From Surface Mesh to Geometry: STL Import of a Vertebra[1]

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

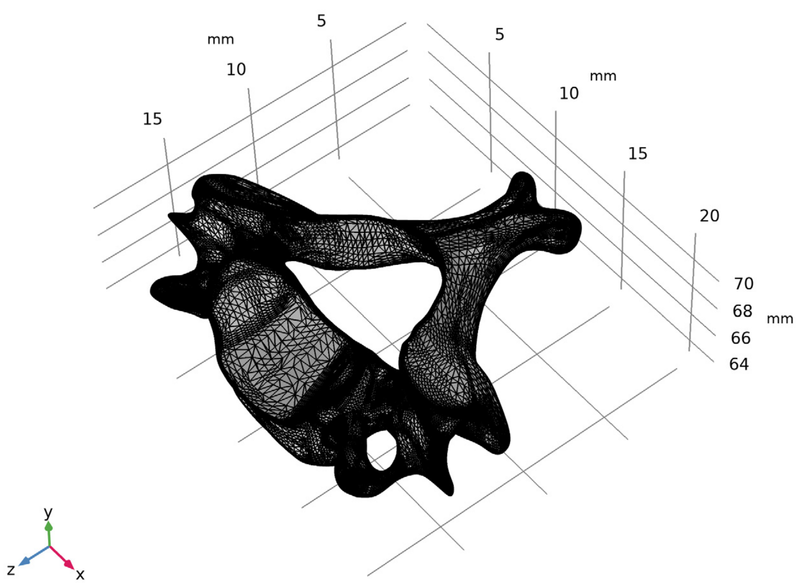
Introduction

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.

COMSOL Multiphysics supports the import of surface meshes from STL files. Using available tools you can partition the imported mesh and generate a solid geometry object. During the mesh-to-geometry conversion process a simplification algorithm helps with smoothing out defects in the imported mesh. You can then further modify the generated geometry using geometric operations.

Model Definition

Import the STL file of a vertebra geometry shown below.



Follow the instructions in this tutorial to

- Import the STL file and create a solid geometry object.
- 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.

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**.

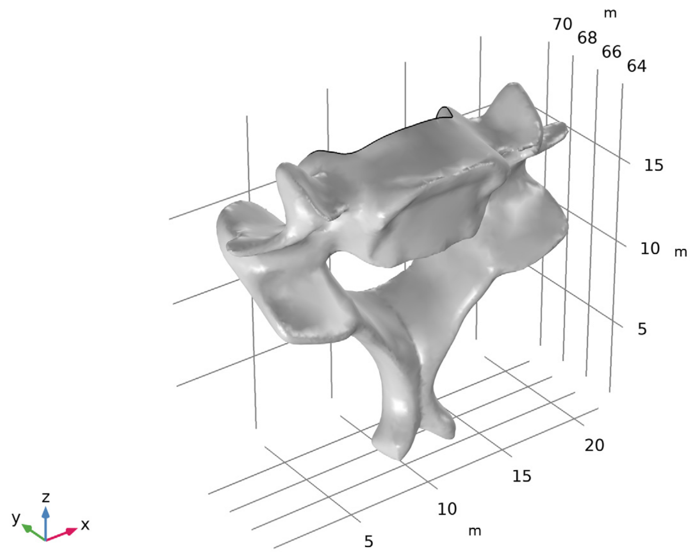
GEOMETRY 1

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. In the **Defect removal factor** text field, type `1.5`.

   In this case, we increase the **Defect removal factor** to remove more triangulation artifacts.
6. 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. According to the information in the **Messages** window a solid object could be created from the mesh.



Notice that after the import completes you can no longer see the reference to the STL file 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 **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.

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

MESH PART 1: LENGTH UNIT

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 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, under **Global Definitions>Mesh Parts** 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**.

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**.
4. In the **Home** toolbar, click **Build All**. This will update the length unit of the **Geometry 1** sequence, and re-import the geometry from the mesh.

   Now 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.
3. In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.
4. In the **Rotation** text field, type `90`.
5. Locate the **Axis of Rotation** section. From the **Axis type** list, choose **x-axis**.
6. Click **Build Selected**.
7. Click the **Zoom Extents** button in the **Graphics** toolbar.

Rotate 2 (rot2)

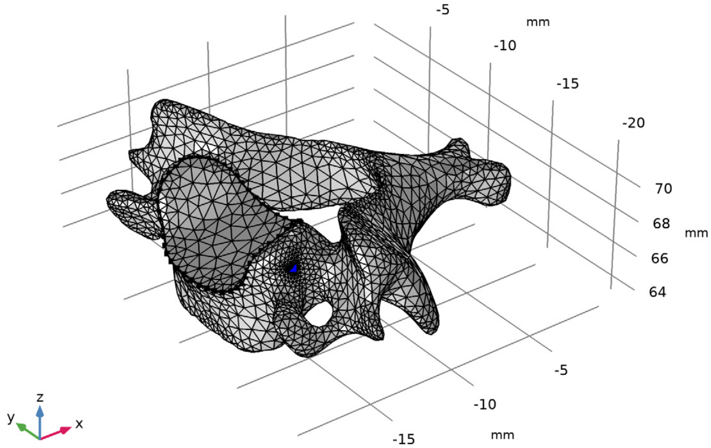
1. In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.
2. Select the object **rot1** only.
3. In the **Settings** window for **Rotate**, locate the **Rotation Angle** section.
4. In the **Rotation** text field, type `-90`.
5. Click **Build Selected**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

MESH 1

A first step to assess the quality of an imported geometry is to create a mesh. Problem regions, for example small faces or short edges, are detected and reported by the mesher.

Warning 1

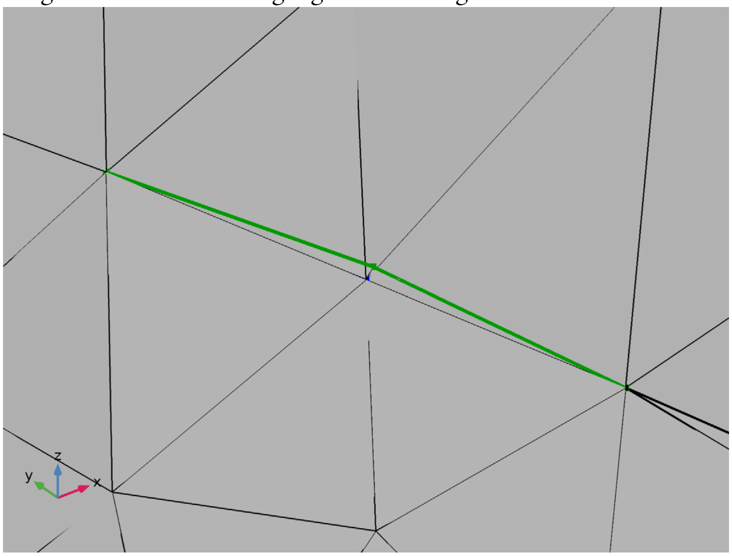
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



Two short edges, which are listed in the **Warning 1** node that appears, have been detected.

2 In the list, select **4**.

3 Click the **Zoom to Selection** button in the **Graphics** toolbar. If the short edge is 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 two edges are located at the boundary of a small face that forms a fold in the geometry. To find where the two short edges originate from switch to the **Mesh Part 1** sequence.

MESH PART 1

For a better overview of all boundaries on the imported mesh open the **Selection List** window.

1 In the **Model Builder** window, under **Global Definitions>Mesh Parts** click **Mesh Part 1**.

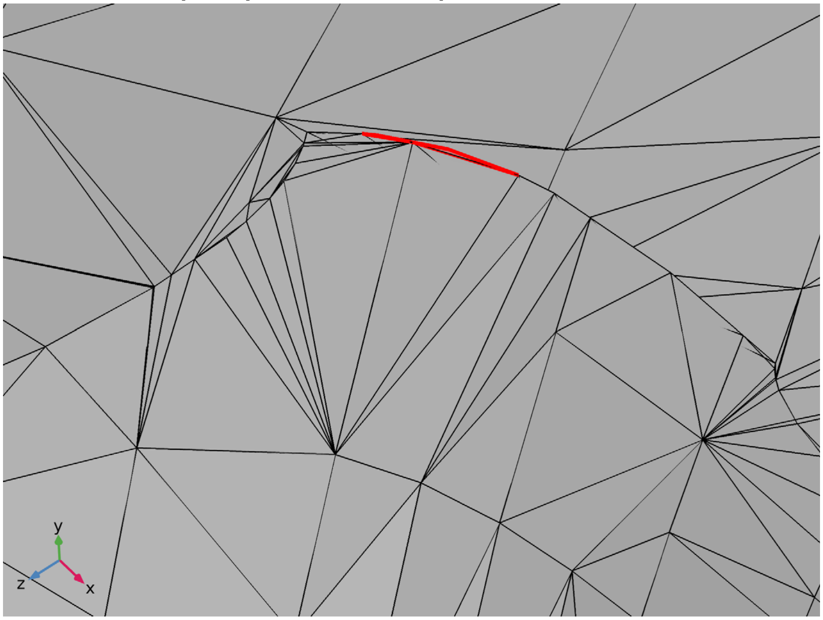
2 In the **Home** toolbar, click **Windows** and choose **Selection List**.

3 Click the **Select Boundaries** button in the **Graphics** toolbar.

4 In the **Settings** window for **Selection List**, in the list, choose **3 (meshed)** and **4 (meshed)**. Hold down **Ctrl** while clicking in the list to select multiple boundaries.

5 Click the **Zoom to Selection** button in the **Graphics** toolbar. If needed, zoom out and rotate the geometry before zooming in on the boundary again.

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



On the imported mesh Boundaries 4 and 5 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. Isolated boundaries such as these are automatically removed during the last step of the geometry creation process, when the solid object is created.

In the following we will delete these boundaries from the imported mesh. First, we join all boundaries to get rid of Boundary 3 that is responsible for the small fold in the final geometry.

7 Switch to the **Model Builder** window.

Join Entities 1

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

This adds the **Joined Entities 1** operation to the **Mesh Parts 1** sequence.

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**.

According to the **Warning 1** node displayed under **Join Entities 1** not all selected boundaries could be joined. This is expected since the two isolated boundaries cannot be joined. Instead we will delete these.

Delete Entities 1

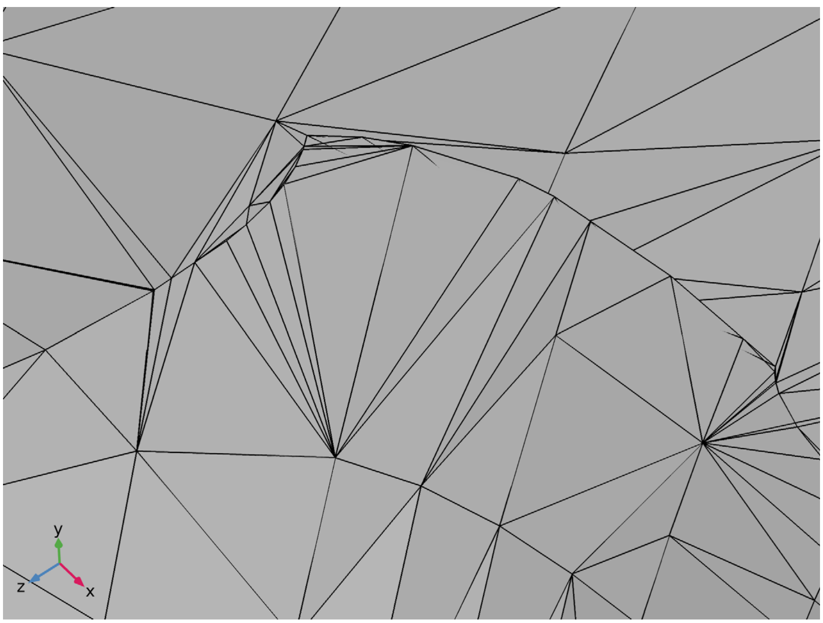
1 In the **Mesh** toolbar, click **Delete Entities**.

2 In the **Settings** window for **Selection List**, in the list, choose **2 (meshed)** and **3 (meshed)**.

3 Click **Add to Selection** in the window toolbar.

4 Right-click **Delete Entities 1** and choose **Build Selected**.

After the operation completes the **Selection List** window lists only one boundary for the mesh. The isolated boundaries are no longer there, but the small fold in the mesh is still visible.



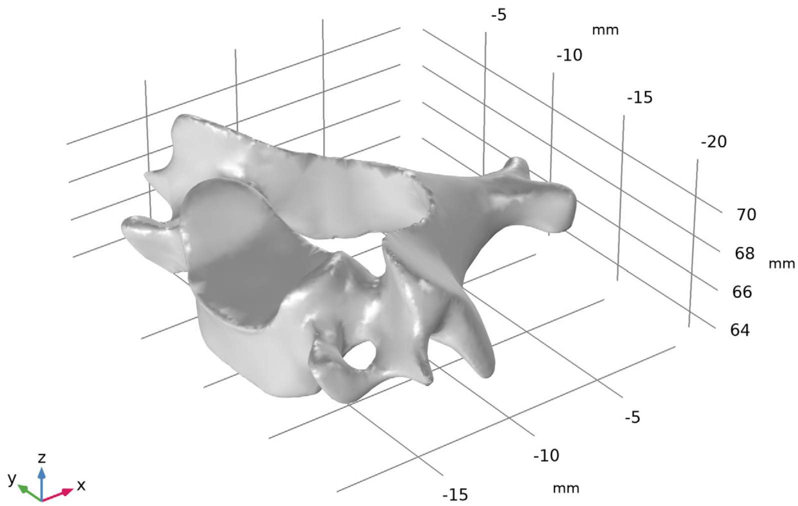
5 Close the **Selection List** window.

GEOMETRY 1

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

2 In the **Home** toolbar, click **Build All**.

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

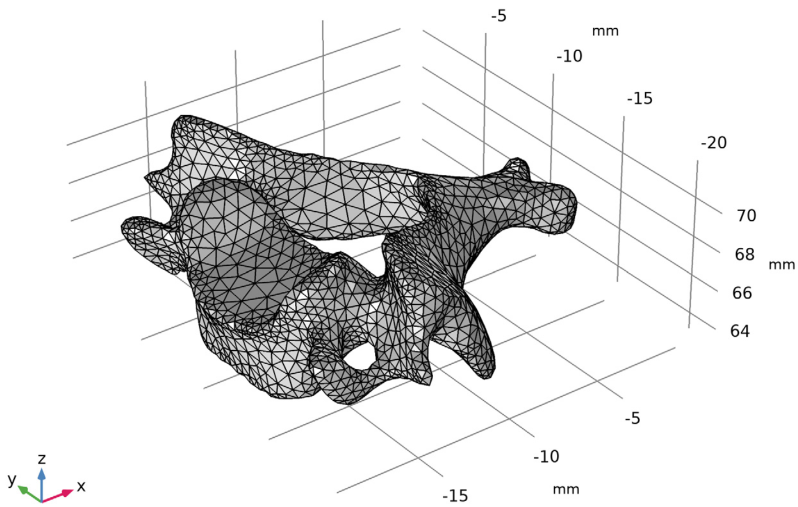


4 Zoom in on the region highlighted in the figure above.

The short edges are no longer present in the geometry. The small fold in the surface has also disappeared because this time it is part of a larger surface, and therefore it could be removed by the mesh simplification algorithm that is part of the conversion to geometry.

MESH 1

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



This time the mesh builds without any warnings.

GEOMETRY 1

Import 1 (imp1)

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

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

3 In the **Relative simplification tolerance** text field, type `0.008`.

By setting a tighter tolerance we are reducing how much the imported mesh can be modified by the mesh simplification algorithm. This tolerance is relative to the entire geometry and together with the **Defect removal factor**, which is relative to the local feature size, controls how much the imported mesh can be modified at a certain location before converting it to geometry. Decreasing the relative simplification tolerance should work in this case, since apart from some small local defects the imported surface mesh seems quite good.

4 Click **Build All Objects**.

With the smaller tolerance the processing time increases, but the generated solid object follows the imported mesh more accurately.

Block 1 (blk1)

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

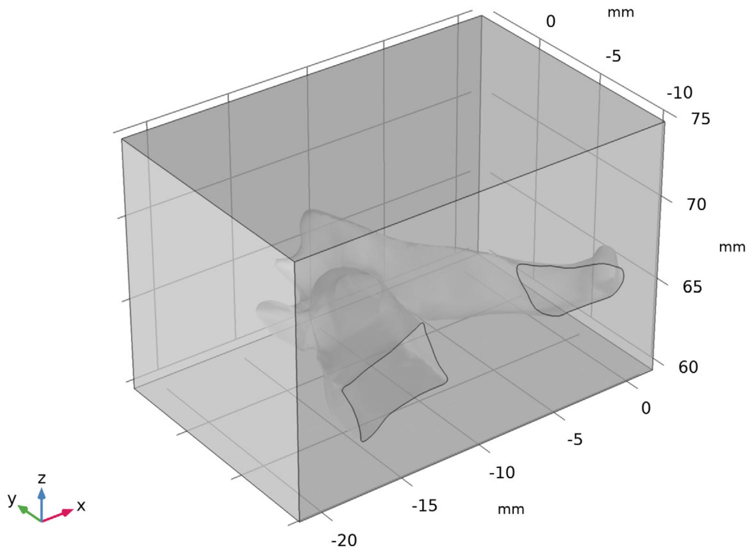
1 In the **Geometry** toolbar, click **Block**.
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.
3 In the **Width** text field, type `23[mm]`.
4 In the **Depth** text field, type `15[mm]`.
5 In the **Height** text field, type `15[mm]`.
6 Locate the **Position** section. In the **x** text field, type `-22[mm]`.
7 In the **y** text field, type `-11[mm]`.
8 In the **z** text field, type `60[mm]`.
9 Right-click **Block 1 (blk1)** and choose **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. Select the **Active** toggle button.
5 Select the object **rot2** only (the vertebra).
6 From the **Repair tolerance** list, choose **Relative**.
7 In the **Relative repair tolerance** text field, type `1.0E-4`.

   Use a larger tolerance to ignore small defects in the geometry representation.
8 Click **Build Selected**.
9 Click the **Transparency** button on the **Graphics** toolbar.

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



The resulting object now contains two domains, inside and outside the imported geometry. As a final step create a mesh for the geometry.

MESH 1

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