STL Import 1 — Generating a Geometry from an Imported Mesh[1]

1. *The STL geometry is provided courtesy of Mark Yeoman, Continuum Blue, UK.*

Introduction

The STL file format is one of the standard file formats for 3D printing, and it is also often used as a format for exchanging 3D scan data. STL files contain only the triangulated surface, which we can also call a surface mesh, of a 3D object. The triangles in the file are identified by their normals and vertex coordinates which together form a faceted representation of the object.

COMSOL Multiphysics supports import of an STL file both as a surface mesh and as a geometry with smooth faces. This tutorial series focuses on using available tools to edit imported surface meshes, the different ways of repairing the meshes, and how to generate a volume mesh from the imported surface mesh, either directly or by first creating a geometry with smooth faces from the mesh. Regardless of which method you choose to follow, COMSOL Multiphysics supports a variety of operations, for example:

- Moving, scaling, and rotating the imported mesh
- Combining the imported mesh with parameterized geometry to run parametric sweeps
- Intersecting imported meshes with each other
- Modifying and remeshing the imported mesh
- Generating a tetrahedral mesh in unmeshed domains
- Generating a swept mesh in unmeshed domains
- Creating a boundary layer mesh
- Using curved mesh to represent curved boundaries

Working with the mesh directly can be more robust in case you need to intersect several imported meshes (or intersect the imported mesh with geometric objects of more complex shapes). This can be important to consider when choosing whether to create a geometry or not.

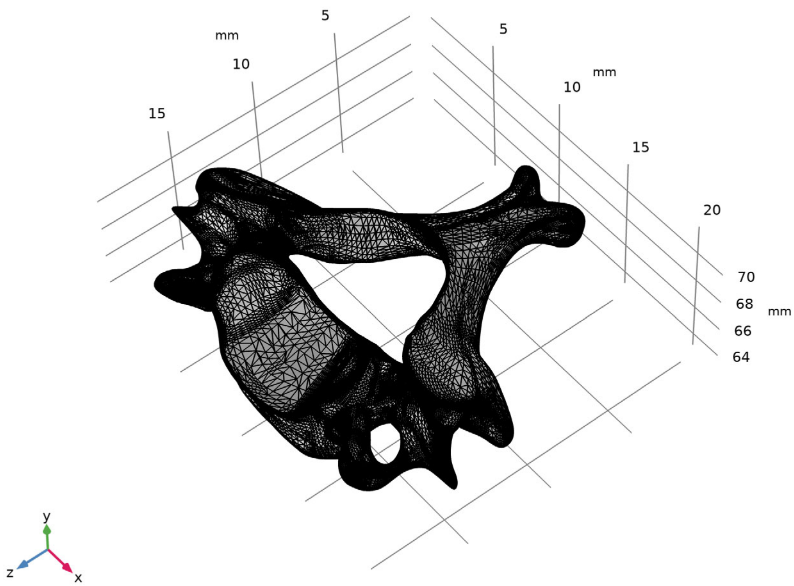
STL Import 2 — Remeshing an Imported Mesh, the second part of this tutorial series, describes the process of preparing a mesh directly by remeshing it, without creating a geometry.

The two tutorials in this series are complementary, and intend to provide a detailed insight into how to work with imported meshes. Apart from arriving at a simulation mesh in two different ways, the tutorials also cover repairing different types of defects, and different ways of visualizing the mesh. Depending on your application and the imported mesh at hand, pick and choose from the tools detailed in the tutorials to arrive at a mesh that suits your needs.

Lastly, it is important to mention that the techniques used in the tutorial series apply to any type of imported surface meshes, such as the formats PLY and 3MF. They also apply when creating a mesh from a Filter or Partition dataset, which you would do when using the results of a simulation as the mesh for a new simulation, for example during a topology optimization study.

Model Definition

Import the STL file of a vertebra geometry shown below.



Follow the instructions in this tutorial to

- Import the STL file
- Identify and fix small defects in the imported STL mesh
- Partition the object and create a surrounding volume to be used for simulation
- Generate a volume mesh for the geometry
- Create a plot to see how close the created surface is to the original STL mesh

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/stl_vertebra_import`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 Click **Done**.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

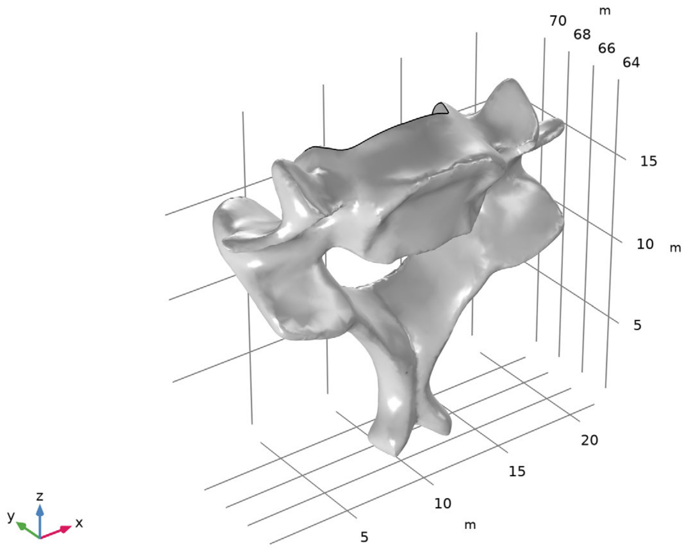
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `c6_vertebra.stl`.

5 Click **Import**.

The import of an STL file consists of three steps: the import of the surface mesh from the file, the generation of a surface geometry from the imported mesh, and finally the creation of a solid object. After the import is complete the geometry appears on the screen.



Notice that after the import completes, the reference to the STL file is no longer visible in the **Settings** window. The source for the geometry import is now **Mesh Part 1**, which is where the mesh import can be found. The **Mesh Part 1** sequence is added under the **Global Definitions>Mesh Parts** node. The output of a mesh import sequence added here can be imported into any geometry sequence in the mph file. The **Import 1** node in the geometry sequence handles the last steps of the process: conversion from mesh to geometry and creation of a solid object.

According to the information in the **Messages** window, a mixed object was generated by the import. This means that the object contains both solids and surfaces. Some parts of the STL source mesh form a "watertight" shell that can be converted into a solid. The additional faces are not needed in this case, so these must be found and deleted.

6 Click **Go to Source** to locate the source mesh in the **Model Builder** window.

MESH PART 1

In the **Model Builder** window, expand the **Global Definitions>Mesh Parts>Mesh Part 1** node.

Information

The **Import** node has an **Information** subnode which contains information about problems in the mesh.

1 In the **Model Builder** window, expand the **Global Definitions>Mesh Parts>Mesh Part 1>Import 1** node, then click **Information**.

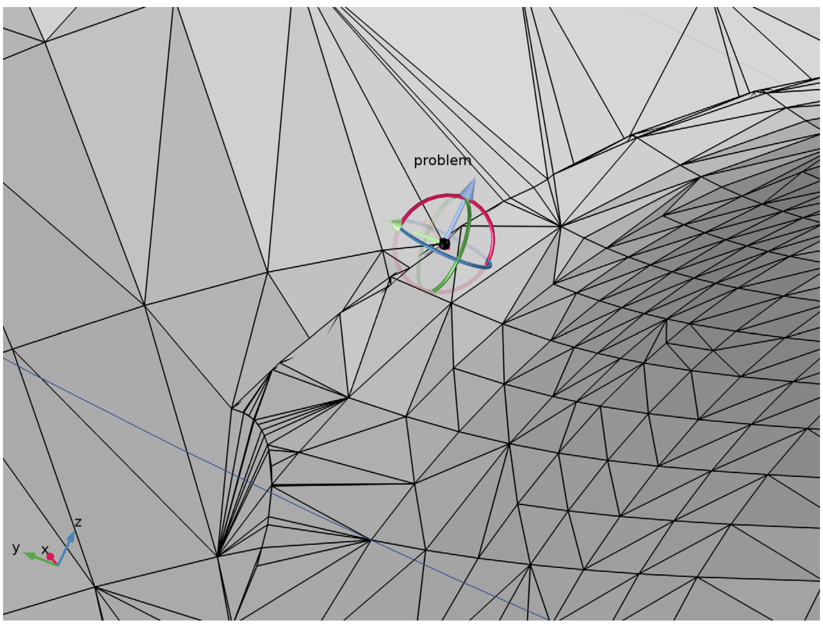
2 In the **Settings** window for **Information**, locate the **Information** section.

3 Click **Center at Coordinates**.

When the mesh import tried to create a domain within the vertebra, this failed due to intersecting mesh elements. Clip around the problem coordinate to inspect it further.

4 Click **Clip Around Coordinates**.

Rotate the mesh and zoom in even more to see the intersecting elements better. Only the first problematic coordinate is reported, but there are more similar intersecting elements in the mesh.



These intersecting elements can be fixed with a mesh **Union** operation with an absolute repair tolerance of 0.1 mm. However, this is not needed here as the simplification done when creating a geometry of the mesh takes care of this.

5 Click **Remove Clipping**.

6 Click the **Go to Default View** button in the **Graphics** toolbar.

The mesh import automatically partitions the STL mesh into boundaries based on, for example, an angle criteria. By inspecting the boundaries, it is possible to find mesh faces that are not properly connected to others. Do this by opening the **Selection List** window, clicking on each of the boundaries in the list, and checking in the **Graphics** window if the boundary is wanted or not.

7 In the **Home** toolbar, click **Windows** and choose **Selection List**.

8 In the **Model Builder** window, click **Mesh Part 1**.

9 In the **Graphics** window toolbar, click next to **Select Domains**, then choose **Select Boundaries**.

If there are many boundaries, reimport the mesh with the import setting **Boundary partitioning: Minimal** as this will partition the mesh into as few boundaries as possible, or, if reimporting the mesh takes a long time, use the **Join** operation to join the boundaries manually, as shown below.

Join Entities 1

1 In the **Mesh** toolbar, click **Join Entities**.

2 In the **Settings** window for **Join Entities**, locate the **Geometric Entity Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Click **Build Selected**.

Information

This results in an information message that not all selected entities could be joined into one. When joining boundaries, the operation tries to remove the edges between the boundaries. The **Information** node is expected due to the extra faces. Use the selection in the **Information** to find and delete the faces.

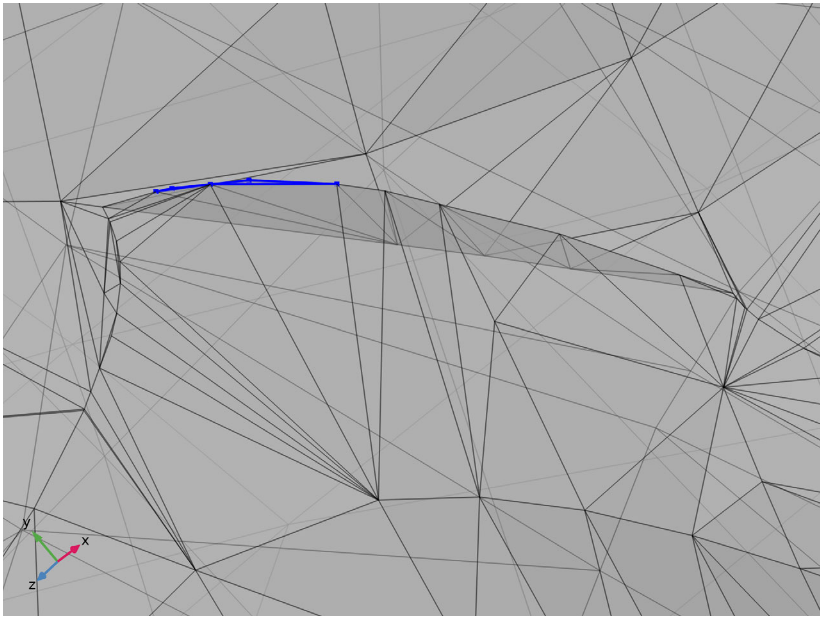
1 In the **Model Builder** window, click **Information**.

2 Expand the **Information** node and click on the second **Information1** node. In the **Settings** window, you find information that the operation was unable to delete two edges. Make sure both edges are selected in the list.

3 Click the **Zoom to Selection** button in the **Graphics** toolbar to zoom in on the selected edges. The button is also available next to the **Selection** list.

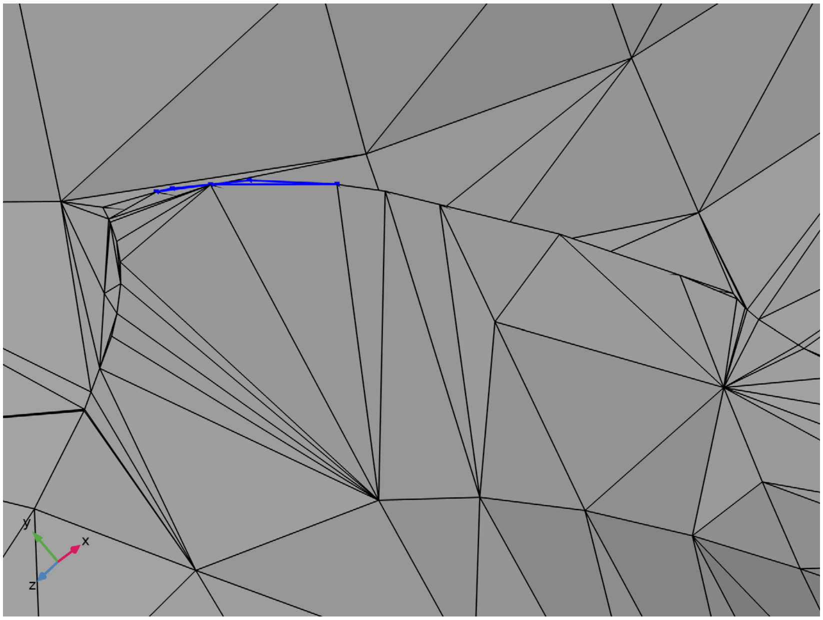
4 Click the **Transparency** button in the **Graphics** toolbar.

If the selected edges are hidden behind the mesh, zoom out, rotate the geometry, then click the **Zoom to Selection** button again to find the area highlighted in the figure below. The part of the mesh with a semielliptical shape forms a fold, as seen in the middle of the image below. Rotate the mesh slightly in the **Graphics** window to see how the mesh elements connect.



To get a better view, it may help to turn off transparency again.

5 Click the **Transparency** button in the **Graphics** toolbar.



The two edges are located at the fold in the mesh. Open the **Selection List** again, make sure **Select Boundaries** is selected, and notice that Boundaries 2 and 3 are in the same location as the short edges on the generated geometry. These are isolated boundaries on the outside of the surface mesh and are attached to the mesh only through a single edge.

In the following steps, delete these boundaries from the imported mesh.

Delete Entities 1

1 In the **Mesh** toolbar, click **Delete Entities**.

2 Click the **Select All** button in the **Graphics** toolbar.

3 In the **Settings** window for **Delete Entities**, locate the **Geometric Entity Selection** section.

4 In the list, select **1** (the large face of the vertebra).

5 Click **Remove from Selection**.

This leaves the two small faces in the list of boundaries to be deleted.

6 Click **Build Selected**.

After the **Delete** operation completes, the **Selection List** contains only one boundary for the mesh. The isolated faces are no longer there. However, the small fold in the mesh is still visible. Follow the steps below to repair this defect in the mesh.

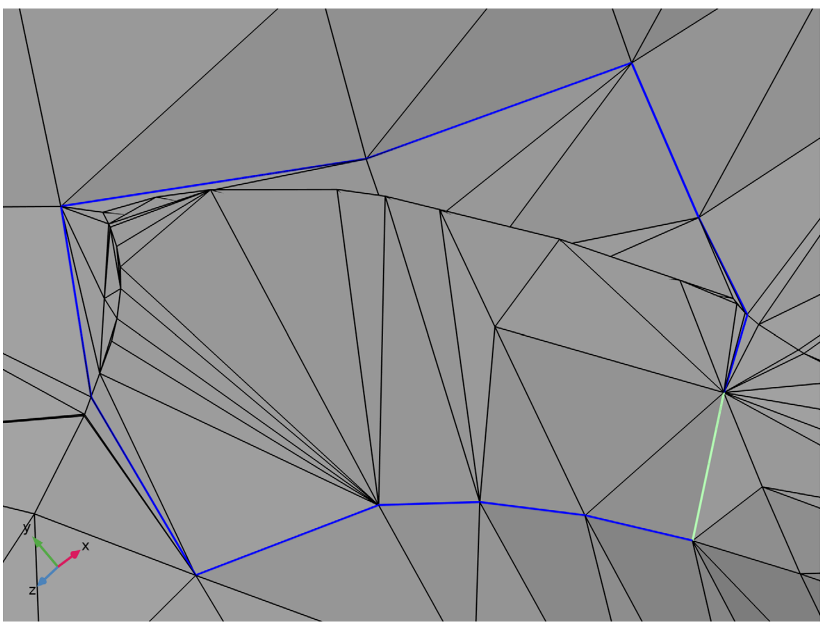
7 Close the **Selection List** window.

Create Edges 1

Sharp irregularities in the mesh, like this fold or pointy spikes, can cause problems when creating a smooth geometric surface. A smooth source mesh is therefore preferred. To remove the fold, start with partitioning the mesh to create a boundary containing the fold, delete this boundary, and finally, generate a new mesh face to cover the resulting hole.

1 In the **Mesh** toolbar, click **Create Entities** and choose **Create Edges**.

2 Select mesh edges around the fold by clicking them in the **Graphics**, similar to what is shown in the image below. The selection is most easily done with transparency off. If there are mesh edges on the opposite side of the view or are located inside a closed mesh face, it is recommended to use a **Clip Plane** or to hide boundaries in the mesh to reach those mesh edges. The selected edges can differ from the ones selected in the figure, as long as the edges form a closed loop, include the fold, and delimit only a small region around the fold. The last requirement is important when generating a new mesh face that follows the original mesh as close as possible.



3 In the **Settings** window for **Create Edges**, click **Build Selected** to generate the edges and partition the boundary.

Delete Entities 2

1 In the **Mesh** toolbar, click **Delete Entities**.

2 Select Boundary 2 only (the boundary with the fold).

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

There is now a hole in the surface mesh, which means that it isn't possible to create a solid geometry from this mesh. So, before trying to create a geometry again, the hole must be sealed off. Two operations can be used for this purpose: **Create Faces** and **Fill Holes**. The **Create Faces** operation is a manual way of generating mesh faces for holes by selecting edge loops. The **Fill Holes** operation automatically generates mesh faces to cap several holes at once. In the following tutorial, use the **Create Faces** operation that generates mesh faces for selected edge loops.

Note that the mesh faces generated by both of these operations will be minimal faces that strive to be as planar as possible.

Create Faces 1

1 In the **Mesh** toolbar, click **Create Entities** and choose **Create Faces**.

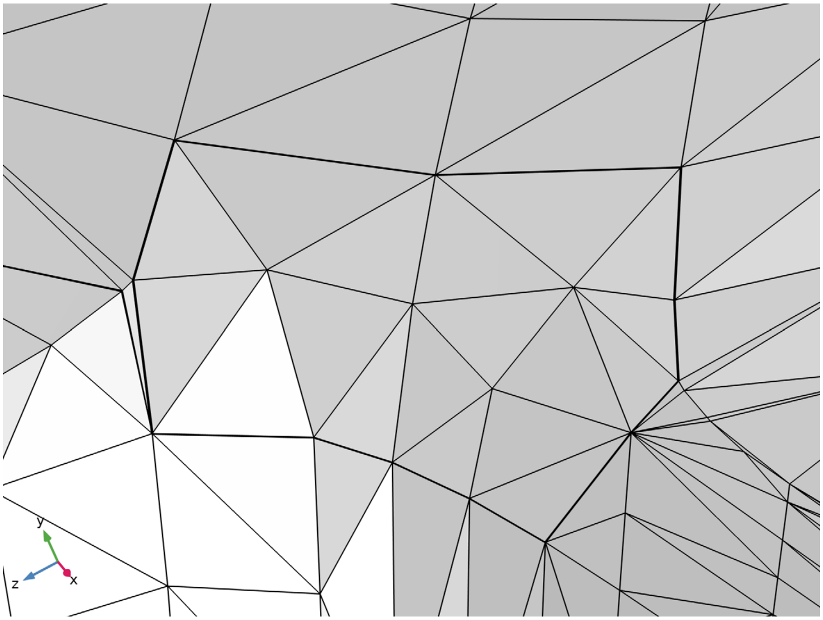
2 Select Edge 1 only (the edge of the hole).

3 In the **Settings** window for **Create Faces**, locate the **Create Faces** section.

4 Clear the **Create domains** check box to avoid getting further **Information** nodes about intersecting elements.

5 Click **Build Selected**.

The hole is sealed off with a new mesh face that is much smoother.



Join Entities 2

Now, join the two boundaries.

1 In the **Mesh** toolbar, click **Join Entities**.

2 Click in the **Graphics** window and then press Ctrl+A to select both boundaries.

3 In the **Settings** window for **Join Entities**, click **Build Selected**.

4 Click the **Go to Default View** button in the **Graphics** toolbar to again see the full mesh.

MESH PART 1: LENGTH UNIT

Before continuing with the geometry, make sure the mesh and geometry have the correct length unit. The meshing sequence in a **Mesh Part** has a length unit, just as a geometry sequence. Since STL files do not include units, the length unit is set to the default geometry length unit, meter.

1 In the **Model Builder** window, click **Mesh Part 1**.

2 In the **Settings** window for **Mesh Part**, locate the **Units** section.

3 Select the **Use units** check box.

4 From the **Length unit** list, choose **mm**.

5 Click **Build All** and zoom out to verify that the unit on the axes are indeed set to mm.

GEOMETRY 1

Although the length unit of the **Mesh Part 1** sequence now has the correct value, you also need to update the length unit of the **Geometry 1** sequence. One way to avoid to update both is to set the length unit of the geometry sequence before importing the file.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Import 1 (imp1)

A rebuild of the **Geometry 1** sequence reimports the geometry from the mesh and updates the length unit.

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Import 1 (imp1)**.

2 In the **Settings** window for **Import**, click **Build Selected**.

While waiting for the import to complete, take a look at the **Import** settings. For large meshes, it can be time consuming to locate and handle irregularities manually. Then, tuning the **Simplify mesh** parameters can provide an automatic way to simplify the mesh and remove faults to generate smoother geometry faces.

The **Relative simplification tolerance** is a global tolerance relative to the size of the geometry. If this tolerance is decreased, the generated geometry follows the shape of the original mesh more closely. An increased tolerance will remove larger details and give a more smooth face at the cost of not following the shape of the original mesh as closely.

The **Defect removal factor** is a local tolerance. An increased **Defect removal factor** removes small irregularities even if they are big relative to the local element size.

At the end of this tutorial, you can find instructions on how to create a mesh plot to see how well the created faces match the source mesh. But first, complete the geometry set up.

The **Messages** window now reports having imported one solid object, which means that it is time to continue to add a surrounding volume and cut the geometry in half to make use of the symmetry plane. To begin with, modify the orientation of the geometry to improve the visualization.

Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **imp1** only, in other words the object for the imported vertebra.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 From the **Axis type** list, choose **x-axis**.

5 In the **Angle** text field, type `90`.

6 Click **Build Selected**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar. This will make sure the vertebra is once again in view.

Rotate 2 (rot2)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **rot1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `-90`.

5 Click **Build Selected**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Block 1 (blk1)

In the following section, create a block, combine it with the imported vertebra geometry to create a domain outside of the vertebra, and to cut the geometry in half.

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `23[mm]`.

4 In the **Depth** text field, type `15[mm]`.

5 In the **Height** text field, type `15[mm]`.

6 Locate the **Position** section. In the **x** text field, type `-22[mm]`.

7 In the **y** text field, type `-11.1[mm]`.

8 In the **z** text field, type `60[mm]`.

9 Click **Build Selected**.

Partition Objects 1 (par1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Objects**.

2 Select the object **blk1** only (the block).

3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.

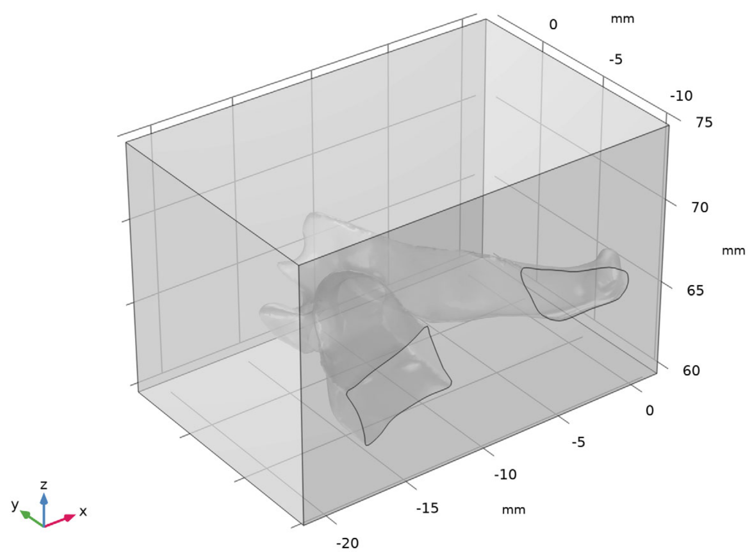
4 Find the **Tool objects** subsection. Click to select the **Activate Selection** toggle button.

5 Select the object **rot2** only (the vertebra).

6 Click **Build Selected**.

7 Click the **Transparency** button in the **Graphics** toolbar.

8 Click the **Go to Default View** button in the **Graphics** toolbar.



The resulting object now contains two domains, inside and outside the imported geometry. Using a transparent view can help to see the parts inside the block. Continue with finalizing the geometry and creating a mesh.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

This builds a tetrahedral mesh on the domains. If this mesh is good enough for the application at hand, it is possible to start setting up the physics at this point. However, many geometries created from mesh require more fine-tuned mesh settings. For example, a specific mesh size setting may cause problems due to the representation of the created face. Then, it often helps to premesh the face with a triangular mesh, restricting the size of the triangles to a more narrow interval. This results in triangles of more similar size and more control over the generated surface mesh. Follow the steps below to generate such a mesh by editing the physics-induced mesh.

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Size** and choose **Build Selected**.

Building a feature node in the sequence of operations will make sure that any further operation that is added will be placed directly after the built one.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundary 6 only (the large face of the vertebra).

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** check box. In the associated text field, type `0.7[mm]`.

6 Select the **Minimum element size** check box. In the associated text field, type `0.5[mm]`.

This will lower the maximum element size the triangles are allowed to take down to 0.7 mm. In the same way, it will increase the minimum element size allowed to 0.5 mm. The rest of the settings are still taken from the settings of the first **Size** node in the sequence.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Free Tetrahedral 1** and choose **Build All** to once again build the mesh on the domains.

Hide some boundaries and turn off transparency to better see the mesh on the boundaries of the vertebra.

2 Click the **Transparency** button in the **Graphics** toolbar.

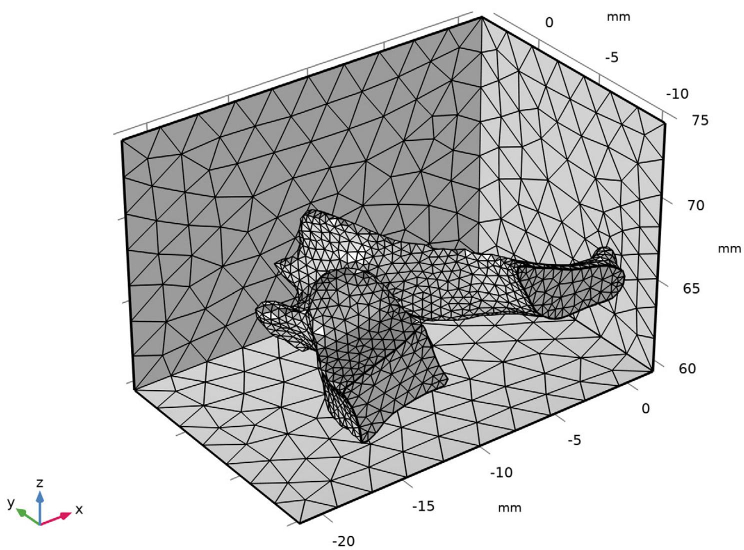
3 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

4 Click to clear the **Activate Selection** toggle button.

5 In the **Graphics** window toolbar, click ▾ next to **Select Domains**, then choose **Select Boundaries**.

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

7 In the **Graphics**, click to hide Boundaries 1, 2, and 4, to get the view in the image below.



The mesh is now ready for simulation.

COMPARING THE MESHED GEOMETRY WITH THE STL MESH

In order to see how close the meshed geometry follows the source STL mesh, create a plot with both meshes. In this final part of the tutorial you will do this in a new component in the model, and also test the effect of the **Simplification tolerance** when generating the geometry.

MESH PART 1

1. In the **Model Builder** window, under **Global Definitions>Mesh Parts** click **Mesh Part 1**.
2. Right-click **Global Definitions>Mesh Parts>Mesh Part 1** and choose **Create Geometry** to reimport the vertebra into a new Component.

MESH 2

Next, generate a triangular mesh for the vertebra. As the purpose of this part of the tutorial is to check the shape of the face generated by the import, there is no need for a tetrahedral mesh.

Free Triangular 1

1. In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.
2. Select Boundary 1 only (the boundary of the vertebra).
3. In the **Settings** window for **Free Triangular**, click **Build Selected**.

Refine 1

Add a **Refine** operation to further refine the mesh and make sure any curvature is resolved.

1. In the **Mesh** toolbar, click **Modify** and choose **Refine**.
2. In the **Settings** window for **Refine**, locate the **Refine Options** section.
3. From the **Refinement method** list, choose **Regular refinement**.
4. From the **Number of refinements** list, choose **2**.
5. Click **Build Selected**.
6. In the **Mesh** toolbar, click **Plot** to automatically create a **Mesh** dataset and add a **Mesh Plot** under **Results**.

RESULTS

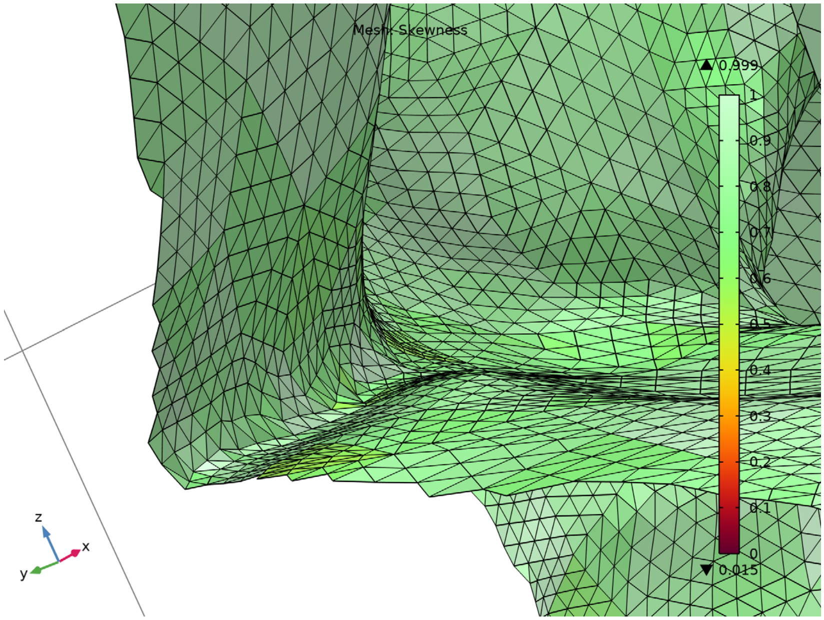
Mesh 2

Use the **Element Filter** option for the **Mesh Plot** to plot only half of the mesh. This will make it easier to compare the two meshes.

1. In the **Settings** window for **Mesh**, click to expand the **Element Filter** section.
2. Select the **Enable filter** check box.
3. In the **Expression** text field, type `x>11[mm]`.

   This means that only mesh elements that are located at x-coordinates larger than 11 mm will be included in the plot.

4 In the **Mesh Plot 1** toolbar, click **Plot**.



The generated geometry face is smooth and has a **Geometric Shape Order** corresponding to the discretization of the physics in the model. In practice, this means that the **Geometric Shape Order** is most often quadratic, unless solving a CFD problem, where linear discretization is used. To plot the curved elements on the boundary, change the corresponding setting on the dataset.

Mesh 2

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Mesh 2**.

2 In the **Settings** window for **Mesh**, locate the **Mesh** section.

3 From the **Geometry shape function** list, choose **Quadratic Lagrange**.

To be able to also plot the source STL mesh in the same plot, duplicate the dataset and change the source of the duplicate to the mesh part.

Mesh 3

1 In the **Model Builder** window, right-click **Mesh 2** and choose **Duplicate**.

2 In the **Settings** window for **Mesh**, locate the **Mesh** section.

3 From the **Mesh** list, choose **Mesh Part 1**.

The STL format only supports linear elements, however once imported, a curved representation of the faces is set up if higher order discretization is used in the physics. Therefore, keep the setting **Geometry Shape Order: Quadratic Lagrange**.

Now that the STL mesh has a dataset, duplicate the **Mesh 2** plot feature and modify the settings to point to the newly created dataset.

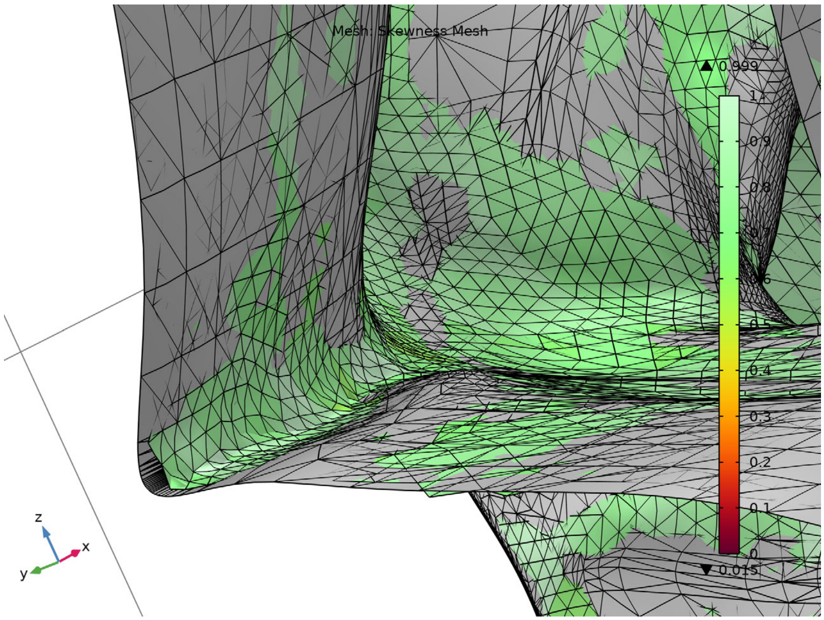
Mesh 3

1. In the **Model Builder** window, under **Results>Mesh Plot 1** right-click **Mesh 2** and choose **Duplicate**.
2. In the **Settings** window for **Mesh**, locate the **Data** section.
3. From the **Dataset** list, choose **Mesh 3**.
4. Locate the **Coloring and Style** section. From the **Element color** list, choose **Gray**.

By giving the source STL mesh a gray color, it will be easier to distinguish between the two meshes. The previously plotted mesh uses the default coloring showing the triangles' quality with a scale from red (indicating poor quality) to green (good quality).

5 In the **Mesh Plot 1** toolbar, click **Plot**.

Rotate and zoom in on one of the almost 90 degree corners in the mesh, located in the bottom-left corner of the image below.



It is clear that the mesh of the generated geometry (green) does not follow the source mesh (gray) closely in the region of the 90 degree corner. In regions with lesser curvature the green and gray mesh elements are more intermixed. This indicates better conformance of the generated mesh to the source mesh in those regions.

Now, go back to the **Import** node and increase the **Relative simplification tolerance**, to see the effect this has on the generated geometry.

GEOMETRY 2

Import 1 (imp1)

1 In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** click **Import 1 (imp1)**.

2 In the **Settings** window for **Import**, locate the **Import** section.

3 In the **Relative simplification tolerance** text field, type `0.02`.

4 Click **Build All Objects**.

MESH 2

Refine 1

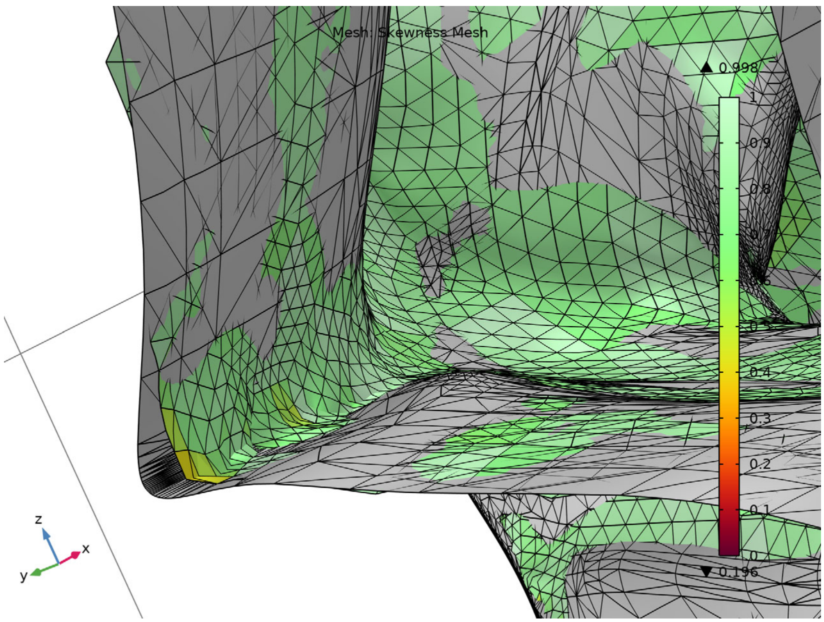
1 In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Refine 1**.

2 In the **Settings** window for **Refine**, click **Build All**.

RESULTS

Mesh 3

Click somewhere in the **Mesh Plot 1** to automatically update the plot.



Looking at the same corner region, the mesh of the generated geometry (green) shows a more rough representation of the source mesh (grey). This is expected as increasing the **Relative simplification factor** allows for a larger deviation from the source mesh when the surfaces of the geometry are generated during the import.

This time, go back and decrease the **Relative simplification tolerance** to 0.001. This is 10 times smaller than the default value of 0.01.

GEOMETRY 2

Import 1 (imp1)

1. In the **Model Builder** window, under **Component 2 (comp2)>Geometry 2** click **Import 1 (imp1)**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. In the **Relative simplification tolerance** text field, type `0.001`.
4. Click **Build All Objects**.

MESH 2

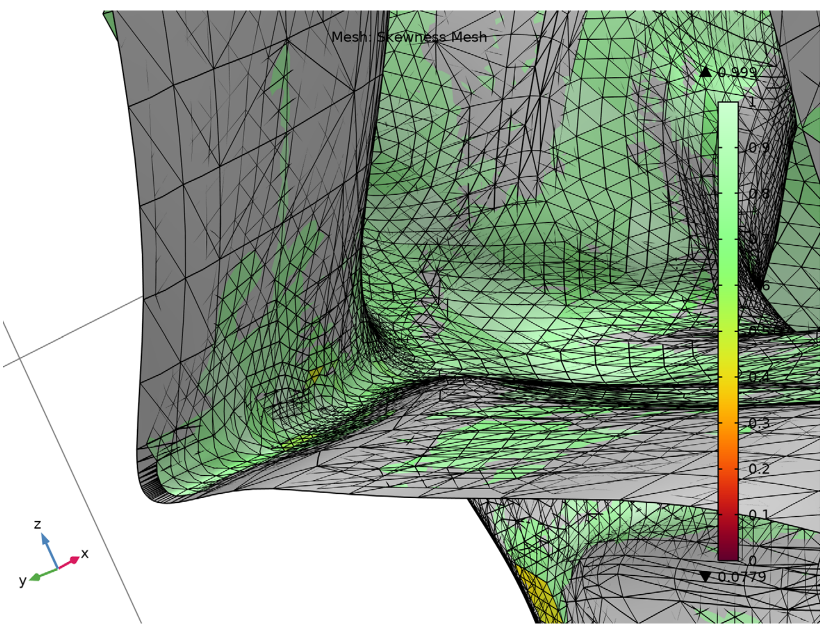
Refine 1

1. In the **Model Builder** window, under **Component 2 (comp2)>Mesh 2** click **Refine 1**.
2. In the **Settings** window for **Refine**, click **Build All**.

RESULTS

Mesh 3

Click somewhere in the **Mesh Plot 1** to automatically update the plot.



The mesh of the generated geometry (green) now follows the source mesh (gray) much closer. This is the effect of the smaller **Relative simplification factor**, which allows for less deviation from the source mesh when the surface is generated during import.