh toolbar click Boundary and select Free Triangular.
2. Go to the Settings window for Free Triangular and from the Selection list box select All boundaries.
3. Click the Build All button to create the mesh.

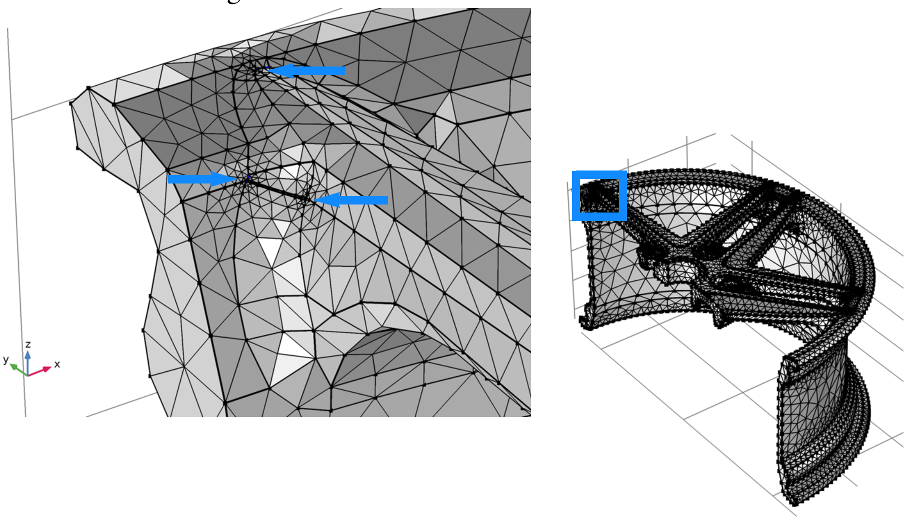
   In the Messages window you can see the number of mesh elements, which is about 16,000. In addition, two warnings are displayed, which indicate that the geometry contains edges that are much shorter than the minimum element size, and that there are faces which are smaller than the minimum element size.

   Next, examine the mesh and look for those areas where the mesher indicates small edges or faces. These regions usually correspond to a denser mesh, some of which are indicated in the figure to the right.



4 Using the Zoom Box button, zoom to the area shown below, where a spoke connects to the rim.

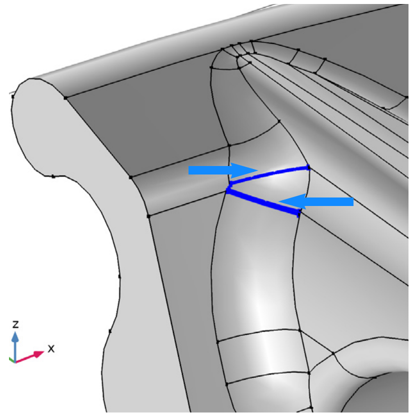
Each spoke contains a dense mesh area due to the small features indicated by the arrows in the figure.



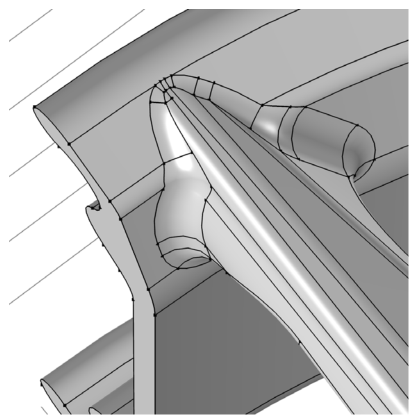
On closer examination you can see that several edges in this area are highlighted as being too short to be meshed with the current mesh settings.

Ignore Edges and Form Composite Faces

1. On the Geometry toolbar click Virtual Operations and choose Ignore Edges.
2. In the graphics area select the edges 217, 219, and 222, highlighted in the figure, to add them to the Edges to ignore list.
3. Click the Build Selected button.



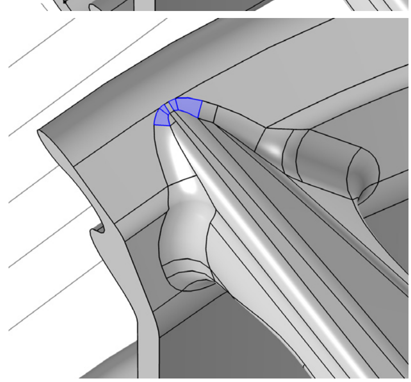
The visualization of the rim in the Graphics window is updated to reflect that the selected edges, and, where applicable, adjacent vertices are no longer part of the geometry which is going to be meshed.



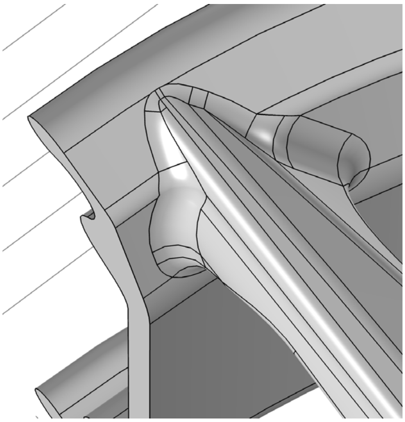
As an alternative to the Ignore Edges operation you can also use the *Form Composite Faces* operation.

4. On the Geometry toolbar click Virtual Operations and choose Form Composite Faces.

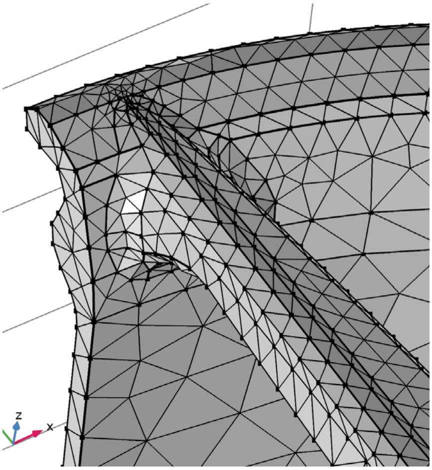
5. Select faces 112, 118, 122, and 126, as highlighted in the figure on the right.



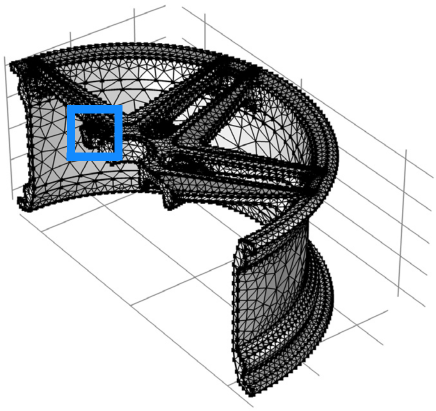
6 Click the Build Selected button. The geometry in the Graphics window is updated with the new composite formed faces.

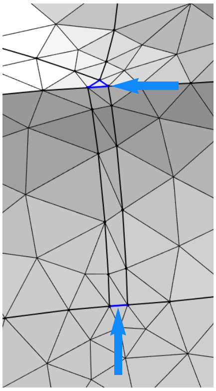


7 To view the new mesh of this region click the Mesh 1 node, then click the Build All button.



1 Click the Zoom extents button to view the entire rim geometry again. Then zoom to the region shown below using the Zoom box button.

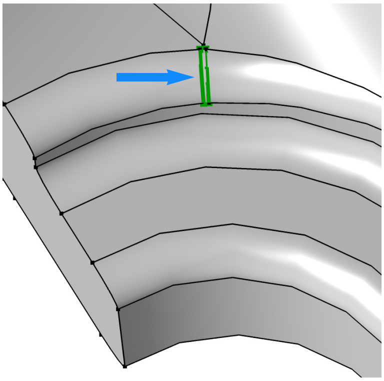




The short edges in this region form a small face which you remove using the *Collapse Edges* operation. The long edges of the sliver face can be removed by adding them to the existing Ignore Edges 1 operation in the geometry sequence.

2 Click the Ignore Edges 1 (ige1) node, then in the Settings window click the Activate button.

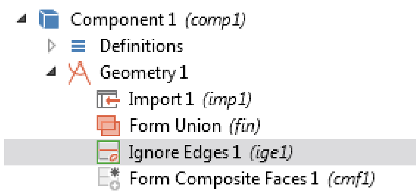
3 Select the edges 197 and 198, shown in the figure to the right. After this latest addition the list should now include edges 197, 198, 217, 219, 222.



4 Click the Build Selected button.

5 Now continue by removing the small triangular face. Before adding the operation

to the sequence take a look in the Model Builder window.

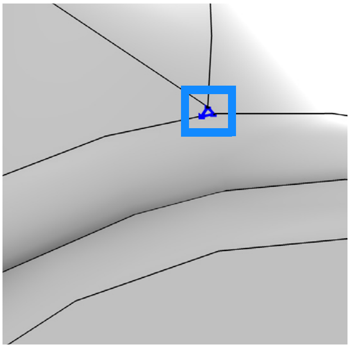


A green rectangle is displayed around the Ignore Edges 1 node telling you that this is the current node. Any operations that you add to the sequence are placed directly after the current node. The Form Composite Faces 1 (cmf1) node is marked, which indicates that the node needs rebuilding.

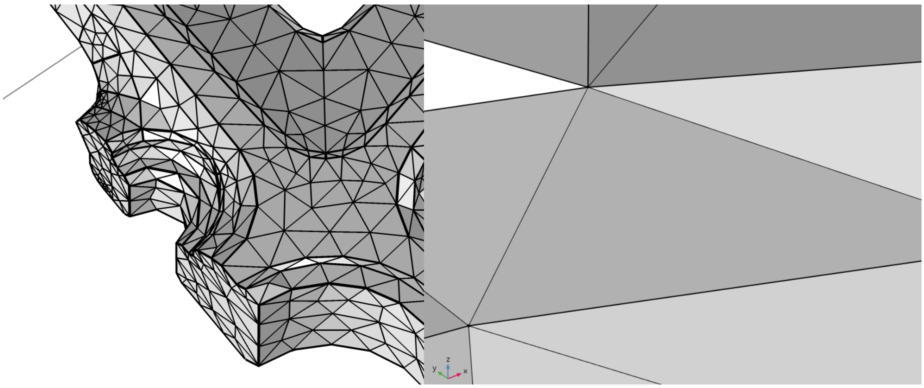
6 Make sure that the next operation is the last one in the sequence by right-clicking the Form Composite Faces 1 (cmf1) node and choosing Build Selected .

Collapse Edges

1 On the Geometry toolbar click Virtual Operations and choose Collapse Edges .
2 Select edges 201-203 highlighted in the figure. Use the Select box button to select all three edges at once.
3 Click the Build Selected button.
4 To build the mesh click first the Mesh 1 node, then the Build All button.



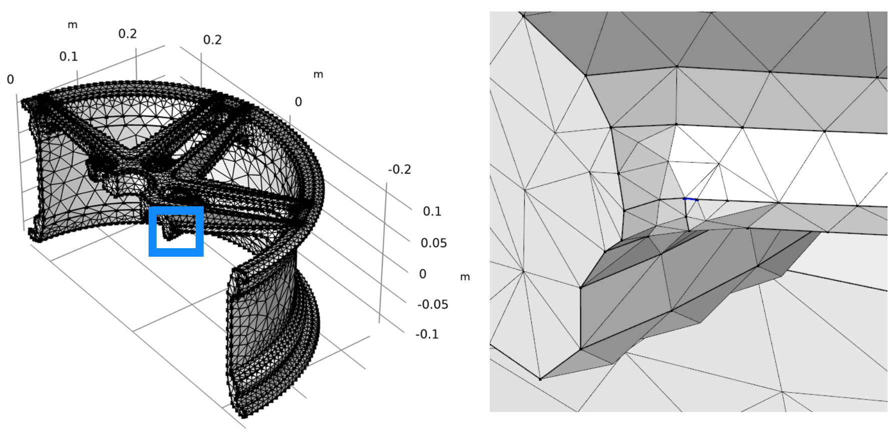
5 The new mesh contains fewer elements since the sliver and small face are no longer visible to the mesher.



Ignore Vertices or Form Composite Edges

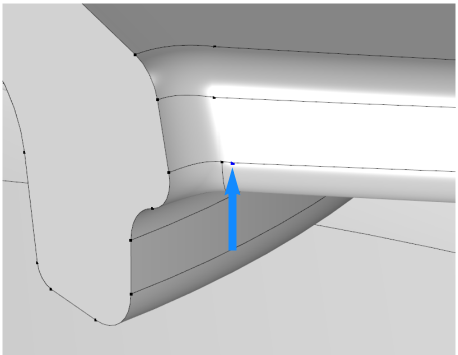
The last virtual geometry operation to try in this example is the *Ignore Vertices* operation to remove a short edge from a segmented edge. In this context the operation is equivalent to the *Form Composite Edges* operation.

1 Click the Zoom extents button to view the entire rim geometry. Then zoom in on the region shown below using the Zoom box button.



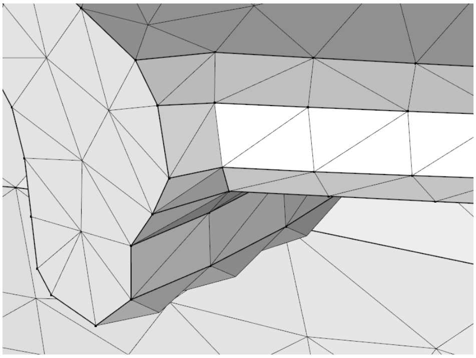
2 On the Geometry toolbar click Virtual Operations and choose Ignore Vertices .

3 Add vertex 108, highlighted in the figure below, to the list of Vertices to ignore, then click the Build Selected button.



4 To mesh the geometry once more right-click the Mesh 1 node and select Build All .

The mesher now sees the two edges as one unit, which is reflected in the way the elements are laid out in this new mesh.



As a last step find a similar short edge in another location on the rim and use the Form Composite Edge operation to hide it from the mesher.

Creating a Fluid Domain Around a Solid Structure

The majority of 3D CAD files include only the geometry of the product to be manufactured. For finite element analysis, however, you find yourself often in a situation where additional geometry is needed, for example, to analyze the flow inside or outside a device. The example in this section, involving the geometry for an exhaust manifold, demonstrates how to create an extra domain for flow analysis. The following steps are covered:

- Importing a Parasolid® file.
- Adding an explicit selection to the geometry sequence.
- Using the cap faces operation to create the additional domain.
- Controlling where an operation is inserted in the geometry sequence.
- Finding and removing fillets from the geometry.

Model Wizard

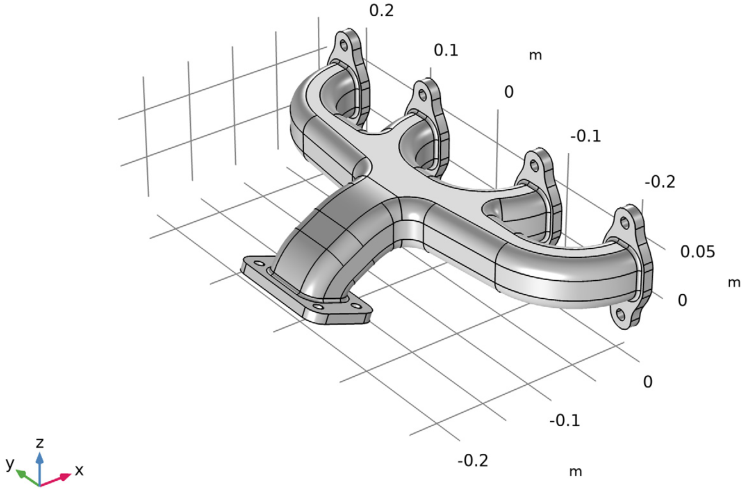
1 Start COMSOL Multiphysics.
2 Select Blank Model to skip the steps of selecting physics interfaces and study type.
3 On the Home toolbar, click Add Component and select 3D.

Importing the Geometry

1 On the Home toolbar, click Import .
2 In the Settings window for Import click the Browse button.
3 In your COMSOL installation directory navigate to the folder `applications/LiveLink_for_Revit/Tutorial_Examples` and double click the file `exhaust_manifold.x_t`.

4 Click Import.

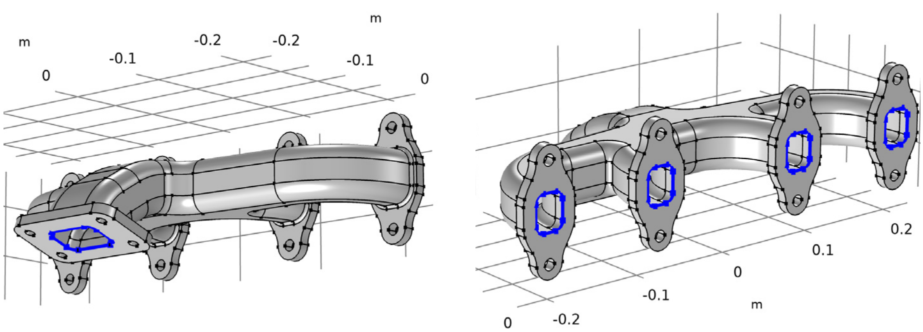
As soon as the import is done the geometry appears in the Graphics window.



Rotate the geometry. As you can see it is hollow inside. As shown below, you can obtain the geometry for the inside in just one operation.

Creating an Explicit Selection

The Cap Faces operation needs an input in form of the bounding edges of the empty volume that should be turned into a solid. For this exhaust manifold these are the edges highlighted in the figure below.



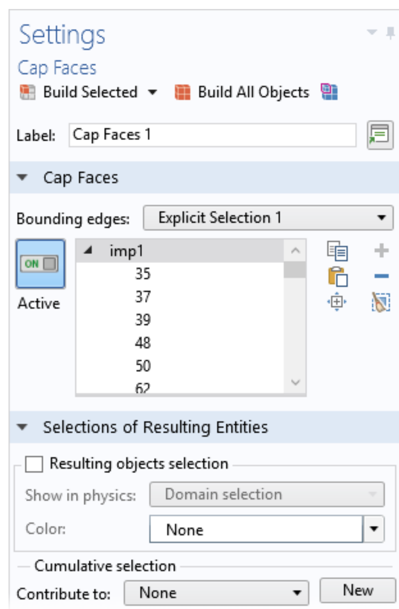
These edges can be selected directly in the Cap Faces operation, however a more efficient way, which only requires the selection of one segment from each edge loop, is to use an explicit selection where you have the option to automatically include continuous edges in the selection.

1 On the Geometry toolbar click Selections and choose Explicit Selection .
2 In the Settings window for Explicit Selection select Edge from the Geometric entity level list.
3 Also select the Group by continuous tangent check box.
4 From the Graphics window select one edge each from the edge loops highlighted in the figure above. Continuous edges are automatically added to the selection. When done check that all edges are highlighted, just as in the figure.

Creating a Domain with Cap Faces

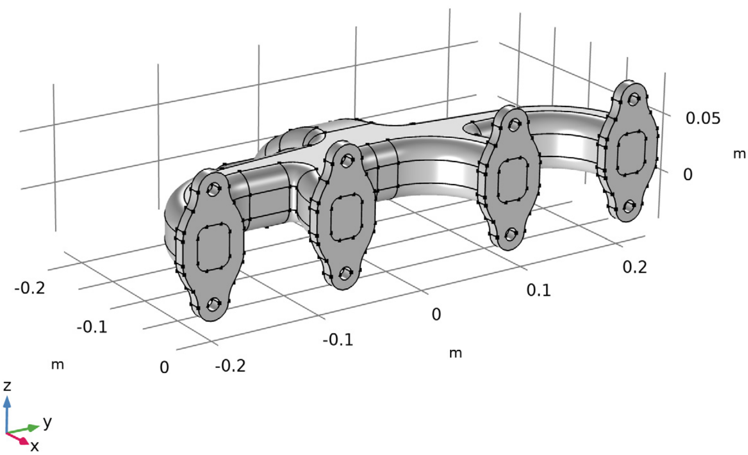
1 On the Geometry toolbar click Defeaturing and Repair and choose Cap Faces .

   The bounding edges of the empty volume inside the manifold are included in Explicit Selection 1 (sel1).
2 In the Settings window, from the Bounding Edges list select Explicit Selection 1.



3 Click the Build Selected button to complete the operation.

The operation closed off the inlets and outlets with new faces. The operation also created a solid domain where it used to be a void inside the exhaust manifold.

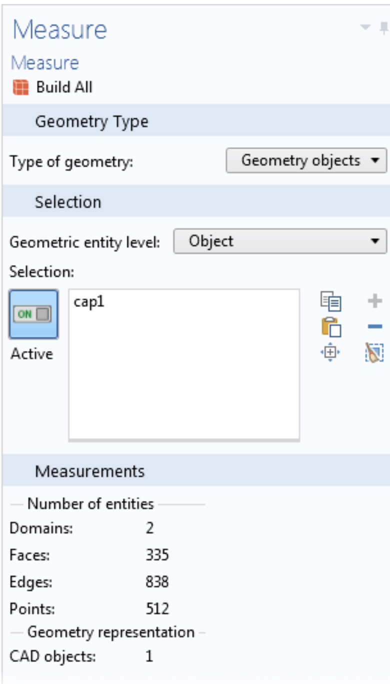


Let's examine this new geometry object using the Measure tool.

4 In the Model Builder tree, right-click Geometry 1 and choose Measure .

5 Select the object in the Graphics window.

According to the information displayed in the Measure window the object `cap1` contains 2 domains.



Removing Fillets from the Geometry

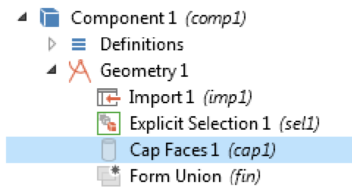
Assume that you are preparing the geometry for a heat transfer analysis for which you have decided to remove some fillets.

Note: A geometry object that contains more than one domain is a non-manifold object which does not support defeaturing operations such as deleting fillets.

In order to remove the fillets you need to insert the Delete Fillets operation before the Cap Faces 1 node in the geometry sequence.

1 Under the Geometry 1 node, right-click Cap Faces 1 (cap1) and choose Build Preceding .

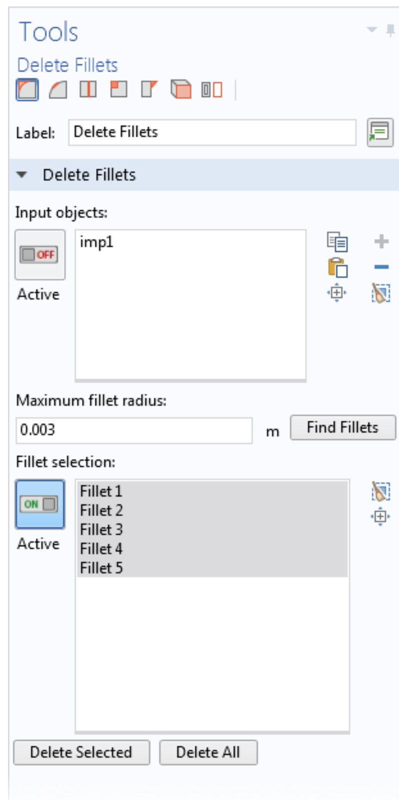
The Cap Faces 1 (cap1) node becomes unavailable, and the Explicit Selection 1 (sel1) node becomes the current node, which is indicated by a green rectangle around its icon. You can now apply the defeaturing operation, as it is inserted before the Cap Faces 1 (cap1) node.



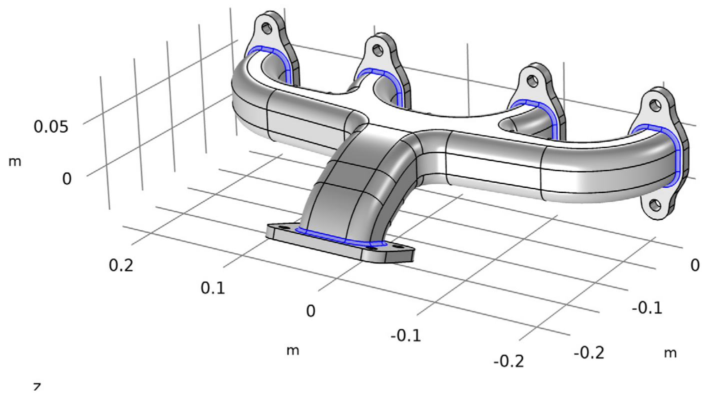
2 On the Geometry toolbar click Defeaturing and Repair and choose Delete Fillets .

3 In the Tools window `imp1` already appears in the Input objects list, and you can enter `0.003` in the Maximum fillet radius text field.

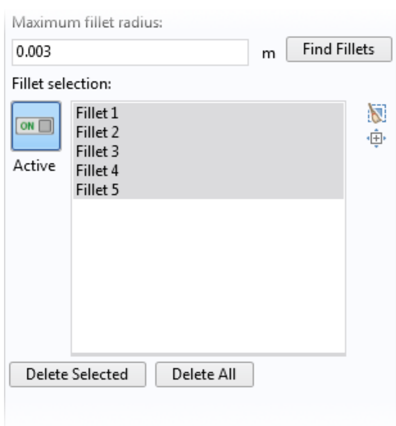
4 Click the Find Fillets button to search for fillets with a radius less than 0.003 m in the geometry.



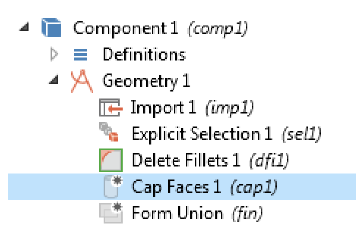
5 Five fillets are found by the tool. These appear in the Fillet selection list, and they are also highlighted on the geometry.



6 Remove all fillets by clicking Delete All in the Tools Setting window.



As the operation completes and the fillets are removed note that the Delete Fillets 1 (dfi1) node is inserted into the geometry sequence, just above the Cap Faces 1 (cap1) node. The Cap Faces 1 (cap1) node is still unavailable, meaning that it is currently not built.



7 To re-build the entire geometry sequence click the Build All button on the Geometry toolbar.

The geometry is now ready for meshing and analysis!