



INTRODUCTION TO Design Module

Introduction to the Design Module

© 2005–2025 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement. Portions of this software are owned by Siemens Industry Software, Inc. © 1986–2025. All Rights Reserved. Portions of this software are owned by Spatial Corp. © 1989–2025. All Rights Reserved.

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

Version: COMSOL 6.4

Contact Information

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

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

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: www.comsol.com/support/learning-center

Part number: CM024002

Contents

Introduction	5
Creating Sketches with Constraints and Dimensions	6
Working with the Loft Operation	21
About CAD File Formats	28
The Geometry Cleanup Wizard	30
Removing Small Geometric Entities with Repair	40
Removing Small Geometric Entities with the Defeaturing Tools	44
Creating Modeling Domains Around an Imported Geometry.	53

Introduction

This guide introduces you to the Design Module that provides tools such as constraints and dimensions, and projections to create 2D geometry, and the geometry features loft, thicken, midsurface, fillet, chamfer, offset faces, project to faces, and transform faces for creating and modifying geometry in 3D.

Using the loft operation you can generate 3D geometry based on cross-sectional profiles, while the midsurface enables the simplification of imported geometry objects for shell type analyses. Using a combination of midsurface and thicken operations you can even reparameterize and optimize the thickness of imported geometry. The offset faces and transform faces operations enable the editing of imported 3D geometry objects to change the size and position of geometric features. You can, for example, reparameterize the diameter and location of holes to optimize an imported design.

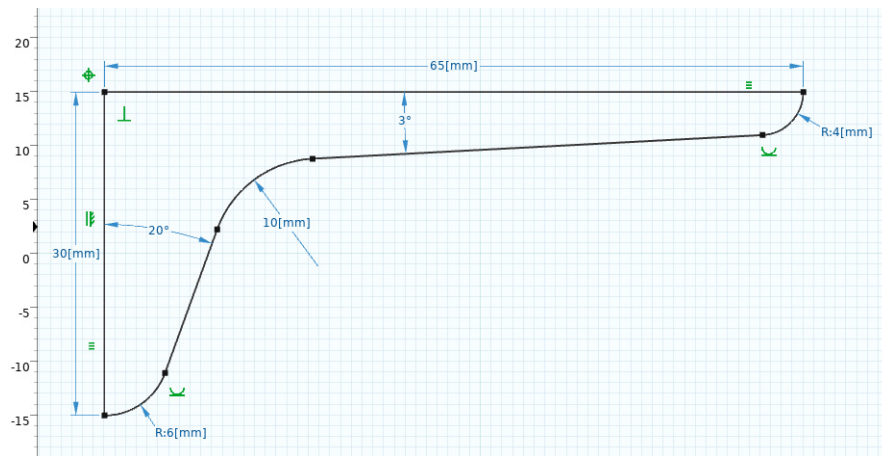
The module adds support for importing several 3D CAD file formats, and includes repair and defeaturing functionality for preparing imported geometry for analysis. The detailed tutorials that follow start you off with becoming efficient in using the provided functionality.

Creating Sketches with Constraints and Dimensions

The constraint and dimension features enable an efficient workflow to create 2D geometries, including on work planes in 3D. By using the drawing tools available in sketch mode you can draw an initial sketch; then, by adding constraints and dimensions, you can obtain the final geometry.

A constraint is a requirement on geometric entities that does not have a value; for example, a requirement that two edges should be parallel. A dimension is a requirement that has a value; for example, a requirement that specifies the distance between two vertices. The constraints and dimensions added to the geometry are visualized with symbols and arrows in the Graphics window.

Follow this tutorial to learn how to use the drawing tools in sketch mode and to apply constraints and dimensions to create the geometry of the bracket pictured below.



Creating a New Model

- 1 Start COMSOL Multiphysics; then, under New, select Model Wizard.
- 2 Select 2D, then click Done to skip the steps of selecting physics interfaces and study types.
- 3 Under Component 1, select Geometry 1.

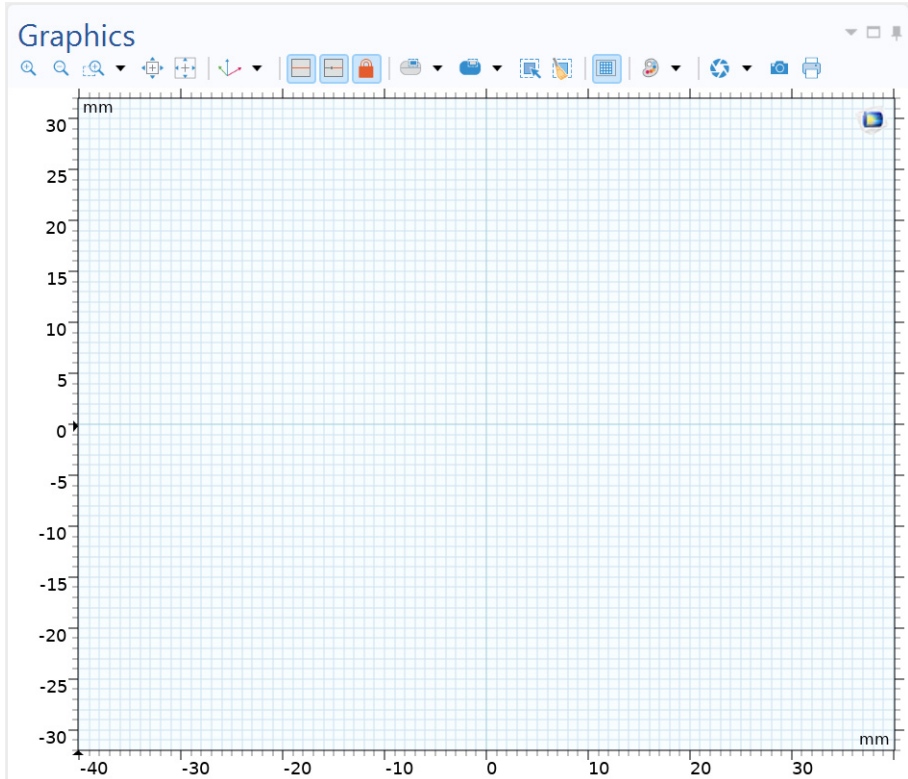
4 In the Settings window, under Units, change Length unit to mm.

The bracket is larger than the default zoom level in the Graphics window.

Adding constraints and dimensions is easier if the original sketch is drawn closer to its final size; therefore you can start by zooming out a few steps.

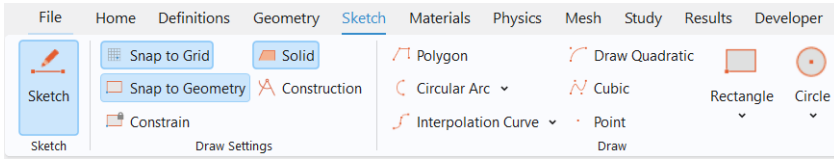
5 In the Graphics toolbar, click Zoom Out () five times.


Your Graphics window should look similar to the one below.



Enabling Sketch Mode and Constraints and Dimensions

1 From the ribbon, select the Sketch tab.

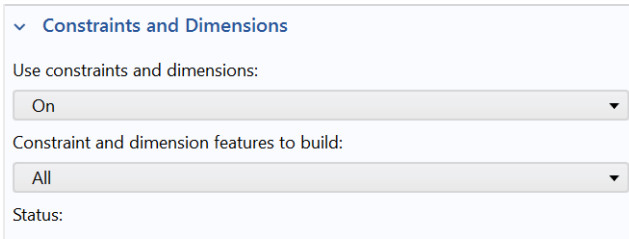


2 Sketch mode should be enabled by default, if not select the Sketch () button.

In sketch mode, you can draw the geometry directly in the Graphics window using the interactive drawing tools from the Sketch tab. Snapping to the grid and to the geometry while drawing are enabled by default. The Solid button determines the default output type of the shapes you draw. When the button is selected, solid objects are created; otherwise, curve objects are created. You can also change the object type in the settings for the created features. In sketch mode only the edges and the vertices of the geometry are visualized, the domains become visible after you exit sketch mode.

The constraint and dimension functionality will be activated automatically as soon as you add a constraint or dimension feature to your drawing. Here, enable it manually in order to review related settings.

3 From the Model Builder select Geometry 1. In the settings for Geometry, under Use constraints and dimensions, select On.



The setting Constraint and dimension features to build is initially set to All, meaning that such features in the geometry sequence are always active regardless of the currently built node. You can change this if you only want the constraints and dimensions up to a specific build target in the sequence to be active. This may be useful to debug a geometry sequence for example.


When building a geometry there may be several solutions for the geometry shape given the applied constraints and dimensions. It is only when the applied constraints and dimensions uniquely define the drawing that predictable changes can be obtained by modifying dimension values. Thus, as you work on the drawing and apply new constraint or dimension features to the geometry,

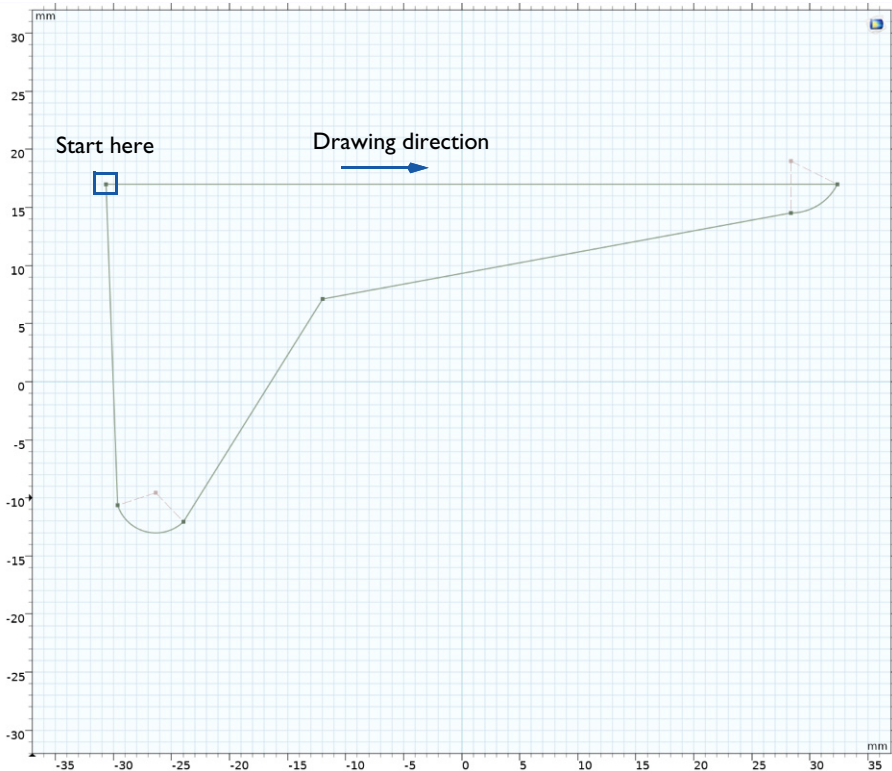
the returned shape may not always correspond to the desired one. To avoid this effect, it is recommended to add constraints before adding dimensions.


Constraint features define the relations between the geometric entities, thereby reducing the number of possible solutions when you start adding dimensions.

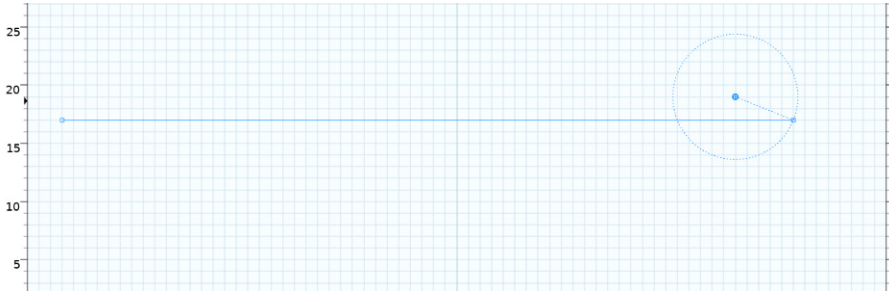
If you are unhappy with the provided solution after adding a constraint or dimension, use undo; then use the mouse to drag the edges and vertices closer to the desired shape. It can also help if the initial sketch is not too different in size from the final object.

Drawing the Bracket

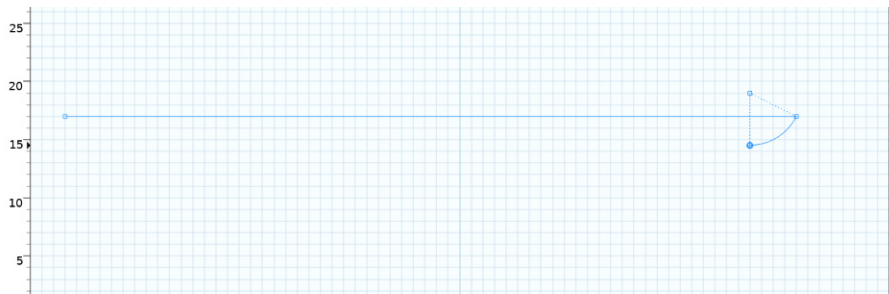
- 1 From the Sketch tab, select Polygon ()
- 2 In the Graphics window click to start drawing the bracket. Start at the corner highlighted below, and draw the first segment in the direction of the arrow.




- 3 Move the pointer to the right, and at the end of the first edge where the circular arc starts click once to place a vertex.
- 4 To switch to drawing a circular arc, right-click in the Graphics window, then from the context menu choose Circular Arc (), then choose Start, Center, Angle.
- 5 As you move back the mouse pointer to the Graphics window, the preview of the drawing indicates the center of the arc. Click to place it according to the figure below.



- 6 Move the mouse to form the arc, then click to place a vertex where the arc ends.



- 7 Right-click, then from the context menu choose Polygon () to continue with the next straight edge.
- 8 Continue to draw the bracket. Click to place the end vertices of the segments, and switch to the Circular Arc tool when needed for the second arc. Try to create a sketch that resembles the drawing under Step 2 above.
- 9 To close the shape, position the last vertex on top of the first vertex, then click to place it, the shape will be closed automatically.

When finished, the Composite Curve 1 node is added to the geometry sequence. This node contains the two polygon and circular arc features that you have drawn. Collecting several features into a composite curve ensures that the end points are always coincident, so that no additional constraints are needed.

- Under Geometry 1, expand the Composite Curve 1 node, then click Polygon 1. The settings for Polygon contains a table with the point coordinates you have just drawn.

In sketch mode you can modify the polygon by dragging the edges and vertices in the Graphics window.

x (m)	y (m)	Constrain
-29.63658834410586	-10.63041000962971	
-30.63448337402344	17	
32.365516662597656	17	

- Drag an edge or vertex on the canvas. Before constraints and dimensions are applied to the sketch it is possible to move the entities without any restrictions. To restore the geometry click the Undo () button, or press Ctrl+Z.

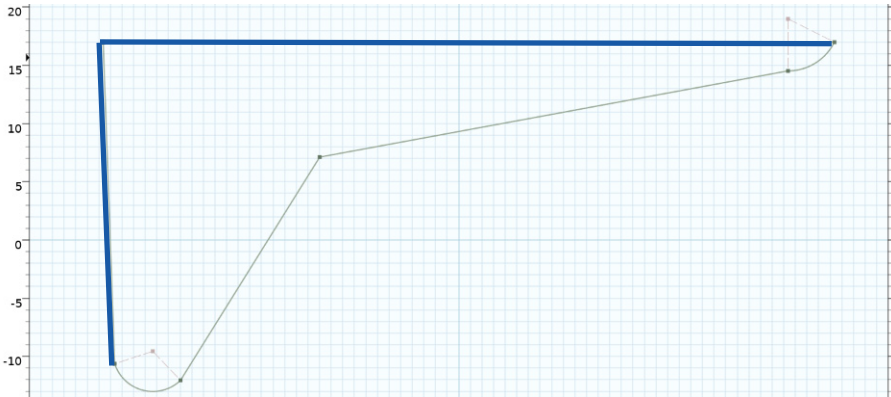
Another way to modify the polygon is by editing the values in the coordinates table. The Constrain () button next to the coordinates locks the vertex so that it is not possible to drag it in the Graphics window. The coordinates that are constrained this way become *built-in dimensions* and are not possible to modify by constraint and dimension features. For this example, leave all coordinates unconstrained as you will continue with applying constraints and dimensions to obtain the final shape of the bracket.

Adding Constraints to the Sketch

- From the Sketch toolbar, select Constraint ()

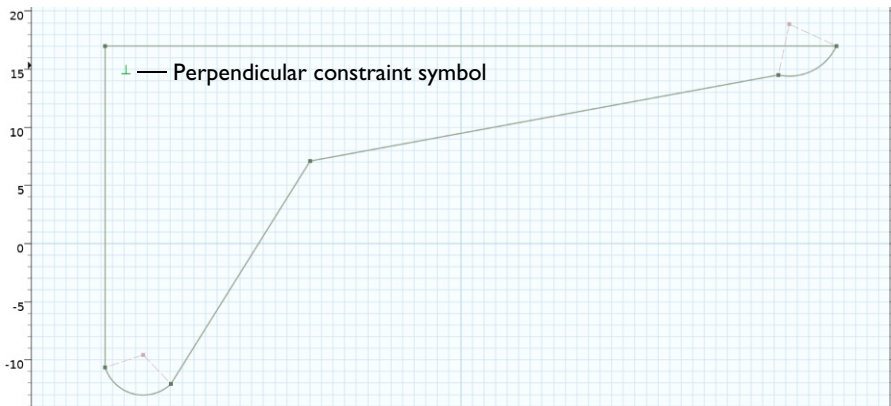
This activates the smart constraint mode where you can constrain geometric entities by selecting them in the Graphics window. Depending on the geometry and selected entities, a constraint is automatically suggested by the software. You may choose to apply the suggested constraint or change it to a different constraint.

2 From the Graphics window, select the two edges highlighted below.



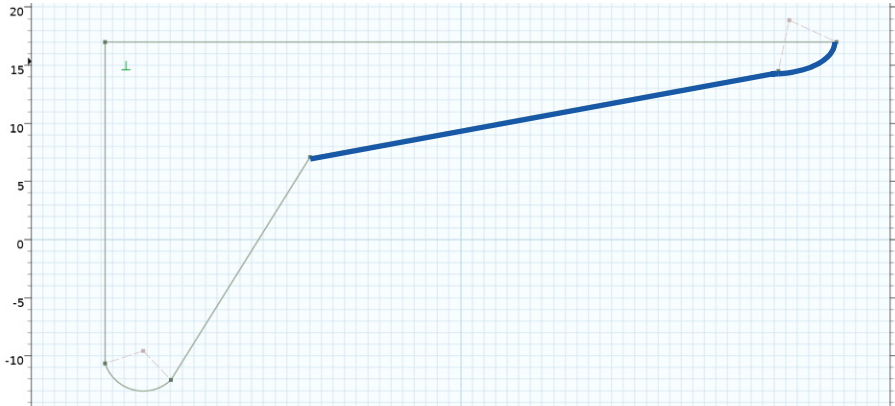
The perpendicular constraint is suggested, and its symbol appears under the pointer.

3 Click in the Graphics window, close to the selected edges, to place the symbol for the perpendicular constraint. You can always drag the symbol to adjust its position after placing it. The drawing is updated right away.



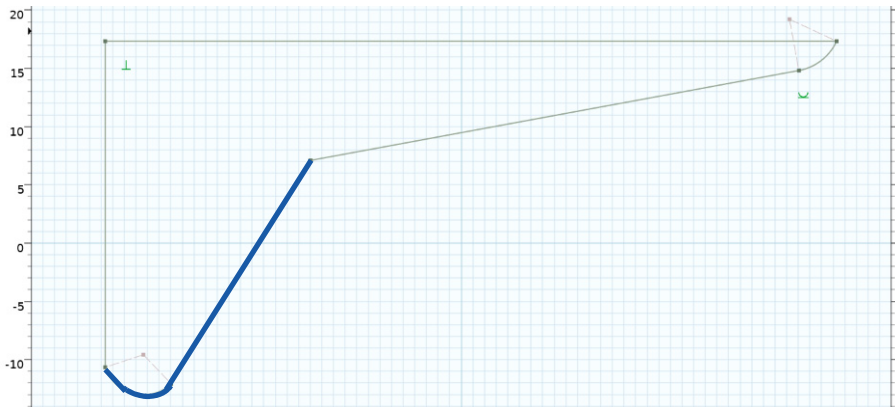
Note that in the Model Builder, the Perpendicular 1 node has been added under the geometry sequence. Similar to all other geometry features, you can select the node to change its settings.

4 Select the circular arc and straight edge highlighted below.



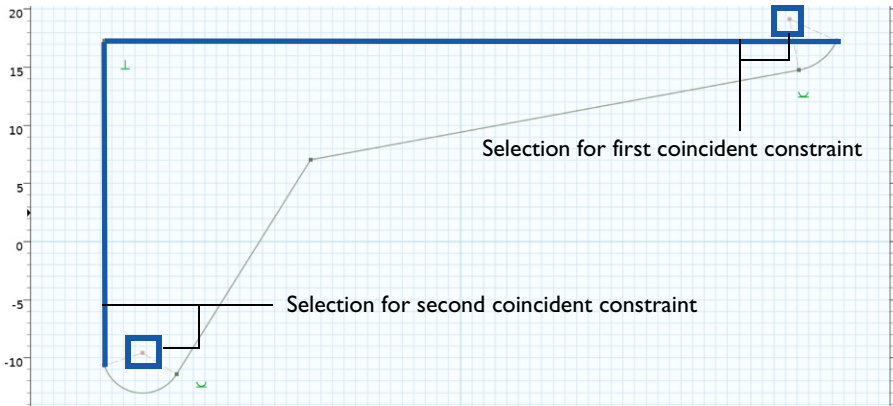
5 Click on the canvas close to where the edges meet to apply the suggested tangent constraint.

6 Continue with selecting the next straight and circular edge segments, then apply the suggested tangent constraint.

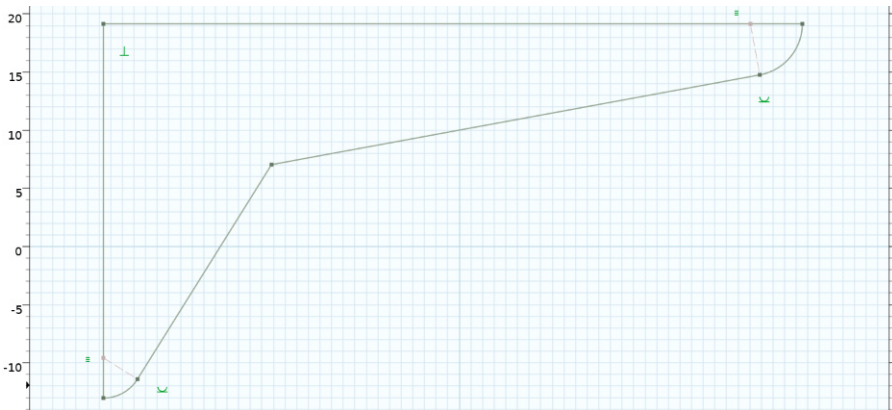


7 Next add constraints to position the centers of the circular arcs to lie on the two straight edges. First select the horizontal edge and the center of the circular arc

to the right, apply the suggested coincident constraint, then repeat for the remaining arc and the vertical edge.



The geometry should now look similar to the one displayed below.



You have now applied the constraints needed to define the relations between the entities in the drawing.

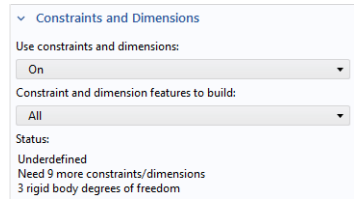
- 8 Right-click in the Graphics window, and from the context menu choose Constraint to exit the smart constraint mode.
- 9 Test again to drag an edge or a vertex of the geometry. Note how the relations prescribed by the constraints are maintained as you do this.

The goal when applying constraints and dimensions is to obtain a fully constrained sketch that is not possible to modify by dragging in the Graphics window. Such a geometry is called *well defined*. The bracket geometry is currently still *underdefined*. It is not recommended to leave a sketch in an


underdefined state as several solutions exists, and modifying a dimension, for example by a parametric sweep, may lead to unpredictable results.

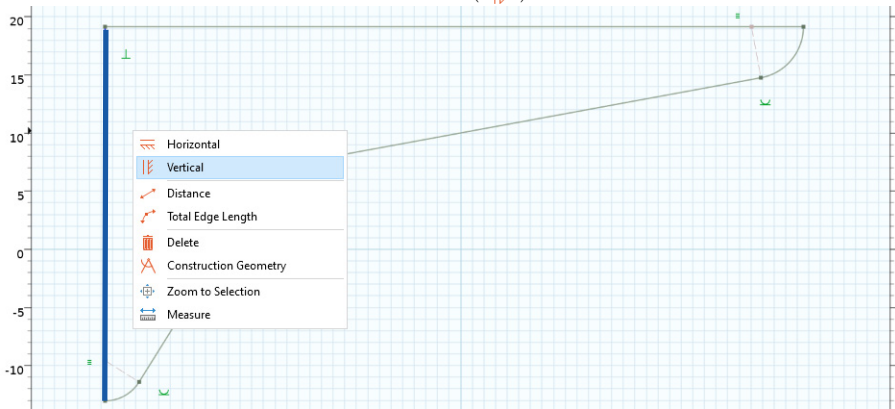
10 Press Ctrl+Z as needed to undo any changes to the sketch.

11 To check how many more constraints or dimensions are needed until the geometry becomes well defined, click the Geometry 1 node in the Model Builder.

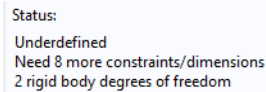


As you can see from the information in the Settings window the geometry is underdefined, and it needs 9 more constraints or dimensions. The rigid body translational and rotational degrees of freedom are not yet locked, meaning that the entire sketch can be moved around freely as a rigid body. To lock the rotational degree of freedom you can for example apply the *Vertical* or *Horizontal* constraints to an edge.

12 In the Graphics window select the edge highlighted below, then right-click and from the context menu choose Vertical ().




13 To check the status of the geometry in the Model Builder click the Geometry 1 node.



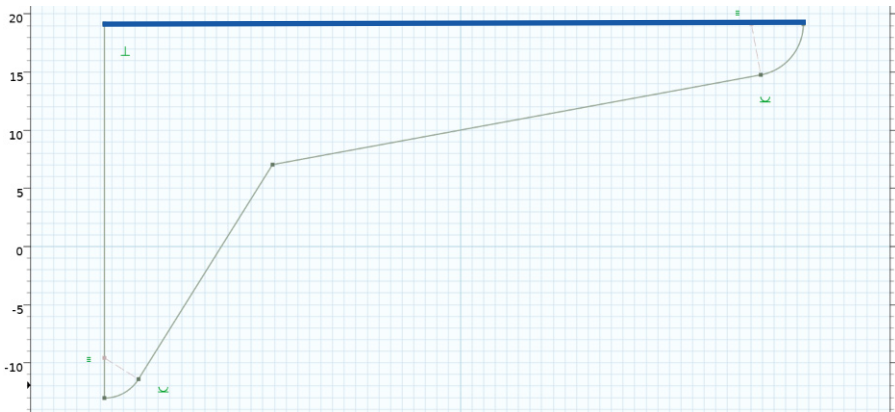
Only 8 more constraints or dimensions need to be applied until the geometry becomes well defined.

Adding Dimensions to the Sketch

- 1 Right-click in the Graphics window, and from the context menu choose Dimension () to enter the smart dimension mode.


The smart dimension tool works similarly to the smart constraint tool: you start by selecting entities, then either apply the suggested dimension, or change it first, and finally enter a dimension value before continuing with adding more dimensions.

- 2 From the Graphics window, select the edge highlighted below.

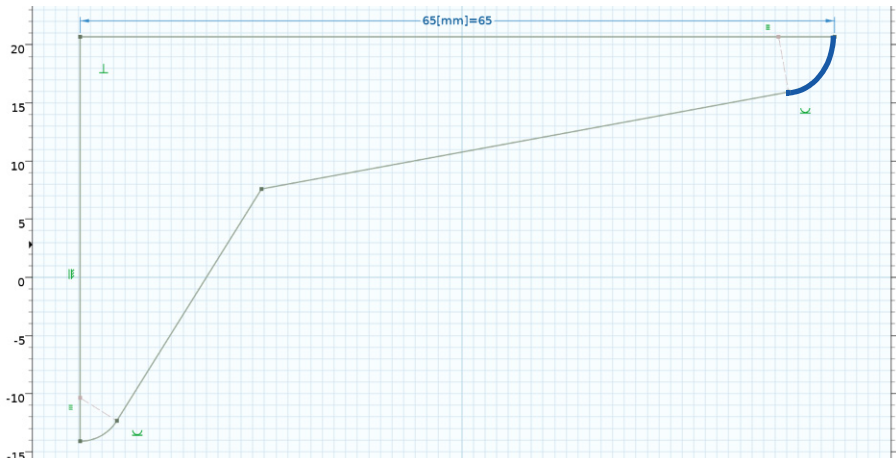


- 3 To apply the suggested Distance dimension, click above the edge to place its symbol.

The Distance 1 node is added to the geometry sequence, and the Settings window for Distance is displayed where you can start typing directly to edit the value.

- 4 In the Settings for Distance, in the Distance field enter 65[mm].
- 5 Click Build Selected (), or press F7, to build the geometry using the new value.

6 Select the circular arc highlighted below.

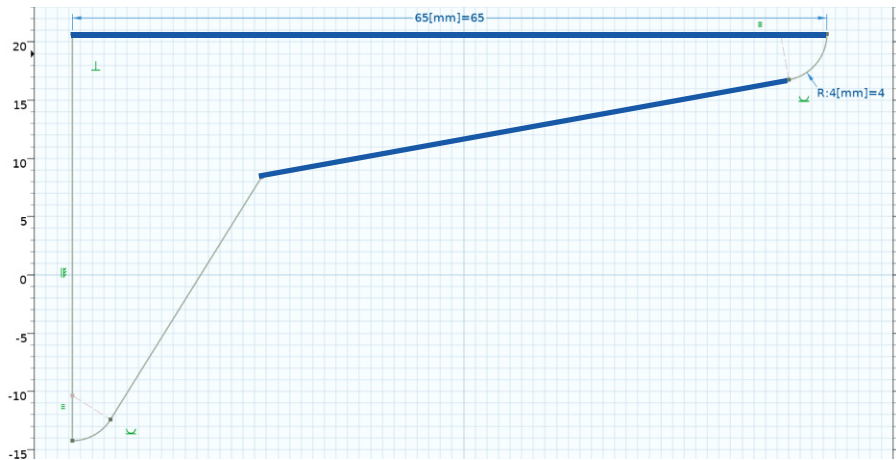


7 Click to apply the suggested Radius constraint.

8 In the Settings for Radius, in the Radius field enter 4 [mm].

9 Press F7 to build the geometry.

10 Select the two edges highlighted below.



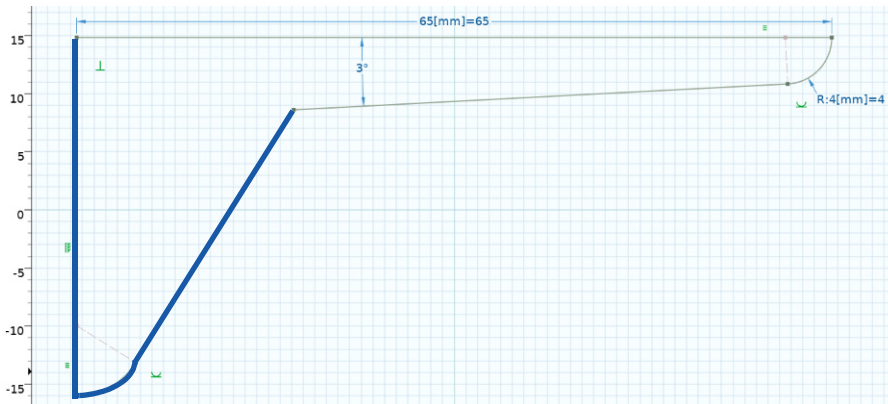
11 Click somewhere between the edges to apply the suggested Angle dimension.

12 In the Settings for Angle, in the Angle field enter 3 [deg].

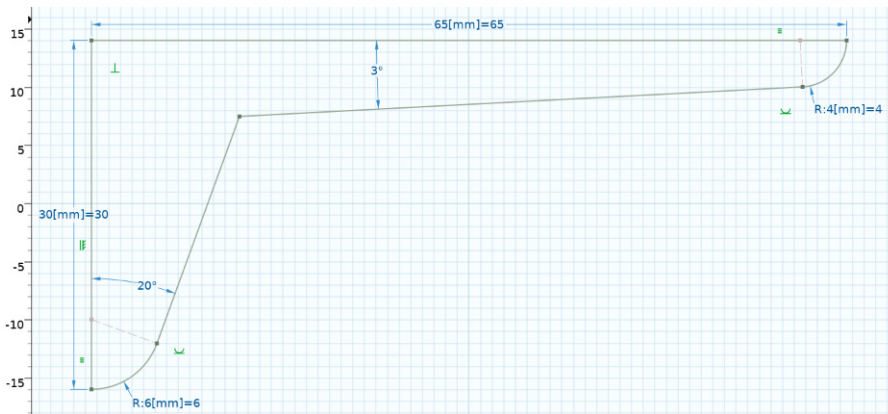
13 Press F7 to build the geometry.

14 Repeat Steps 2–13 to add a distance, a radius, and an angle dimension for the edges on the left side of the bracket, highlighted in the following figure. Use

the following dimension values: 30[mm] for the Distance dimension applied to the vertical edge, 6[mm] for the arc radius, and 20[deg] for the angle.



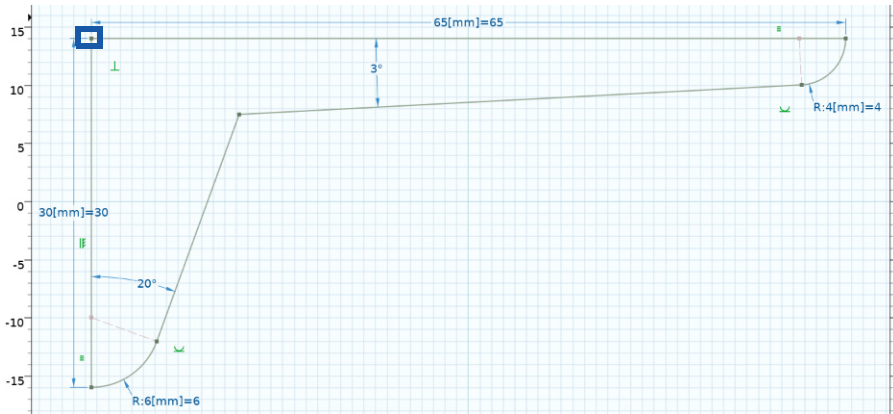
When done your drawing should look similar to the one displayed below.



The applied constraints and dimensions are now sufficient to define the shape, but the two translational rigid body degrees of freedom are not yet constrained.

- 15 Test to move around the geometry by dragging an edge or vertex on the canvas. Note that it is not possible to rotate it due to the applied Vertical constraint. Restore the geometry if needed by pressing Ctrl+Z.

- 16 As the smart dimension mode is still active you can select the vertex below for applying the last dimension that will constrain the remaining degrees of freedom.

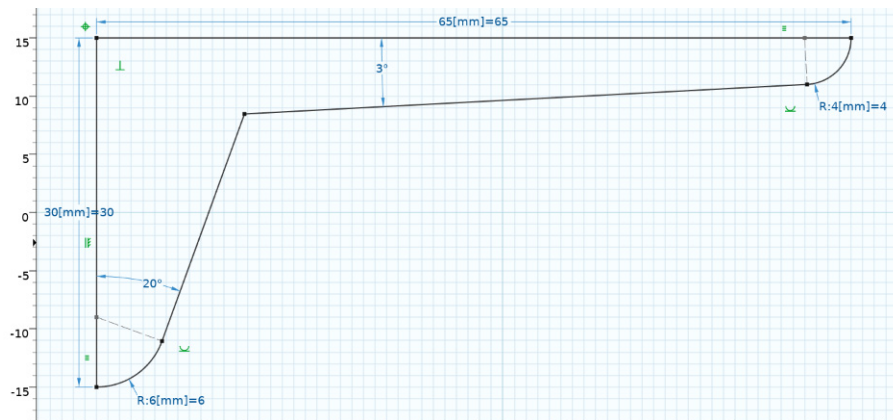


- 17 Click close to the vertex to place the symbol for the suggested Position dimension.

- 18 In the settings for Position, x and y coordinates text fields enter -35 [mm] and 15 [mm], respectively.


- 19 Press F7 to build the geometry.

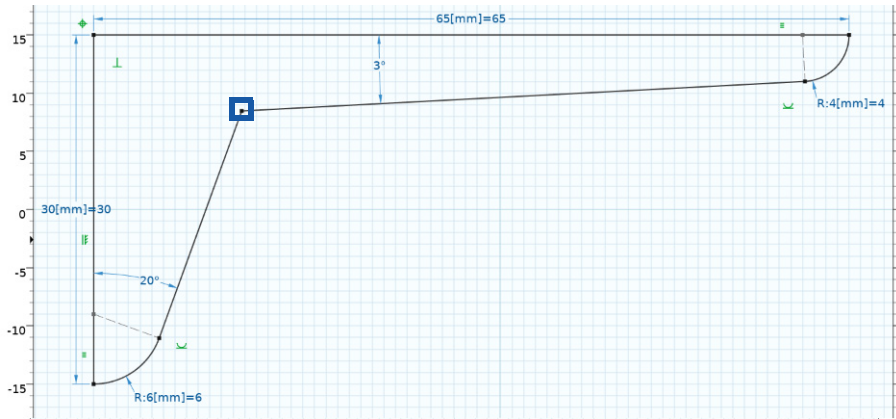
The geometry is now fully defined. This is also indicated by the black color of the entities.



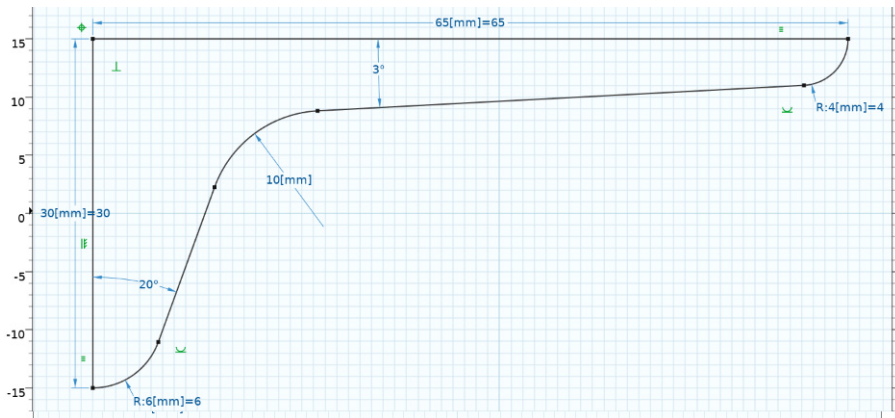
- 20 Right-click in the Graphics window, and from the context menu choose Dimension to exit the smart dimension tool.


Applying a Sketch Fillet

- 1 From the Sketch tab select Fillet ().
- 2 Select and drag the vertex highlighted below to add a fillet. A preview of the fillet is displayed while dragging, and when you let go of the vertex the Fillet feature is added to the geometry sequence.



- 3 In the settings for Fillet, in the Radius text field enter 10[mm].
- 4 Press F7 to build the geometry. The bracket drawing is now complete.



- 5 Click the Sketch () button to exit sketch mode (this also exits the sketch fillet mode). The solid object of the bracket is now displayed in the Graphics window.

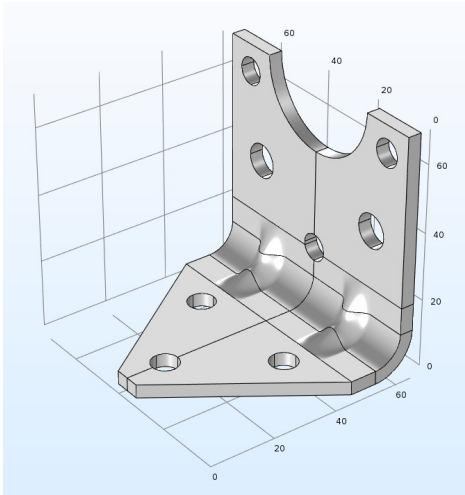
Working with the Loft Operation

The loft operation enables creating 3D surfaces based on cross-sectional profiles. The following tutorial demonstrates how to generate lofted surfaces with tangents matching the tangents of adjacent faces by specifying the loft direction and using guide curves.

The Geometry Sequence

- 1 Start COMSOL Multiphysics, then from the File menu select Open.
- 2 Open the file `bracket_geometry.mph`, which is found in your COMSOL installation directory, under the folder `applications/Design_Module/Tutorial_Examples`.

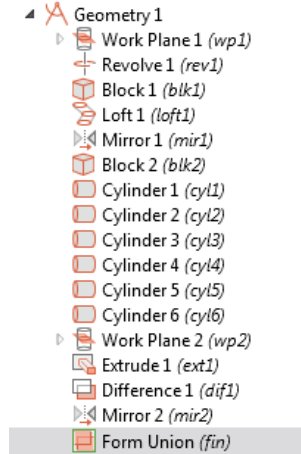
The file contains a sequence of geometry operations that results in the geometry of the bracket displayed below.



3 In the Model Builder, under the node Component 1 > Geometry 1 you can see the sequence of geometry operations that builds the bracket.

The sequence starts by creating the elbow, which is the output of the Revolve 1 operation. The output of the Loft 1 operation and its mirror (Mirror 1) connect the elbow to the two wings, which are the Block 1 and Block 2 operations. The holes in the bracket are generated by subtracting the cylinders from the geometry. Finally the Mirror 2 operation generates the full bracket geometry.

4 To study how the geometry is built, step through the sequence of operations by right-clicking on a node then selecting Build Selected.




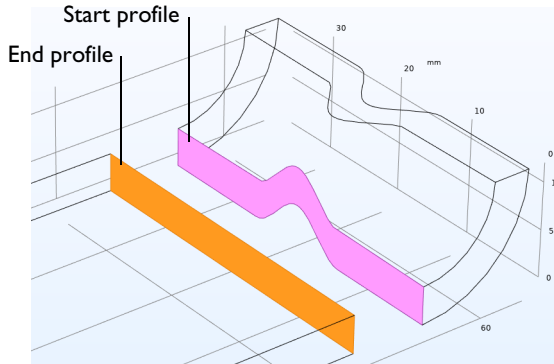
The Loft Operation

1 Right-click the Loft 1 node , then select Build Preceding .


2 In the Settings window for Loft, expand the Start Profile and End Profile sections.

The loft operation generates a solid object to connect the rectangular cross-section of the bracket with the cross-section of the elbow.

- 3 To get a better view of the start and end profiles click the Wireframe Rendering  button in the Graphics toolbar and use the mouse to zoom in. The faces selected as the start and end profiles are highlighted in the image below.

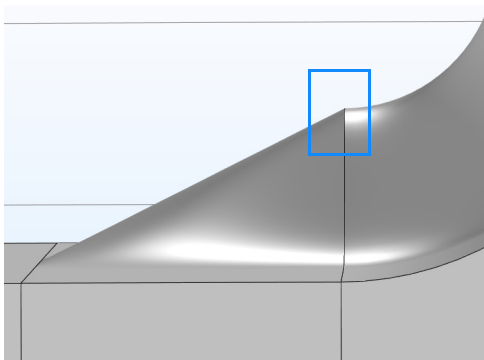


While in this case the input consists of a start and an end profile, it is possible to define additional profiles in the Profiles section.

- 4 In the Loft toolbar click the Build Selected  button.

LOFT DIRECTION

- 1 Click the Wireframe Rendering  button in the Graphics toolbar.




- 2 In the Graphics window move and rotate the object to get a view perpendicular to the zx -plane. Then zoom in to get a view similar to that in the figure.

In case on your screen the lofted object does not appear to match the start profile you can adjust the visualization settings for the Graphics window.

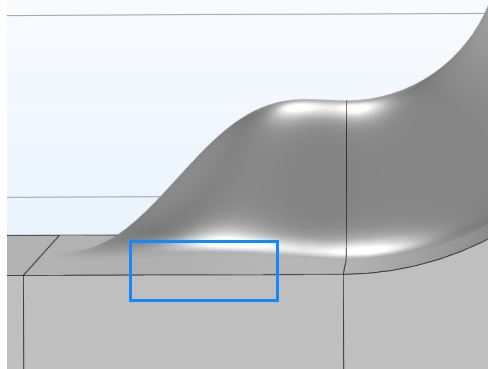
- 3 From the File menu select Preferences; then on the Graphics page set Detail to Fine; confirm with clicking OK.

The default setting for the start and end profiles does not prescribe a loft direction. As a result of this the tangent of the lofted surface does not match the tangent of the adjacent surfaces.

In the following steps you will test the available loft direction options to create a surface with continuous tangent across the profile.

- 4 In the Start Profile section, from the Loft direction list select Parallel.
- 5 In the End Profile section, from the Loft direction list select Parallel.
- 6 For both the Start Profile and End Profile sections make sure that the Relative to list is set to Adjacent faces.
- 7 Click Build Selected .

With the loft direction parallel to the adjacent faces the tangent of the lofted surface matches that of the adjacent surfaces along the edges of the start and end profiles. While this results in a better surface close to the profiles it may be good to examine the surface in other places. In this case the edge highlighted in the figure is no longer straight.

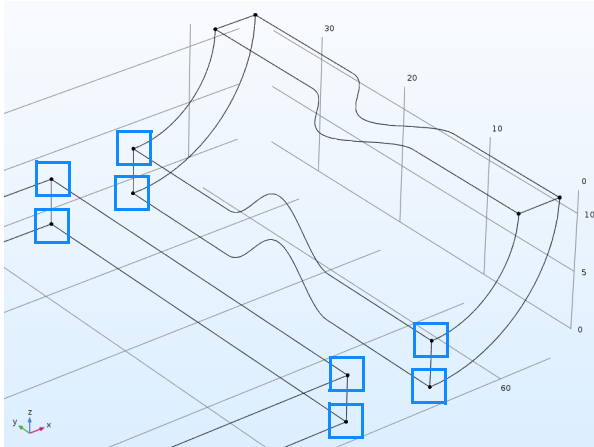


- 8 In the Start Profile section, from the Loft direction list select Perpendicular.
- 9 From the Relative to list select Profile faces.
- 10 In the End Profile section, from the Loft direction list select Perpendicular.
- 11 From the Relative to list select Profile faces.

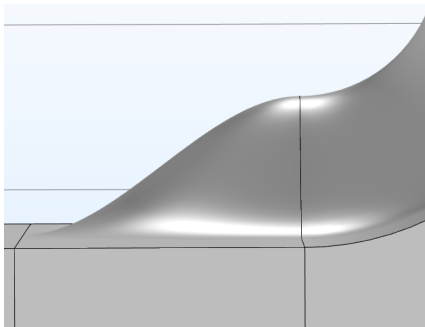
The perpendicular to profile faces loft direction ensures that the tangent of the lofted surfaces matches the normal vector of the profile faces at vertices along

the profile curve. In this case there are four vertices, one at each corner of the start and end profiles.

Vertices along the profile curves




Click Build Selected  .





The overall result is better compared to the Parallel loft direction, since the constraint on the tangent influences the entire surface, even though it is applied only at the vertices.

GUIDE CURVES

To get an even better result create guide curves to be used with the loft operation. The guide curves connect the profile curves, thus prescribing a shape that the lofted surface will follow. Each guide curve must have a continuous tangent, and intersect each profile curve exactly once. Note that the guide curves cannot be used with the loft direction set to parallel to adjacent faces. Both the Perpendicular and Not prescribed loft directions support the use of guide curves.


- In the Model Builder right-click the Loft 1 node, then select Build Preceding  .

The above step is important as it makes the Block 1 node the current node, which is symbolized by a green square around the icon for the node. The geometry operations that you add to the sequence are going to be inserted after the current node.

- In the Geometry toolbar click More Primitives , then choose Polygon .
- In the Settings window for Polygon, from the Data source list select Table.
- In the table enter the following coordinates:


X (MM)	Y (MM)	Z (MM)
$1X - rOut - 2 * thk$	$1Y / 6 - 1$	thk
$1X - rOut$	$1Y / 6 - 1$	thk

You can copy and paste the expressions from the table above.

- Click Build Selected  .
- To create the second curve right-click Polygon 1 and from the menu select Duplicate.
- Use the following coordinates for Polygon 2:

X (MM)	Y (MM)	Z (MM)
$1X - rOut - 2 * thk$	$1Y / 3 + 1$	thk
$1X - rOut$	$1Y / 3 + 1$	thk

Since the curve is a copy of the previous one you need only edit the y coordinates.


- Click Build Selected  .
- Right-click Polygon 2 and from the menu select Duplicate.
- In the Coordinates table for Polygon 3 enter:

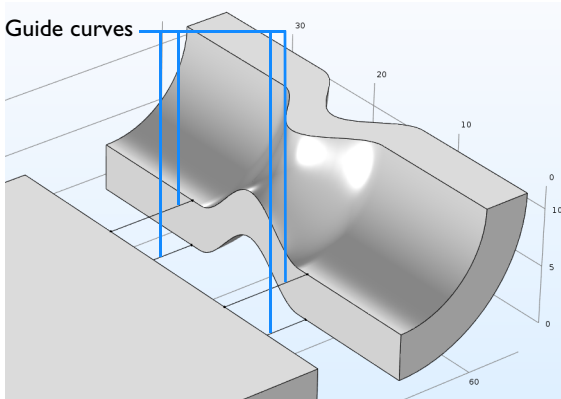
X (MM)	Y (MM)	Z (MM)
$1X - rOut - 2 * thk$	$1Y / 3 + 1$	0
$1X - rOut$	$1Y / 3 + 1$	0

- Click Build Selected  .
- Finally, to create the last curve right-click Polygon 1 and from the menu select Duplicate.

13 In the Coordinates table for Polygon 4 enter:

X (MM)	Y (MM)	Z (MM)
1X-r0ut-2*thk	1Y/6-1	0
1X-r0ut	1Y/6-1	0

14 Click Build Selected .



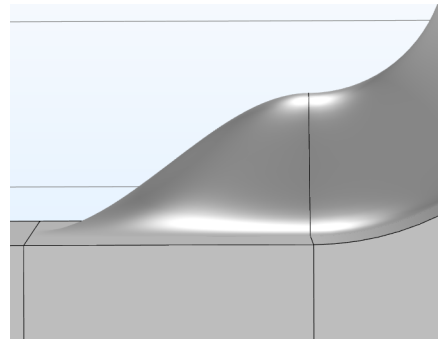
15 In the Model Builder click the Loft 1 node, then in the Settings window for Loft expand the Guide Curves section.

16 Under Guide objects click the Active button.

17 From the Graphics window select the four guide curves: po11, po12, po13, po14.

18 Click Build Selected .

Using a loft direction perpendicular to profile faces together with guide curves results in lofted surfaces that have desired continuity for the surface tangent.



About CAD File Formats

To better understand the file import related functionality of the Design Module, 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. The Design Module 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; instead, they store only triangular meshes of the surfaces. The most common examples of these types of formats are STL and 3MF.

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.

The Geometry Cleanup Wizard

This tutorial demonstrates the use of the Geometry Cleanup Wizard to defeature the geometry of a wheel rim imported from a Parasolid® file. The Geometry Cleanup is capable of automatically identifying and eliminating small geometric details, such as small faces, sliver faces, short edges, thin domains, and gaps in the geometry. One of the actions that the cleanup tool can use is the Remove Details operation, which utilizes virtual geometry operations that 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.


Virtual operations work on the finalized geometry, that is, the geometry after the Form Union or Form Assembly nodes. This means that using these operations you can remove small domains that result from intersecting solid objects. Another benefit is that they keep the curvature of the geometry, which may be important when removing larger faces, or for certain physics applications where altering the curvature of the geometry can, for example, give rise to stress concentrations.

Model Wizard

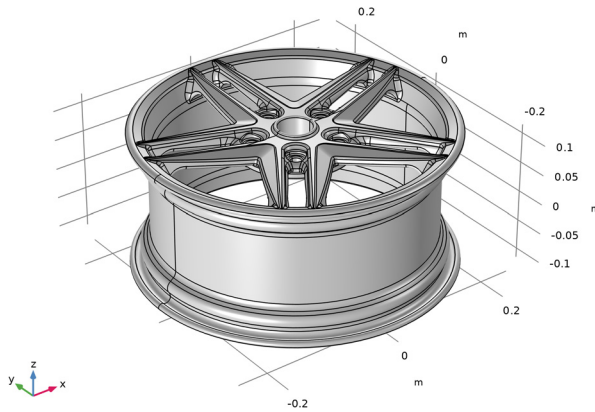
- 1 Start COMSOL Multiphysics.
- 2 Select Blank Model to skip the steps of selecting physics interfaces and study type.
- 3 In the Home toolbar, click Add Component  and select 3D.

Importing the Geometry

Before importing the CAD file, make sure that the CAD kernel is used.

- 1 In the Model Builder window, under Component 1 (comp1) click Geometry 1.
- 2 In the Settings window for Geometry, from the Geometry representation list, choose CAD kernel.
- 3 In the Home toolbar, click Import .
- 4 In the Settings window for Import, click Browse.
- 5 In your COMSOL installation directory navigate to the folder `applications/Design_Module/Tutorial_Examples` and double click the file `wheel_rim.x_b`.

6 Click Import.

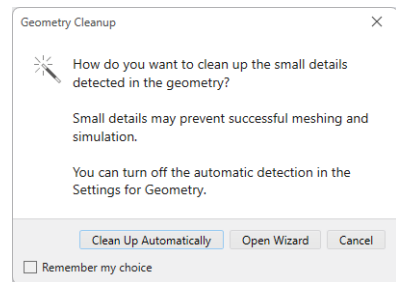



Geometry Cleanup

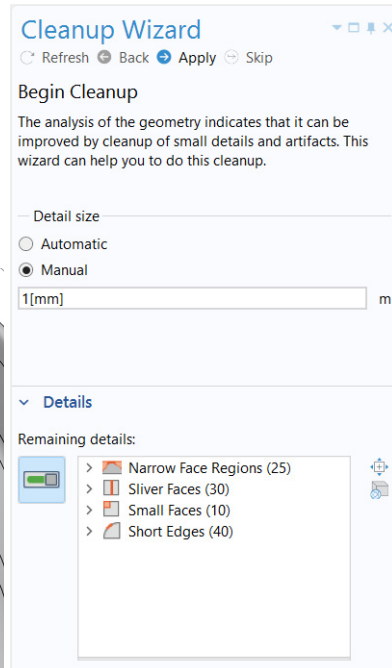
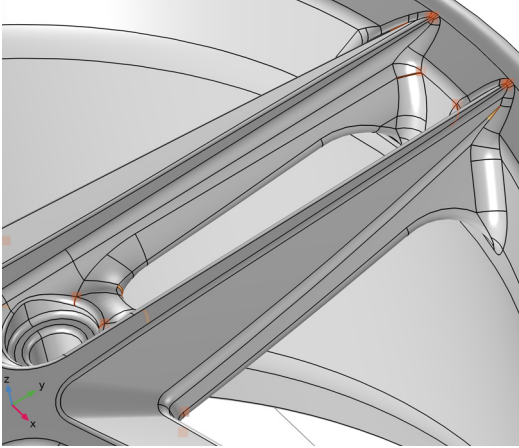
When leaving the geometry branch of the model tree, the geometry is automatically analyzed for small details.

- 1 In the Model Builder window, under Component 1 click Mesh 1.
- 2 In the Geometry Cleanup dialog that opens, click Open Wizard to open the Cleanup Wizard, to review the detected details and cleanup steps, and to modify the used tolerance.



The Cleanup can identify narrow face regions, sliver faces, small faces, short edges, narrow domain regions, and gaps. These issues are highlighted in the Graphics when the Remaining details section is active. Since this geometry has sector symmetry, the same small details are present around all of the spokes. Increase the Detail size used for the cleanup to detect more details to remove.



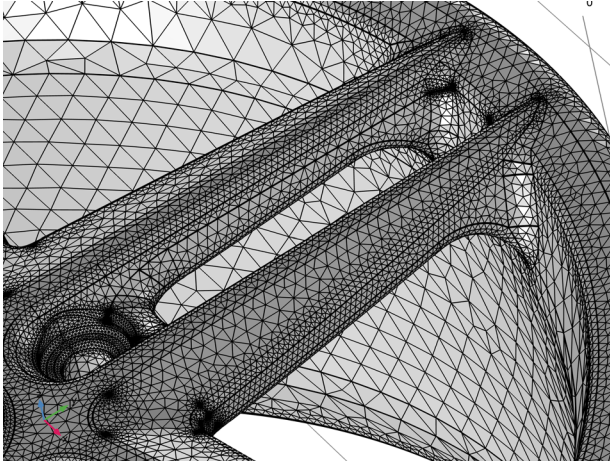
- 3 Go to the Cleanup Wizard window and click the Manual button.
- 4 In Detail size text field, type 1 [mm].
- 5 Click the Refresh  button in the window toolbar. Zoom in and rotate the rim to see some of the detected details around two of the spokes, as in the view below.




There are four types of details in the geometry: narrow face regions, sliver faces, small faces, and short edges.

- 6 Expand the sections corresponding to the detected detail types in the list. It is now possible to select the details to highlight them on the screen.
- 7 In the tree, select Sliver Faces (30) > Face 406 (3.1E-4 m) and click  Zoom to selection to zoom in on this face for a closer inspection.
Before proceeding with the cleanup process, mesh the geometry to observe the effects of the cleanup later.
- 8 Go to Mesh 1 and click Build All  .

9 Zoom out to see the mesh around the two spokes as in the figure below.



The mesh resolves well the small geometric details but it is quite large with close to 600,000 domain elements. Notice the dense mesh regions that appear as dark spots at the location of small details. Changing the element size to a coarser setting can reduce the number of elements, but this usually also leads to lower mesh element quality due to the presence of the small geometric details.

10 In the settings window for Mesh, locate the Physics Controlled Mesh section. From the Element size list, choose Extra coarse and click Build All .

This results in a mesh with about 200,000 tetrahedral elements, but the dense mesh regions remain as the mesh generator tries to resolve the details even with this coarser setting. The two information nodes that appear below the Mesh 1 node indicate that some edges and faces are much smaller than the minimum elements size setting.

Return to the Cleanup Wizard to see how the mesh can be improved by removing the small details.

- 11 Go to the Cleanup Wizard window and click Apply in the window toolbar to apply the specified detail size.

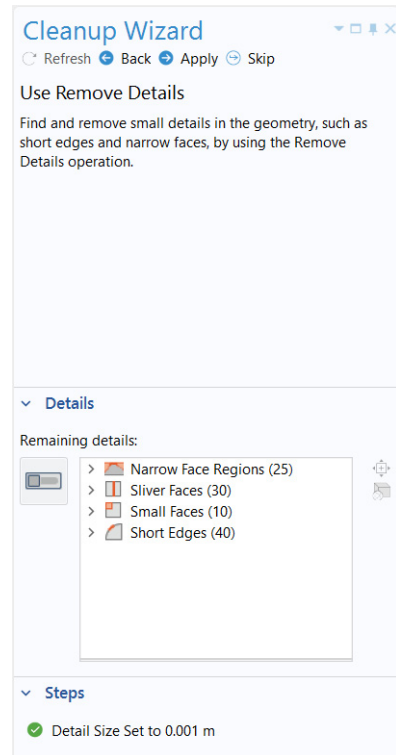
This opens the next page in the wizard that suggests to add a Remove Details operation to handle the detected details.


- 12 Click the Apply button again in the window toolbar to add and build the Remove Details node.

The Remove Details operation is a collection of virtual operations that simplify the geometry for meshing. How this is done by the sequence of operations added under Remove Details is explored in the next section of this tutorial.


The cleanup is now completed and all detected small details have been removed.

- 13 Click Done in the toolbar to close the Cleanup Wizard window.

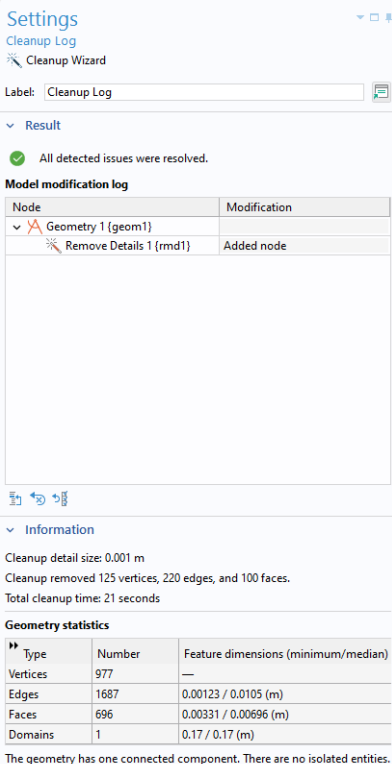


14 A Cleanup Log  node is added last in the geometry sequence, and it becomes selected with its Settings window open.

This is where you can find information about the actions performed by the geometry cleanup process. In this case, it only added a Remove Details operation, but for more complex geometries, this list can be longer. When selecting a modification in the list you can use the buttons under the table to easily go to the node in the model tree to review the changes, and to revert some or all of the modifications. Restarting the Cleanup Wizard can also be done with the button in the window toolbar.

15 In the Model Builder window, click on Mesh 1 and choose Build All .

Notice that the dense mesh regions no longer appear in this mesh of approximately 160,000 domain elements. The information nodes do not appear either under the Mesh 1 node, implying that mesh element quality has improved compared to the previous mesh.



Settings
Cleanup Log
Cleanup Wizard

Label: Cleanup Log

Result

All detected issues were resolved.

Model modification log

Node	Modification
Geometry 1 (geom1)	
Remove Details 1 (rmd1)	Added node

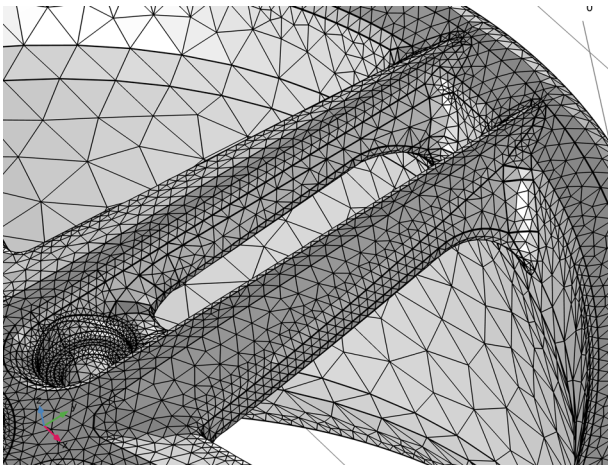
Information

Cleanup detail size: 0.001 m
Cleanup removed 125 vertices, 220 edges, and 100 faces.
Total cleanup time: 21 seconds

Geometry statistics


Type	Number	Feature dimensions (minimum/median)
Vertices	977	—
Edges	1687	0.00123 / 0.0105 (m)
Faces	696	0.00331 / 0.00696 (m)
Domains	1	0.17 / 0.17 (m)

The geometry has one connected component. There are no isolated entities.



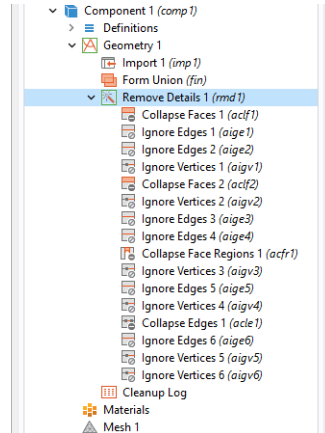
Manual Mode for Remove Details



Automatic geometry cleanup is convenient as it saves time by automatically determining suitable tolerances and operations to remove geometric detail before meshing. Reviewing and modifying the Remove Details operation added for cleaning up this wheel rim geometry is possible by switching to manual mode.

- 1 Click the Remove Details 1  node in the Model Builder.
- 2 In the Settings window, locate the Mode of operation list and set it to Manual.
- 3 Expand the sequence to reveal the automatically generated sequence of virtual geometry operations.

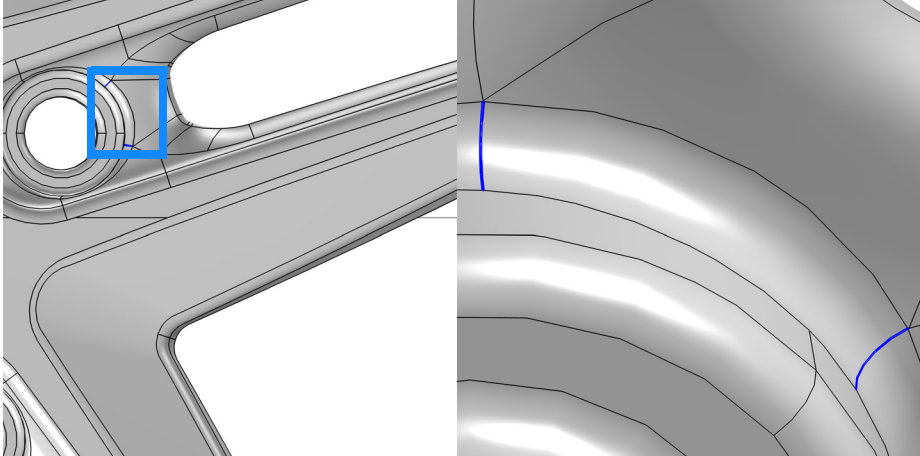
In manual mode you can examine and edit each individual operation. You could, for example, exclude entities that you do not want to remove from the geometry. You could also add further virtual operations to the automatically generated sequence to remove additional entities.

Details are removed in different passes using increasing tolerances to remove larger and larger details. A simple geometry may only need one pass, but with a more complex geometry, like this wheel rim, the Remove Details operation goes through several passes and a cleanup pass at the end. Take a closer look at some of the virtual operations in the sequence.




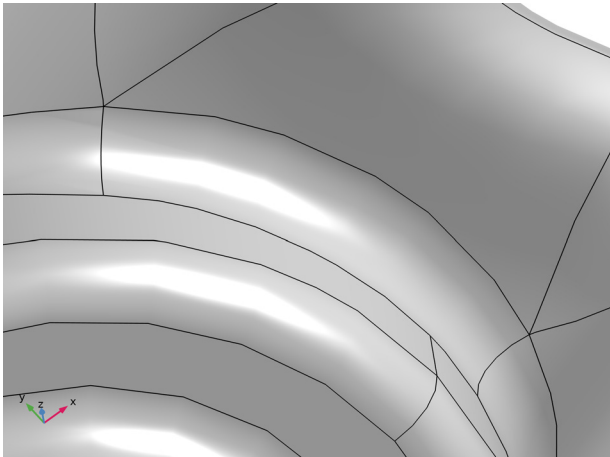
- 4 In the Model Builder window, under Component 1 (comp1)>Geometry 1>Remove Details 1 (rmd1) click Collapse Faces 1 (acfl1) .
- 5 In the Settings window for Collapse Faces, under Faces to collapse, toggle the  Activate Selection toggle button.


6 Use the Zoom to Selection  button to find some of the faces on the rim.




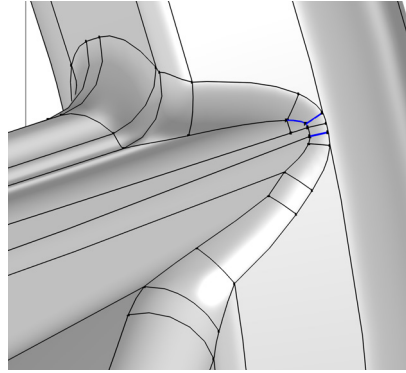
Both small faces and sliver faces are removed in this operation. Small faces with sides that are similar in length are usually collapsed into a vertex, whereas sliver faces that have a high aspect ratio are more often collapsed into one of the longer edges. As an alternative to the Collapse Faces operation you can also use the *Merge Edges* operation that gives more control over which edges are kept when collapsing faces.

7 Click the Build Selected  button.




8 In the Model Builder window, click Ignore Edges 3 (aige3) .

- 9 In the Settings window for Ignore Edges toggle the Activate Selection toggle button, then find some of the edges on the geometry by using Zoom to Selection .




All edges in the list are next to sliver faces, similarly to the image displayed to the right. The sliver faces are wider compared to those that were collapsed in the previous operation. The faces on the two sides of the edge are continuous across the edge, and it is thereby suitable to form a larger composite face by ignoring the edge.

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

- 10 Click the Build Selected  button. The geometry in the Graphics window is updated with the newly formed composite faces.

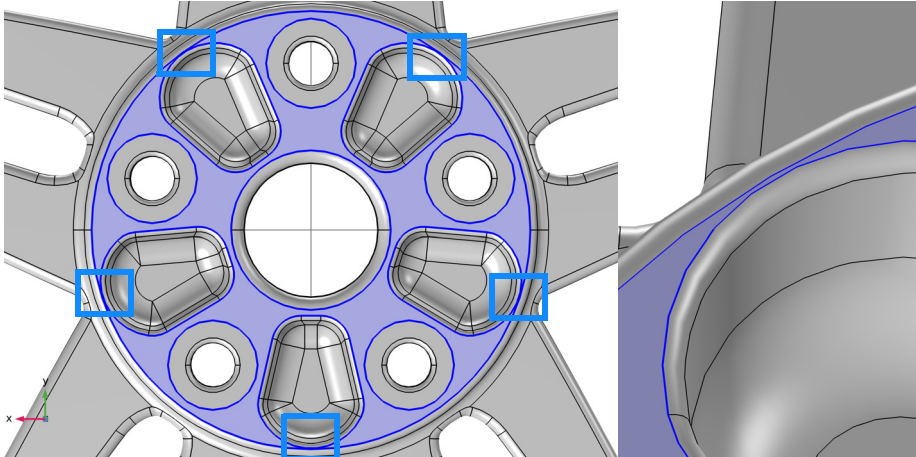
When removing edges, the adjacent vertices are deleted by downstream Ignore Vertices operations in the sequence. The sequence of virtual operations generated by Remove Details includes several Ignore Edges and Ignore Vertices operations to remove slivers with increasing face width.


In the following take a closer look at the *Collapse Face Regions* operation, that is added to the sequence to detect and remove narrow face regions.

- 11 In the Model Builder window, click Collapse Face Regions 1 (acfr1) .

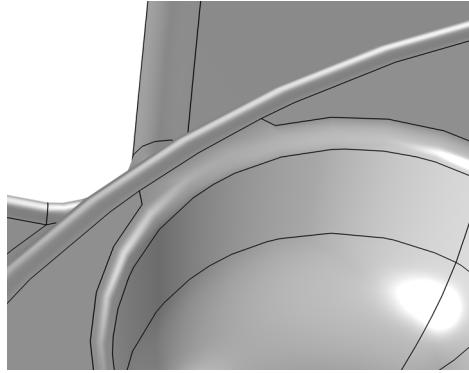
- 12 In the Settings window toggle the Activate Selection toggle button.

The face with the narrow regions is on the back of the rim.





- 13** Click Build Selected  in the Settings window for Collapse Face Regions.

The operation removes a detected narrow region by first partitioning the edges adjacent to it by creating vertices. Then, new edges are created between the vertices to partition the face. This creates a sliver face in place of the narrow region. As the last step, the created sliver face is collapsed into an edge.




Finally, switch Remove Details back to automatic mode, and mesh the geometry.

- 14** In the Model Builder window, click Remove Details 1 (rmd1) .
- 15** In the Settings window for Remove Details, locate the Mode of operation list and set it to Automatic.
- 16** In the Mesh toolbar, click  Build All.

Removing Small Geometric Entities with Repair


The step-by-step instructions in this tutorial demonstrate how small geometric details can be removed from imported CAD objects using the Repair operation. A difference when compared to the Remove Details operation used in the previous tutorial is that Repair is applied to geometric objects, which can then be exported to CAD formats, such as IGES, STEP, ACIS[®], and Parasolid[®]. Remove Details uses virtual geometry operations with a resulting virtual geometry that cannot be exported to the mentioned CAD formats.

Model Wizard

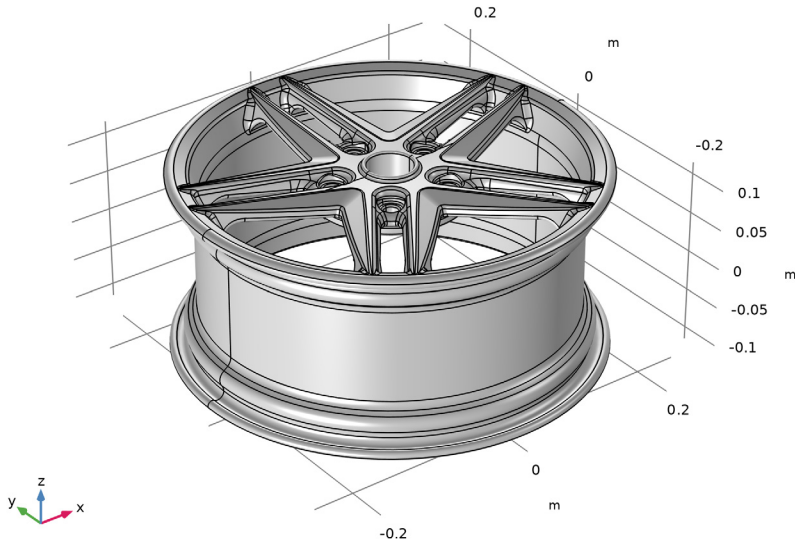
- 1 Start COMSOL Multiphysics.
- 2 Select Blank Model to skip the steps of selecting physics interfaces and study type.
- 3 In the Home toolbar, click Add Component  and select 3D.

Importing the Geometry



This tutorial uses the wheel rim geometry that appeared in the previous example. Make sure that the CAD Import Module geometry kernel is used.

- 1 In the Model Builder window, under Component 1 (comp1) click Geometry 1.
- 2 In the Settings window for Geometry, from the Geometry representation list, choose CAD kernel.
- 3 In the Home toolbar click Import .
- 4 In the Settings window for Import click the Browse button.
- 5 In your COMSOL installation directory navigate to the folder applications/Design_Module/Tutorial_Examples and double click the file wheel_rim.x_b.
- 6 Click Import.

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



Repairing the Geometry

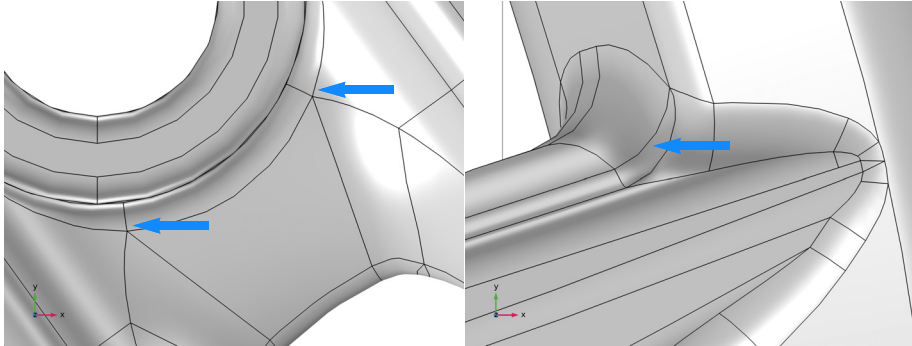
- 1 In the Geometry toolbar click Defeaturing and Repair  and choose Repair .
- 2 In the Graphics window select the wheel rim to add it to the Input objects list.
- 3 In the Absolute repair tolerance text field enter 1[mm].

Absolute repair tolerance:
 m

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. In this case, the detail size for this wheel rim geometry is known from the previous example. In addition to using the Geometry Cleanup Wizard to determine the size of small details, you can also measure the size of geometric entities using the Measure tool found in the Geometry toolbar.

- 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.



The small faces and slivers are no longer present in the geometry. According to the Information section in the Settings window for Repair, 100 short edges and 18 sliver faces were removed.

Generating a Tetrahedral Mesh

Now that the defeaturing is completed, create a volume mesh for the wheel rim.


- 1 In the Model Builder window, under Component 1 click Mesh 1.


The Geometry Cleanup dialog appears since narrow face regions, which are not removed by the Repair operation, remain in the wheel rim geometry.

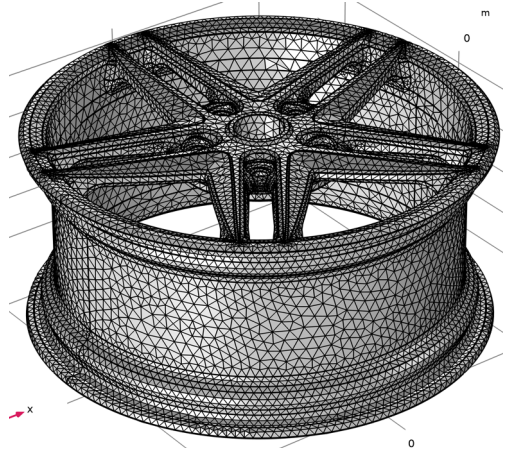
- 2 In the Geometry Cleanup dialog that opens, click Clean up Automatically to automatically clean up the geometry.

The Remove Details 1 and Cleanup Log nodes are added to the model tree. In the Information section of the Remove Details 1 node you can see that some face regions were collapsed by the operation.

- 3 In the Model Builder, click Mesh 1, and in the Settings change the Element size to Extra coarse.

- 4 Click Build Mesh  .

5 To restore the view of the mesh, in the Graphics toolbar, click the Go to Default View  button. The mesh contains approximately 164,000 tetrahedral elements.



Removing Small Geometric Entities with the Defeaturing Tools

As an alternative to the repair operation described in the previous tutorial you can also apply defeaturing tools to remove small details 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 give you more control with selective removal of features. The objects resulting after both of these methods are possible to export to CAD formats, such as IGES, STEP, ACIS®, and Parasolid®.

The defeaturing tools can detect and remove the following type of geometric details:


- small faces
- sliver faces
- short edges
- fillets

When removing small faces, sliver faces, and short edges it is good practice to start by searching for and removing entities that are only slightly larger 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.

The Cleanup Wizard can also serve as a diagnostic tool for locating small features, and can be used in combination with the defeaturing tools. After opening the Cleanup Wizard, you can check the size of some small details 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 examples.

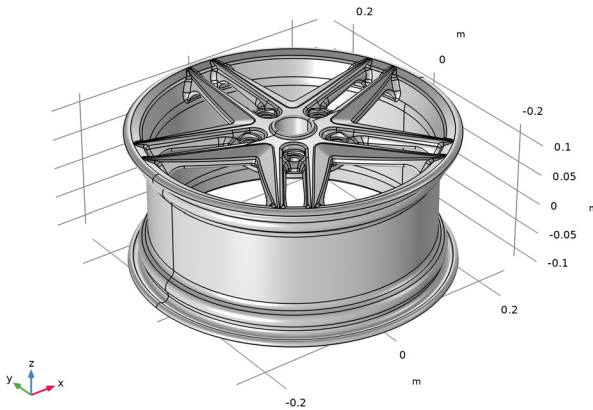
Model Wizard

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




Importing the Geometry

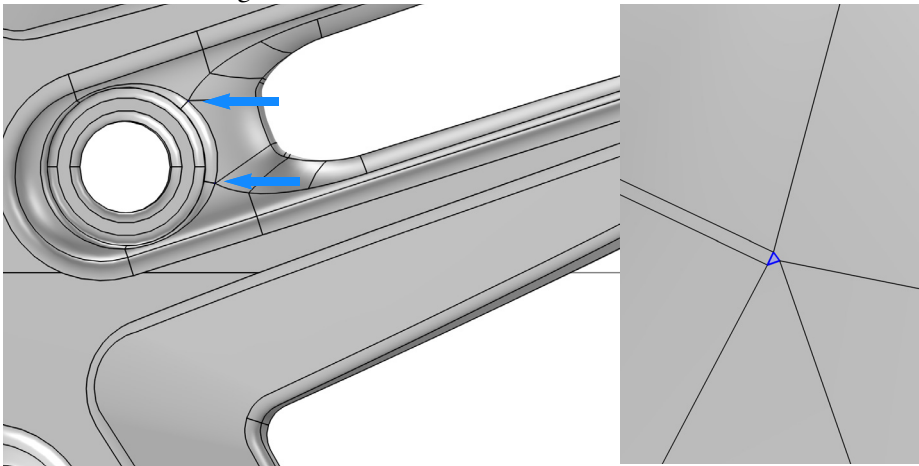
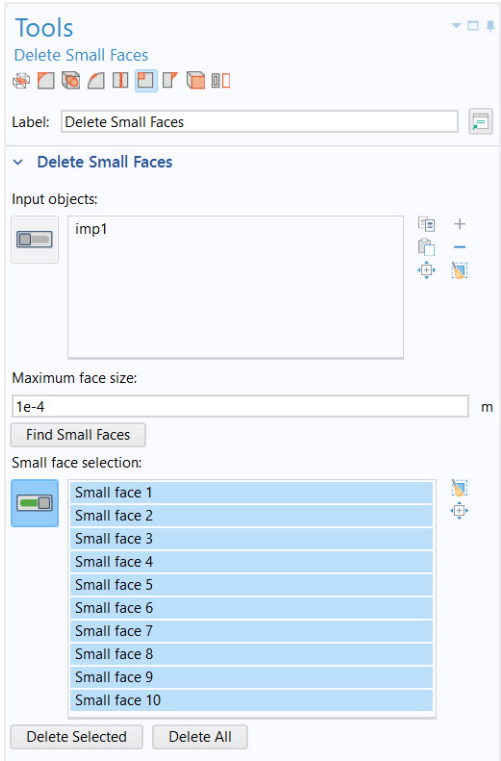
Before importing the CAD file, make sure that the CAD kernel is used.

- 1 In the Model Builder window, under Component 1 (comp1) click Geometry 1.
- 2 In the Settings window for Geometry, from the Geometry representation list, choose CAD kernel.
- 3 In the Home toolbar click Import .
- 4 In the Settings window for Import click the Browse button.
- 5 In your COMSOL installation directory navigate to the folder `applications/Design_Module/Tutorial_Examples` and double-click the file `wheel_rim.x_b`.
- 6 Click Import.




Finding and Deleting Small Faces

- 1 In the Geometry toolbar, click Defeaturing and Repair  and choose 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.
Ten faces are listed in the Small face selection list.
- 4 Use the Zoom to Selection  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.

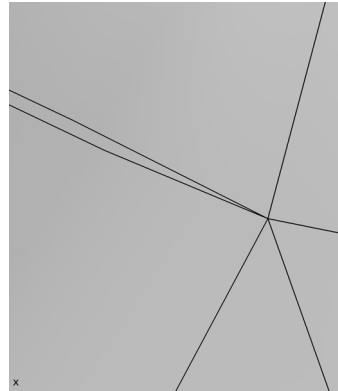


- 5 To delete all faces in the list click the Delete All button.

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, which you can access by clicking one of the buttons at the top of the window. If you happen to click a node in the geometry sequence of the Model Builder tree, the Settings window for that node will be displayed instead of the Tools window. To bring back the Tools window you can always select one of the defeaturing tools in the Geometry toolbar.



Finding and Removing Sliver Faces

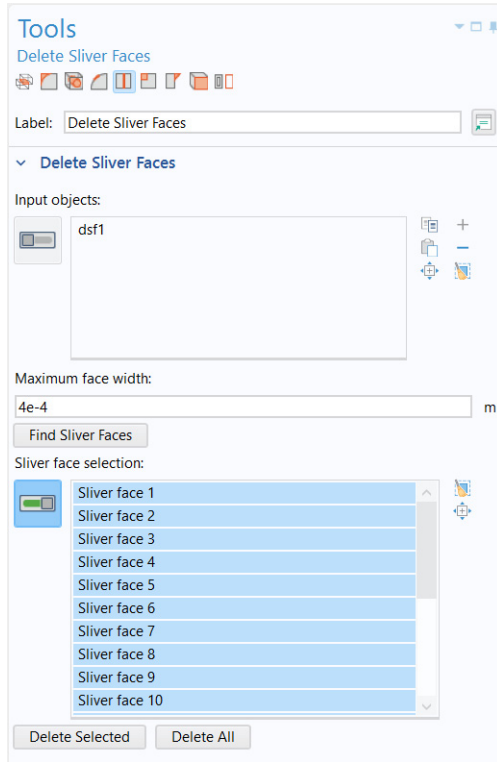
Slivers are faces with a high aspect ratio, just like the ones next to those small faces you have just deleted.

- 1 From the toolbar on the top of the Tools window, click the Delete Sliver Faces  button.
- 2 Enter $4e-4$ for the Maximum face width, then click Find Sliver Faces.


A total of 20 faces are found. In addition to the ten slivers around the bolt holes, there are ten more on the spokes. Use the Zoom to Selection  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.

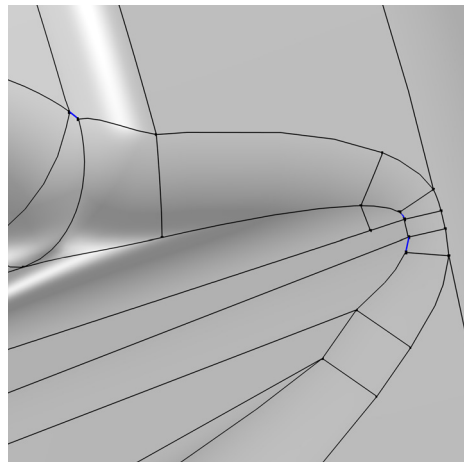
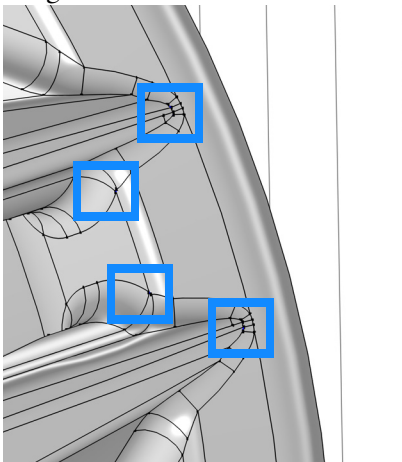
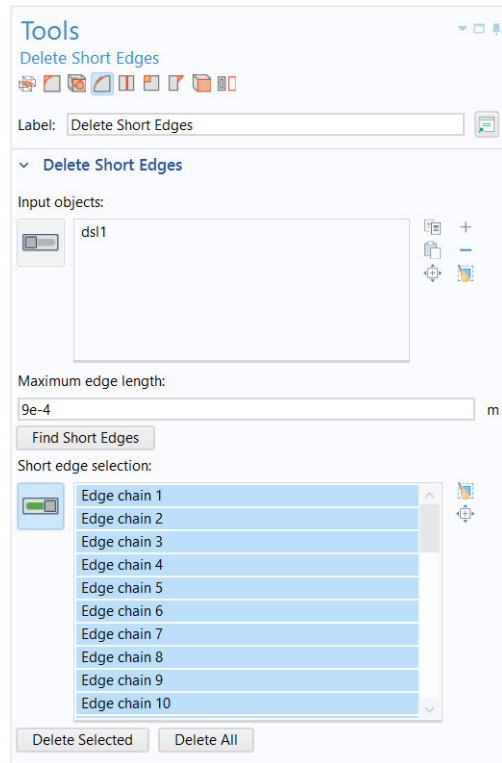


Finding and Removing Short Edges

- 1 From the toolbar on the top of the Tools window, 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.

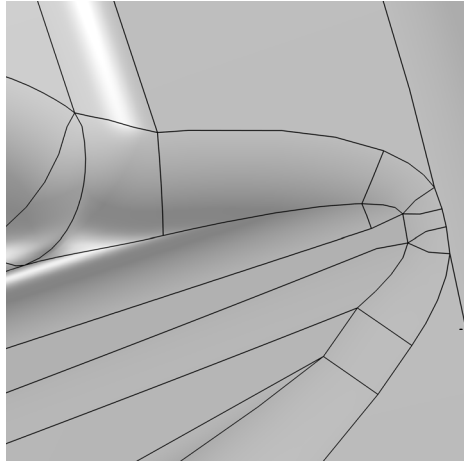
- 5 Increase the Maximum edge length to $9e-4$, then click the Find Short Edges button again. Take some time to find the edges in the list on the geometry and measure their length. They reoccur in similar places on each spoke. Some of the locations are indicated in the figure to the right.



6 Click the Delete All button.


In the resulting geometry all short edges have been collapsed to vertices. The wheel rim geometry is now similar to the one after the last repair operation in the tutorial *Removing Small Geometric Entities with Repair*. The difference is that by using the defeating tools you can have more control over which entities to delete and in which order.

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




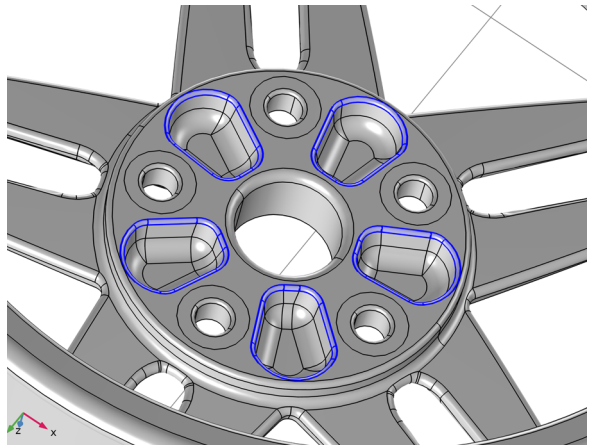
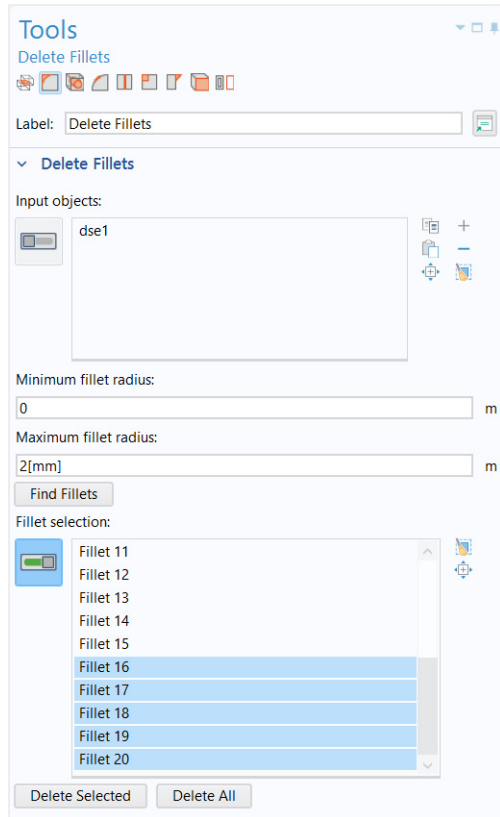
Finding and Deleting Fillets

In the previous tutorial, *Removing Small Geometric Entities with Repair*, we left the wheel rim geometry with narrow face regions remaining on a face. Here you will delete fillets adjacent to some of the narrow face regions to thereby make the face wider.

- 1 From the toolbar on the top of the Tools window, click the Delete Fillets  button.
- 2 If not already selected, add the wheel rim to the Input objects list.
- 3 In the Maximum fillet radius text field enter 2[mm].
- 4 Click the Find Fillets button.


Twenty fillets with a radius smaller than 2 mm are found. Remove only the fillets around the cavities on the back of the wheel rim. These are highlighted in the figure further below.

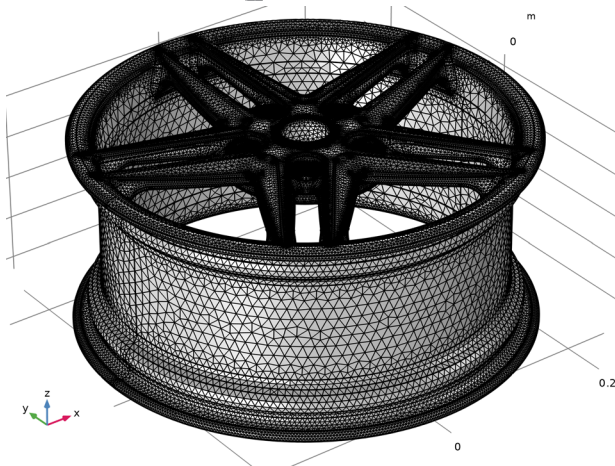
- 5 First, click Clear Selection  next to the Fillet Selection list.
- 6 Select Fillet 16 to 20 from the list, or, in the Graphics window, select a face each from the fillets highlighted in the image to the right.
- 7 Click the Delete Selected button.



Generating a Tetrahedral Mesh

As a final step generate the default tetrahedral mesh for the wheel rim.

- 1 In the Model Builder window, under Component 1 click Mesh 1.
Remaining narrow face regions are detected by the Geometry Cleanup.
- 2 In the Geometry Cleanup dialog that opens, click Clean up Automatically to automatically clean up the geometry.
The remaining narrow face regions are collapsed by the Remove Details 1 node added by the cleanup.
- 3 In the Model Builder, click Mesh 1 and change the Element size to Extra coarse.
- 1 Click Build Mesh  .




The generated mesh consists of approximately 152,000 elements and there should be no Information nodes appearing under the Mesh 1 node due to small geometric entities.

Creating Modeling Domains Around an Imported Geometry

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 a loudspeaker enclosure, demonstrates how to create domains inside and outside an imported CAD assembly. The following steps are covered:


- Importing a STEP file
- Finding and deleting holes and other details from the geometry
- Creating the union of all imported objects needed to generate a domain inside the objects
- Using the Cap Faces operation to create a domain inside the enclosure
- Creating a domain that surrounds the imported enclosure.

Model Wizard

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

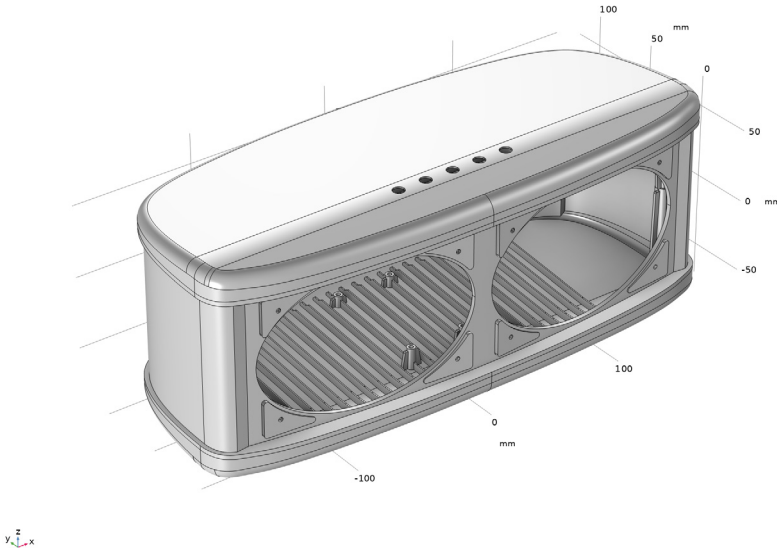
Importing the Geometry

Before importing the CAD file, make sure that the CAD kernel is used.

- 1 In the Model Builder window, under Component 1 (comp1) click Geometry 1.
- 2 In the Settings window for Geometry, from the Geometry representation list, choose CAD kernel.
- 3 In the Home toolbar, click Import .
- 4 In the Settings window for Import click the Browse button.
- 5 In your COMSOL installation directory navigate to the folder `applications/Design_Module/Tutorial_Examples` and double click the file `loudspeaker_enclosure.step`.

- 6 From the Length unit list choose From CAD document.
- 7 Click Import.


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



The enclosure consists of four solid objects.

Opening the Selection List Window

When working with imported CAD assemblies it is helpful to view a list of the imported objects. You can do this in the Selection List window.

- 1 In the Geometry toolbar, click  Selection List.
- 2 In the Selection List window that appears, click through the listed objects to identify each component.

The name of an imported object is derived from the tag of the import feature and the name of the object from the imported CAD file, or a number when the file does not contain any names. The list also indicates the object type, which can be solid (objects with an interior volume), surface, or mixed. The lower



section of the window that lists named selections is empty as there are no named selections in the model.

Three of the four imported solid objects represent the top, bottom, and mid sections of the speaker enclosure, with an additional object in the back that contains the holes for the connections.

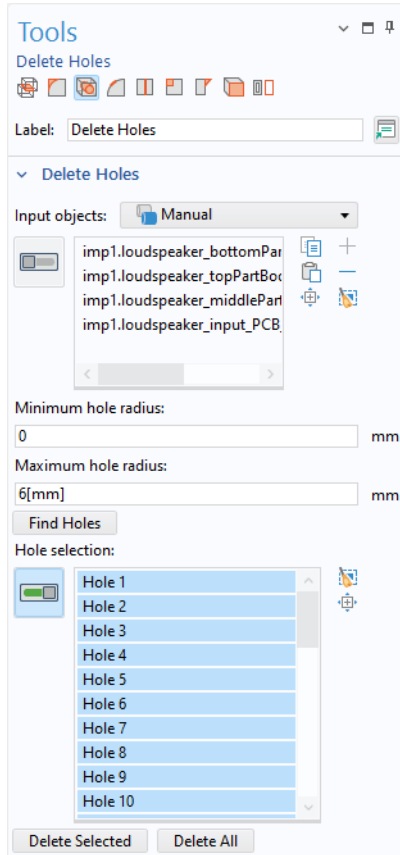
Deleting Holes

To create a domain inside the enclosure we need the imported objects to form a watertight structure. A first step in achieving this is to apply the Delete Holes operation that can find and delete cylindrical holes in solid objects.

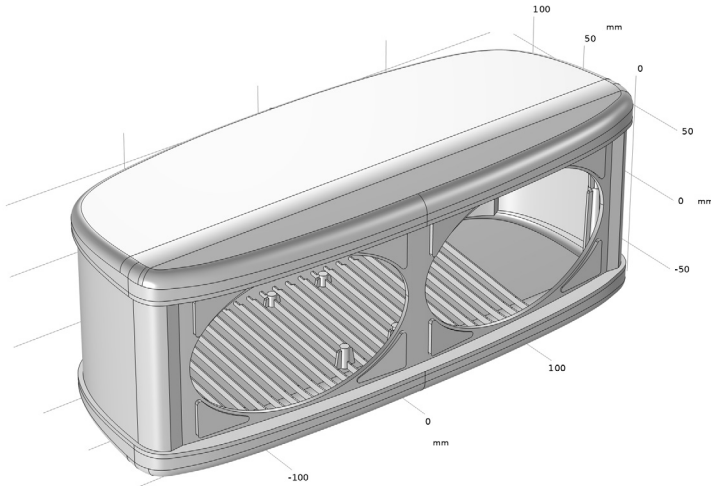
Note: A geometry object that contains more than one domain is a nonmanifold object for which defeaturing operations, such as deleting holes, are not supported. Thus it is recommended to use the defeaturing tools before computing the union of the imported objects.

- 1 In the Geometry toolbar, click  Defeaturing and Repair and choose  Delete Holes.
- 2 In the Tools window all objects already appear in the Input objects list, and you can enter 6[mm] in the Maximum hole radius text field.
- 3 Click Find Holes.

The tool found 36 holes, which appear in the Hole selection list, and they are also highlighted on the geometry.




4 Click Delete All to remove all holes.




The holes are now deleted, and a Delete Holes 1 (dho1) node is inserted into the geometry sequence.

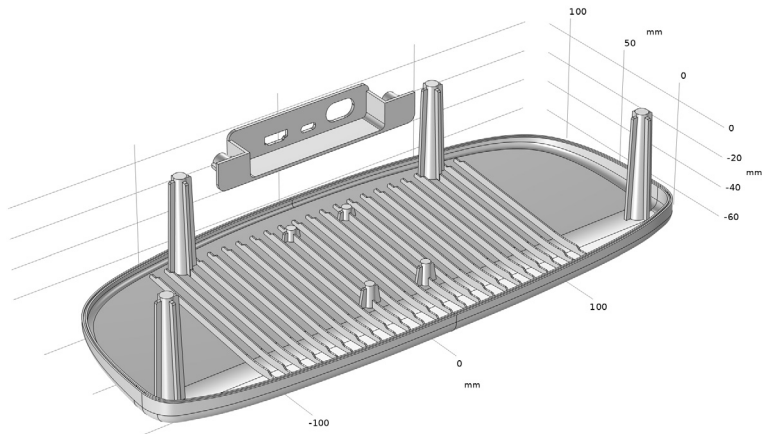
Deleting Details



The holes that now remain in the back of the enclosure for the connections do not have a cylindrical cross section. You can delete them using the Replace Faces tool, which is suitable for deleting various geometric details that may not be important for a simulation. In addition to the holes for the connections, use this tool to delete some of the mounting details from the bottom part of the enclosure.

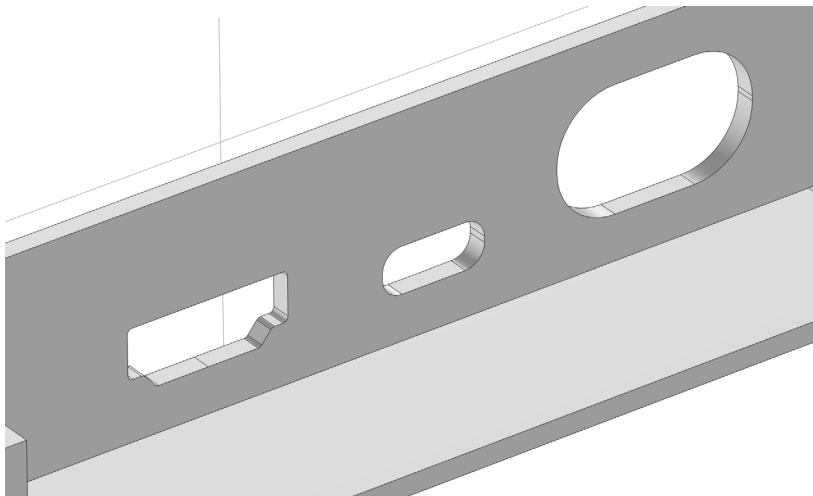
5 Click the  Select Objects button in the Graphics toolbar.


6 Click the  Click and Hide button in the Graphics toolbar.

- 7 In the Graphics window select the objects for the enclosure mid and top sections (dho1(2) and dho1(3)) to hide them from view.




- 8 Click again the  Click and Hide button in the Graphics toolbar to turn off the click and hide mode.
- 9 Click the  Zoom Box button in the Graphics toolbar and zoom in to the holes in the back.



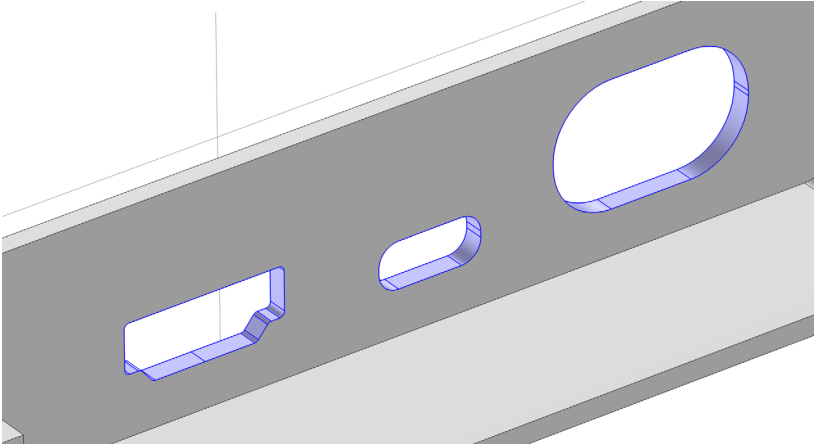
10 In the Tools window for Delete Holes, click  Replace Faces.

Using this tool you can delete larger geometric details, and the resulting gap is automatically healed by one of two methods: the surrounding faces are grown or shrunken to cover the gap; a new face is constructed based on the edges of the gap.

Add the boundaries of the holes to the Faces to replace list in the Tools window for Replace Faces. The easiest to do this is as follows.

11 Click the  Select Box button in the Graphics toolbar.

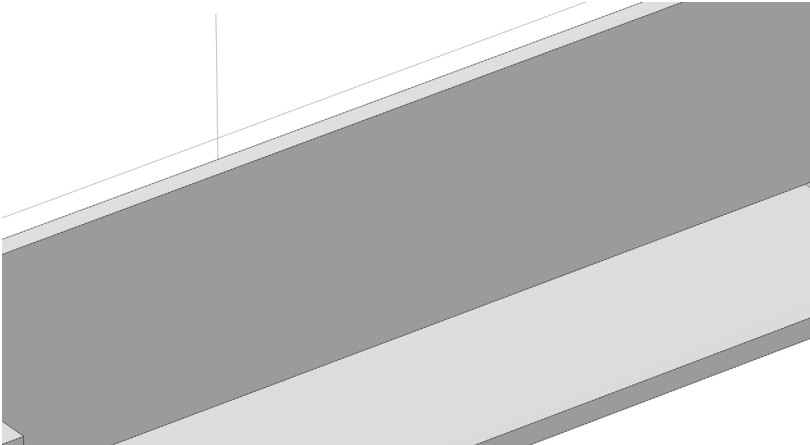
12 Hold the left mouse button to draw a square around the holes to select their faces.





13 In the Tools window for Replace Faces, select the Heal as through hole checkbox.


14 In the Tools window for Replace Faces, click Replace Selected.

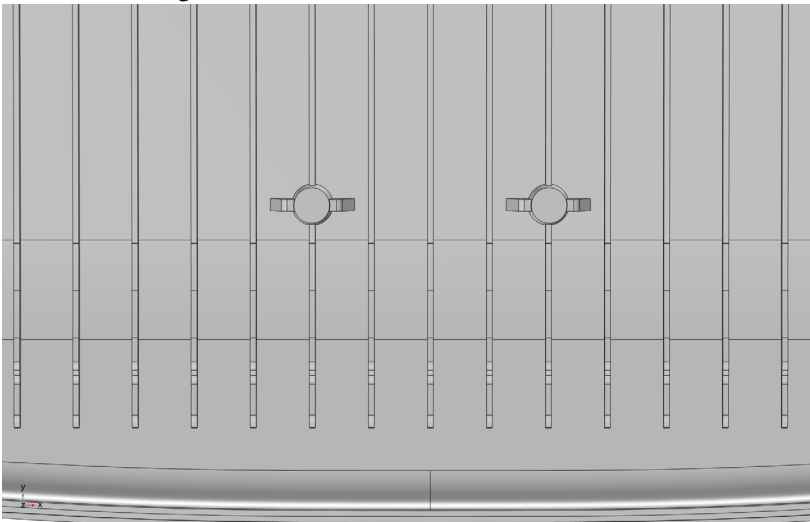
The holes are removed and the surrounding boundaries are extended to cover the gap.




15 Click the  Zoom Extends button in the Graphics toolbar.

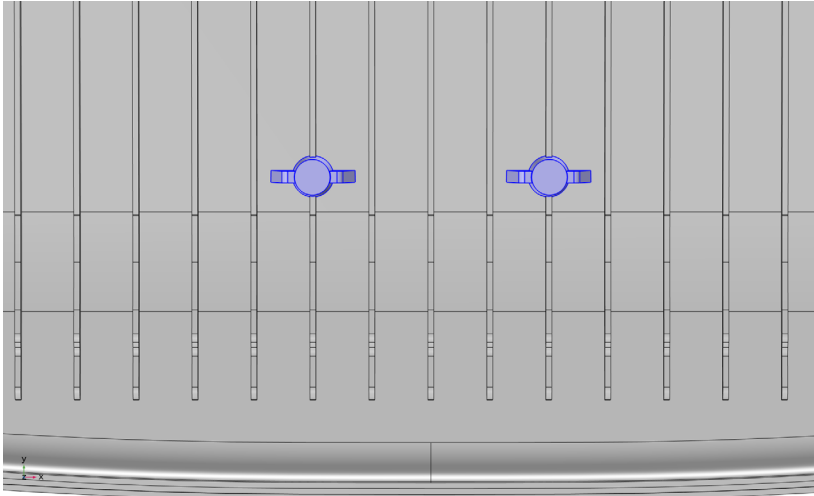
16 Click the  Go to XY View button in the Graphics toolbar.

17 Click the  Zoom Box button in the Graphics toolbar and zoom in to the two mounting details in the front.




18 Click the  Select Box button in the Graphics toolbar.

19 Hold the left mouse button to draw a square around the faces highlighted below.

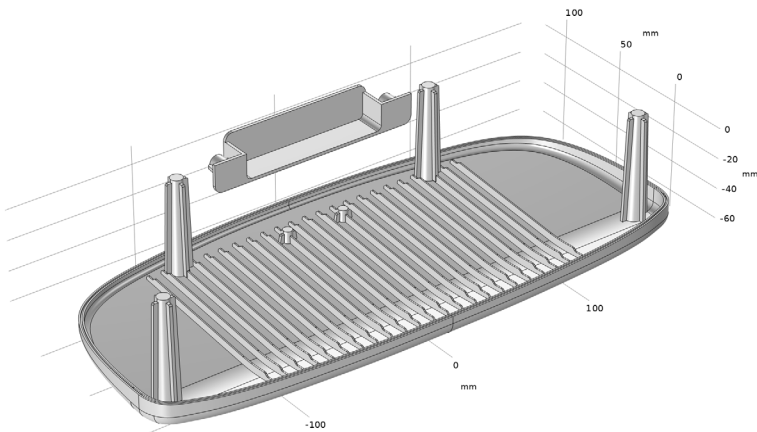



20 In the Tools window for Replace Faces, clear the Heal as through hole checkbox.

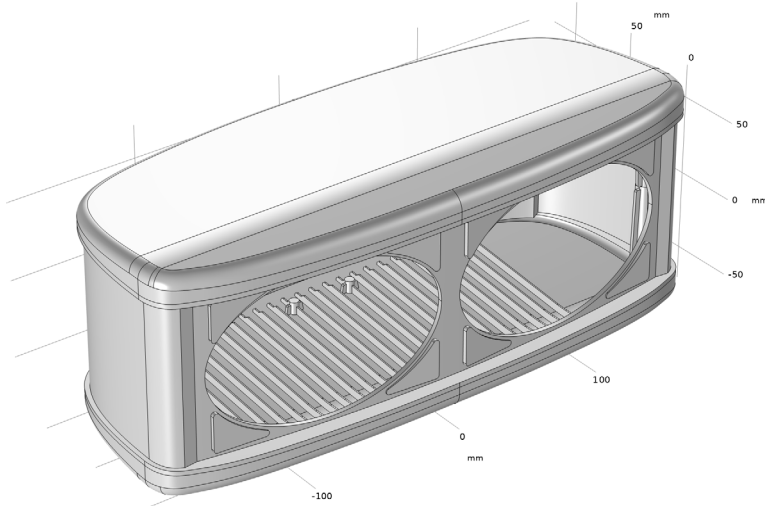
21 In the Tools window for Replace Faces, click Replace Selected.

22 Click the  Go to Default View button in the Graphics toolbar.

The two mounting details are removed from the geometry.




- 23 Click the  Reset Hiding button in the Graphics toolbar, and in the Confirm Delete dialog that opens click Delete all hide for geometry nodes.




Creating the Union of All Objects

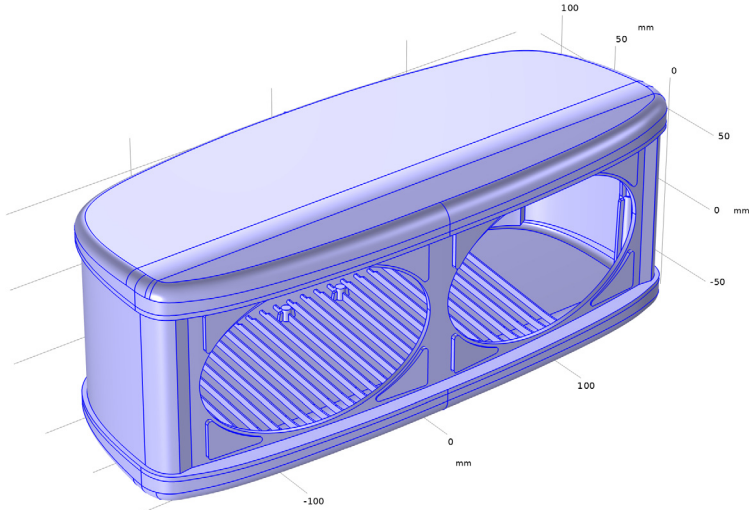
To create faces that cover the openings for the speaker elements use the Cap Faces operation, which can also generate a solid domain inside the enclosure if the created faces result in a new closed volume. To make sure that this is the case you will now create the union of all imported objects.

- 1 In the Geometry toolbar, click  Booleans and Partitions and choose Union.
- 2 In the Settings window for Union, from the Input objects list, choose All objects.
- 3 Clear the Keep interior boundaries checkbox.

With this checkbox cleared the object resulting from the union will have a single domain. If you instead would keep the interior boundaries, the resulting object would have several domains corresponding to the different parts of the enclosure and any intersections between these. This can be useful for assigning different material properties for the different parts.

- 4 In the Settings window for Union, click  Build Selected.
Examine this new geometry object using the Measure tool.
- 5 Right-click Geometry 1 and choose Measure.

6 Select the object uni1 in the Graphics window.

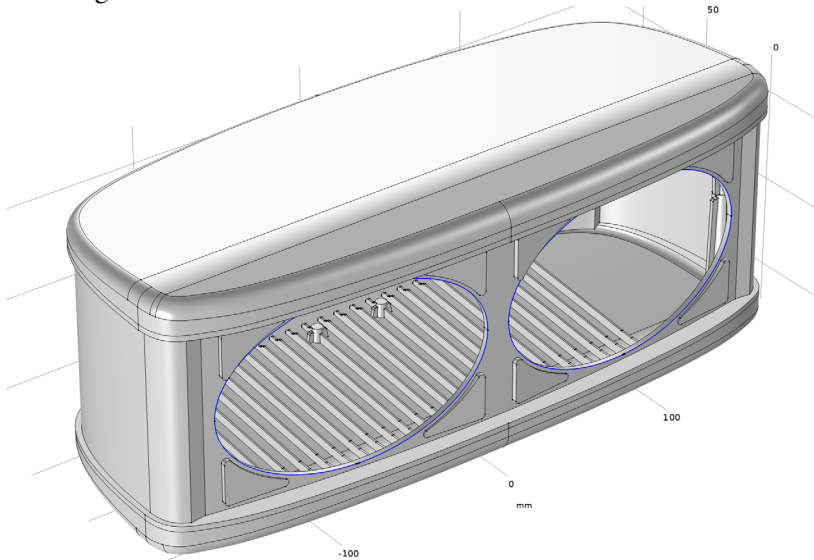


According to the information displayed in the Measure window the object uni1 contains 1 domain and 1400 faces.



Creating an Explicit Selection

The Cap Faces operation, which you will use to create a domain inside the enclosure, needs an input in form of the bounding edges of the empty volume that

should be turned into a solid. For this geometry these are the edges highlighted in the figure below.




You could select these edges 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 In 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 checkbox.
- 4 From the Graphics window select one edge each from the two edge loops highlighted in the figure above. Continuous edges are automatically added to the selection. When done check that all edges are highlighted.
- 5 Switch to the Selection List window, or open it again, to check that Explicit Selection 1 now appears under Edge Selections.

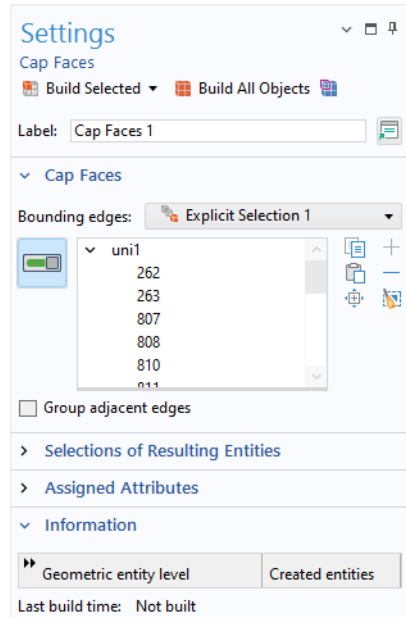
Creating a Domain with Cap Faces


At this time the geometry is ready for creating the inside domain.

- 1 In the Geometry toolbar, click  Defeaturing and Repair and choose Cap Faces.

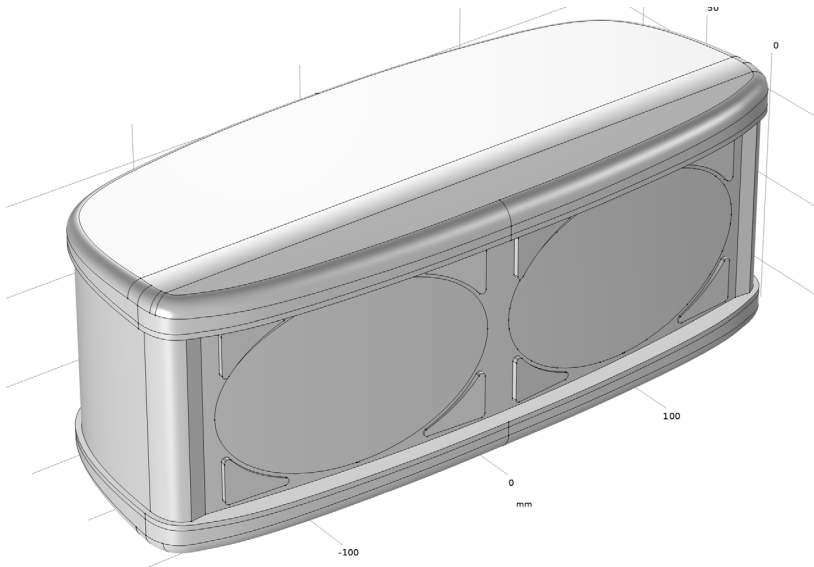
The bounding edges of the empty volume inside the manifold are included in Explicit Selection 1 (sel1).

- 2 In the Settings window for Cap Faces, from the Bounding Edges list choose Explicit Selection 1.



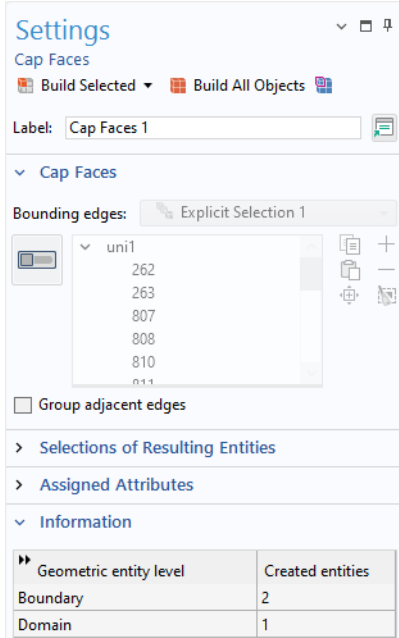
3 Click  Build Selected to complete the operation.

The operation closed off the two openings with new faces, and also created a solid domain where it used to be a void inside the enclosure.





According to the Information section of the Settings window for Cap Faces,


2 boundaries and 1 domain were created.

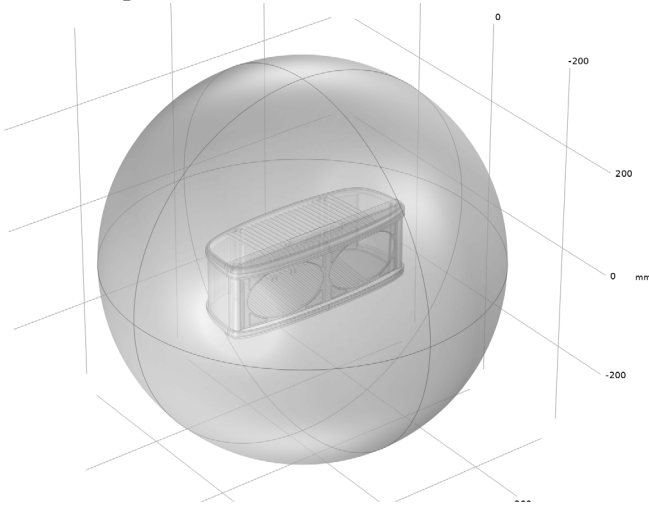


Creating a Surrounding Domain

To create a domain that surrounds the enclosure add a sphere to the geometry and set it up as a bounding ball for the geometry.

- 1 In the Geometry toolbar, click  Sphere.
- 2 In the Settings window for Sphere, locate the Definition section, and from the Defined by list, choose Bounding ball (approximate).
- 3 Select the enclosure object (cap1) in the Graphics window.
- 4 In the Margin text field, type 150[mm].
- 5 Click  Build Selected to create the sphere.

- 6 Click first the  Zoom Extents button, followed by the  Transparency button in the Graphics toolbar.



Note: The objects for the sphere and the enclosure are united by the Form Union (fin) node, which is built automatically when you leave the geometry branch of the model tree. Form Union results in three domains: the enclosure, and the volumes on the inside and outside of the enclosure.

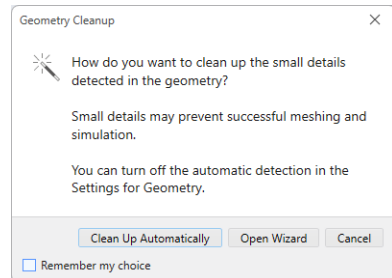
Geometry Cleanup and Meshing

When leaving the geometry branch of the model tree, the geometry is automatically analyzed for small details.

- 1 In the Model Builder window, under Component 1 click Mesh 1.
- 2 In the Geometry Cleanup dialog that opens, click Clean Up Automatically to detect and remove small details from the geometry before meshing.

As the cleanup is completed the Cleanup Log appears indicating that all detected issues were resolved. According to the information in the Settings window, a Remove Details feature was added and 47 vertices and 45 edges were deleted in the process.

- 3 In the Model Builder window, under Component 1 (comp1) click Mesh 1.



4 In the Settings window for Mesh, click  Build All.

