STL Import 2 — Combining Geometry with an Imported Mesh[1]

---

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

Introduction

When working with imported STL meshes, you may run into applications where you want to combine your imported mesh with an imported CAD part, CAD assembly, or with geometry created in COMSOL Multiphysics.

This tutorial series consists of two parts, where the first part, *STL Import 1 — Repairing and Combining STL Files*, focuses on using available tools to edit imported STL meshes: the different operations for repairing the meshes, combining imported meshes with each other, and generating a volume mesh from the imported surface mesh. This second part of the series describes a workflow for combining the mesh obtained in the first part with a parameterized geometry.
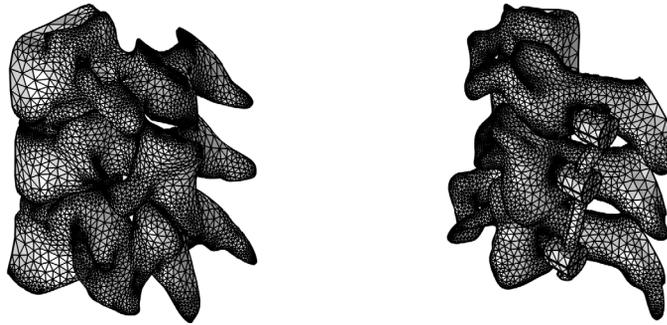

Figure 1: Left: Tetrahedral mesh after repairing, combining, and partitioning imported surface meshes in the first part of the tutorial series. Right: Tetrahedral mesh after combining the mesh from the first part with parameterized geometry created in the second part of the tutorial series.

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 can also be called a surface mesh, of a 3D object. The triangles in the file are identified by their normals and vertex coordinates and are stitched together using a tolerance to form a faceted representation of the object.

COMSOL Multiphysics supports a variety of operations, for example, moving, scaling and rotating an imported mesh, combining the imported mesh with parameterized geometry to run parametric sweeps, and generating unstructured, structured, and boundary layer mesh in the domains. When creating geometry or importing CAD files for combining with an imported mesh, you can view the imported mesh with the geometry to help with positioning the design. This procedure is shown in the instructions of this tutorial.

Lastly, it is important to mention that the workflows presented in this 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

The goal of this tutorial is to combine the mesh of the vertebrae and discs, imported in the first part of this tutorial series, with the geometry of the three pedicle screws and a rod. The screws are created as parameterized geometry parts with the possibility of varying the screw angles and entry points.
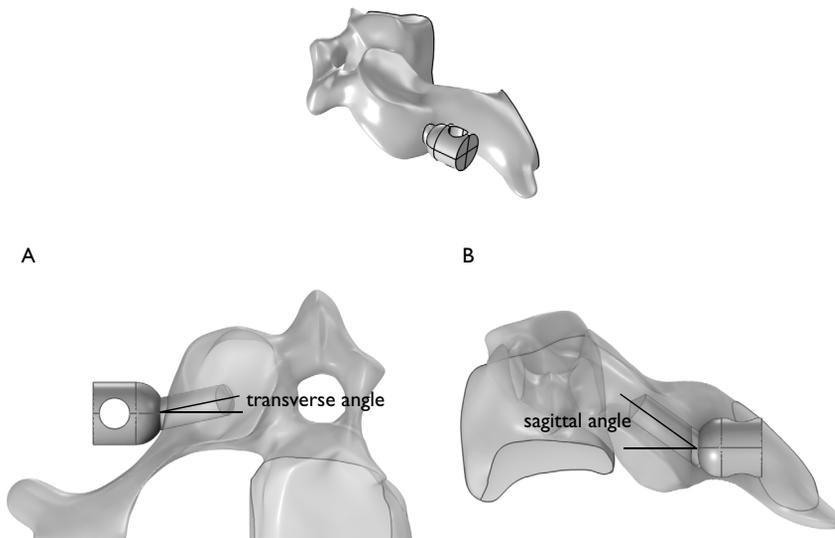

Figure 2: The transverse angle (left) and sagittal angle (right) of the polyaxial screws.

The workflow for the model includes the following steps:

- Setting up the parameters for the geometry
- Importing the mesh of the vertebrae and discs into the geometry sequence as a construction object to help with the positioning of the screws

- Inserting the geometry parts for the screws into the geometry sequence and positioning them in relation to the imported mesh of the vertebrae and discs
- Creating the geometry for the rod that connects the screws by leveraging measurement features for the correct positioning
- Importing the geometry of the screws and rod into the meshing sequence and combining it with the mesh of the vertebrae and discs
- Building a physics-controlled mesh
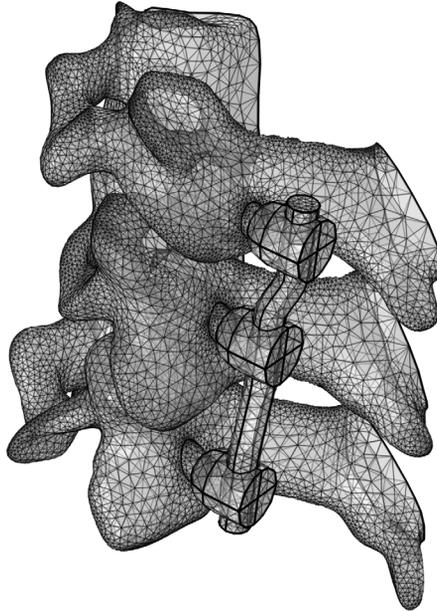
Application Library path: COMSOL_Multiphysics/Meshing_Tutorials/
stl_2_combine_geom_mesh

Modeling Instructions

COMSOL DESKTOP

1 In the **Model Builder** window, click the root node.

2 From the **File** menu, choose **Open**.

3 Browse to the model's Application Libraries folder and double-click the file
`stl_1_repair_imported_meshes.mph`.

This file contains the prepared mesh of the vertebrae and discs. For the step-by-step instructions to import, repair and combine the STL meshes, see the tutorial *STL Import 1 — Repairing and Combining STL Files*.

GLOBAL DEFINITIONS

Under Global Definitions, import the parameters and load the geometry part of a polyaxial screw.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click ⬛ **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file
`stl_2_combine_geom_mesh_parameters.txt`.

POLYAXIAL PEDICLE SCREW

1 In the **Geometry** toolbar, click **Load Part** and choose **Load Part**.

2 Browse to the model's Application Libraries folder and double-click the file
`polyaxial_screw_geom_sequence.mph`.

SELECTION LIST

1 In the **Geometry** toolbar, click ⬛ **Selection List** to open the **Selection List** window.

Keep the **Selection List** open to check the created and imported domains and selections throughout the tutorial.

GEOMETRY 1

Import 1 (imp1)

You will now continue with importing the mesh of the vertebrae and discs as a construction geometry that you can use as an aid when positioning the screws.

1 In the **Geometry** toolbar, click ⬛ **Import**.

Notice that according to the default option in the Combine Geometry with Mesh dialog, an Import node will also be added to the Mesh 1 sequence.

2 In the **Combine Geometry with Mesh** dialog, click **OK**.

3 In the **Settings** window for **Import**, locate the **Source** section.

4 From the **Source** list, choose **Mesh or 3D printing file (STL, 3MF, PLY)**.

5 From the **Mesh** list, choose **Combined Mesh**.

6 Locate the **Simplify and Repair** section. In the **Relative simplification tolerance** text field, type `0.001`.

7 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

8 Click **Build Selected**.

COMPONENT 1 (COMP1)

Inspect the Import node in Mesh 1 now before you continue with the geometry. This Import node was added to prepare for combining the geometry and mesh once you are done creating the geometry. Physics interfaces that you add to this model will use the domains and boundaries defined by **Mesh 1**, not the geometry under **Geometry 1**. In the **Model Builder**, this is indicated by the green rectangle around the icon for **Mesh 1**

In the **Model Builder** window, expand the **Component 1 (comp1) > Meshes** node.

MESH 1

Import - Geometry

1 In the **Model Builder** window, expand the **Component 1 (comp1) > Meshes > Mesh 1** node, then click **Import 2**.

2 In the **Settings** window for **Import**, type `Import - Geometry` in the **Label** text field.

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

4 In the **Settings** window for **Mesh**, type `Combined Vertebrae and Screws` in the **Label** text field.

GEOMETRY 1

Before adding an instance of the screw, create a work plane along which to align the part. The work plane also sets the entry point of the screw.

The angles and entry point coordinates are parameterized in this tutorial so that it is easy to set up a parametric study to compare different screw fixation techniques. The screw dimensions can of course also be parameterized, but in this tutorial, you will only vary the length of the screw thread for the different vertebrae and keep the other parameters fixed.

C3 Pedicle Screw

Add node groups for the two screws and the rod to organize the feature nodes.

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Node Group**.

2 In the **Settings** window for **Group**, type C3 Pedicle Screw in the **Label** text field.

C3 screw position

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, type C3 screw position in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type c3_zw.

5 Click to expand the **Local Coordinate System** section. In the **xw-displacement** text field, type -c3_xw.

6 In the **yw-displacement** text field, type -c3_yw.

Polyaxial Pedicle Screw 1 (pi1)

1 In the **Geometry** toolbar, click **Part Instance** and choose **Polyaxial Pedicle Screw** to add an instance of the screw that you will align with the newly created work plane.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
thread_l	c3_L	11 mm	Screw thread length
sagittal_angle	s_angle	35 °	Sagittal angle
transverse_angle	t_angle	10 °	Transverse angle

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Screw alignment (wp3)**.

5 Find the **Coordinate system to match** subsection. From the **Work plane** list, choose **C3 screw position (wp1)**.

6 Click to expand the **Object Selections** section. In the table, select the **Keep** checkbox for **Rod help points**.

7 Click **New Cumulative Selection** to create a cumulative selection for the help points that you want to delete once the geometry is done.

8 In the **New Cumulative Selection** dialog, type Help Points in the **Name** text field.

9 Click **OK**.

Make sure **Help Points** was added under the **Contribute to** column for the **Rod help points** selection. Otherwise, select it manually.

Similarly, create a cumulative selection for the material domains of the screws.

10 In the **Settings** window for **Part Instance**, click to expand the **Domain Selections** section.

11 In the table, select the **Keep** checkbox for **Screw domain**.

12 Click **New Cumulative Selection**.

13 In the **New Cumulative Selection** dialog, type Screw Domains in the **Name** text field.

14 Click **OK**.

15 In the **Settings** window for **Part Instance**, click 🔲 **Build Selected**.

Make sure **Screw Domains** was added under the **Contribute to** column for the **Screw domain** selection. Otherwise, select it manually.

16 Click to expand the **Boundary Selections** section. In the table, enter the following settings:

| Name	Keep	Physics	Contribute to
Thread boundaries (Thread domain)	√		None

17 Click **New Cumulative Selection**.

18 In the **New Cumulative Selection** dialog, type Simplify threads in the **Name** text field.

19 Click **OK**.

1 In the **Model Builder** window, click **Polyaxial Pedicle Screw 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, click ⬛ **Build Selected** and rotate the geometry to verify that it looks like the image below.
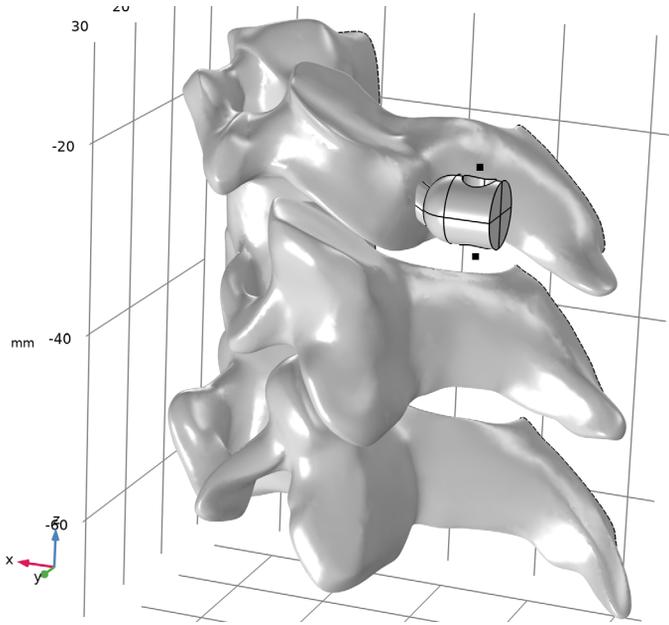
C4 Pedicle Screw

Repeat the procedure for the C4 vertebra screw.

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Node Group**.

2 In the **Settings** window for **Group**, type C4 Pedicle Screw in the **Label** text field.

C4 screw position

1 In the **Geometry** toolbar, click 🛢 **Work Plane**.

2 In the **Settings** window for **Work Plane**, type C4 screw position in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type c4_zw.

5 Click to expand the **Local Coordinate System** section. In the **xw-displacement** text field, type -c4_xw.

6 In the **yw-displacement** text field, type -c4_yw.

Polyaxial Pedicle Screw 2 (pi2)

1 In the **Geometry** toolbar, click ⤷△ **Part Instance** and choose **Polyaxial Pedicle Screw**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
thread_l	c4_L	12 mm	Screw thread length
sagittal_angle	s_angle	35 °	Sagittal angle
transverse_angle	t_angle	10 °	Transverse angle

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Screw alignment (wp3)**.

5 Find the **Coordinate system to match** subsection. From the **Work plane** list, choose **C4 screw position (wp2)**.

6 Locate the **Object Selections** section. In the table, enter the following settings:

Name	Keep	Contribute to
Rod help points	√	Help Points

7 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Screw domain	√	√	Screw Domains

8 Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Thread boundaries (Thread domain)	√		Simplify threads

9 Click ▦ **Build Selected**.

C5 Pedicle Screw

Repeat the procedure for the C5 vertebra screw.

1 Right-click **Geometry 1** and choose **Node Group**.

2 In the **Settings** window for **Group**, type C5 Pedicle Screw in the **Label** text field.

C5 screw position

1 In the **Geometry** toolbar, click ⬢ **Work Plane**.

2 In the **Settings** window for **Work Plane**, type C5 screw position in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Plane** list, choose **zx-plane**.

4 In the **y-coordinate** text field, type `c5_zw`.

5 Click to expand the **Local Coordinate System** section. In the **xw-displacement** text field, type `-c5_xw`.

6 In the **yw-displacement** text field, type `-c5_yw`.

Polyaxial Pedicle Screw 3 (pi3)

1 In the **Geometry** toolbar, click 🔺 **Part Instance** and choose **Polyaxial Pedicle Screw**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
thread_l	c5_L	12 mm	Screw thread length
sagittal_angle	s_angle	35 °	Sagittal angle
transverse_angle	t_angle	10 °	Transverse angle

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Screw alignment (wp3)**.

5 Find the **Coordinate system to match** subsection. From the **Work plane** list, choose **C5 screw position (wp3)**.

6 Locate the **Object Selections** section. In the table, enter the following settings:

Name	Keep	Contribute to
Rod help points	√	Help Points

7 Locate the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Screw domain	√	√	Screw Domains

8 Locate the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Thread boundaries (Thread domain)	√		Simplify threads

9 Click ![icon] **Build Selected**.

Check the bottom section of the **Selection List** to see that there are now selections created that you can use in geometry operations and some of them also when setting up the physics.

Rod

To create the geometry for the connecting rod, sweep a circular face along a polygon that goes through the four help points of the screw heads.

1 Right-click **Geometry 1** and choose **Node Group**.

2 In the **Settings** window for **Group**, type Rod in the **Label** text field.

Rod cross section

1 In the **Geometry** toolbar, click ![icon] **Work Plane**.

2 In the **Settings** window for **Work Plane**, type Rod cross section in the **Label** text field.

3 Locate the **Plane Definition** section. From the **Offset type** list, choose **Through vertex**.

4 On the object **pi1(2)**, select Point 1 only by clicking on the topmost vertex of the screw head help points or by selecting and adding it from the **Selection List**.

5 Locate the **Local Coordinate System** section. From the **Origin** list, choose **Vertex projection**.

6 On the object **pi1(2)**, select Point 1 only.

7 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

8 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox to make the rod cross section a construction object that is automatically deleted when finalizing the geometry.

9 Click ![icon] **Build Selected**.

10 Click ![icon] **Go to Plane Geometry**.

11 In the **Sketch** toolbar, click **Circle**.

12 In the **Graphics** window, place the center in the origin of the work plane, then move the mouse cursor, and click once more in the **Graphics** to draw the circle.

Rod cross section (wp4) > Circle 1 (c1)

1 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

2 In the **Radius** text field, type d_rod/2.

3 Click ![icon] **Build Selected**.

Rod

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

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

Centroid Measurement

1 In the **Graphics** window, click to select the topmost of the six help points for the screw heads.

2 Right-click in the **Graphics** window and select **Centroid Measurement** from the Graphics context menu.

In the **Settings** window, you can change the name of the generated coordinate parameters to something more descriptive, if you like, but for this tutorial, keep the default names.

3 Repeat steps **1** and **2** for the other five help points for the screw heads.

Polygon 1 (pol1)

Next, draw a polygon through the help points of the screw heads.

1 In the **Geometry** toolbar, click ⬭ **More Primitives** and choose **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 Click the **Load** button. From the menu, choose **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file stl_2_combine_geom_mesh_rod_coord.txt.

The naming and numbering assumes that you added the Centroid Measurement features for the points top-down and kept their original names. If you changed the names, you need to insert the parameter names manually. To do so, in the table for the polygon, paste in, for example, geom1.cm and then use Ctrl+space to easily select and fill in the table.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

6 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

7 Click ▊ **Build Selected**.

Rod domain

1 In the **Geometry** toolbar, click ⬚ **Sweep**.

2 In the **Settings** window for **Sweep**, type Rod domain in the **Label** text field.

3 Locate the **Cross Section** section. From the **Entities to sweep** list, choose **Rod cross section**.

4 Clear the **Create cross-sectional faces** checkbox.

5 Locate the **Spine Curve** section. From the **Edges to follow** list, choose **Polygon 1**.

6 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

7 Locate the **Assigned Attributes** section. From the **Construction geometry** list, choose **Off**.

8 Click 🔲 **Build Selected**.
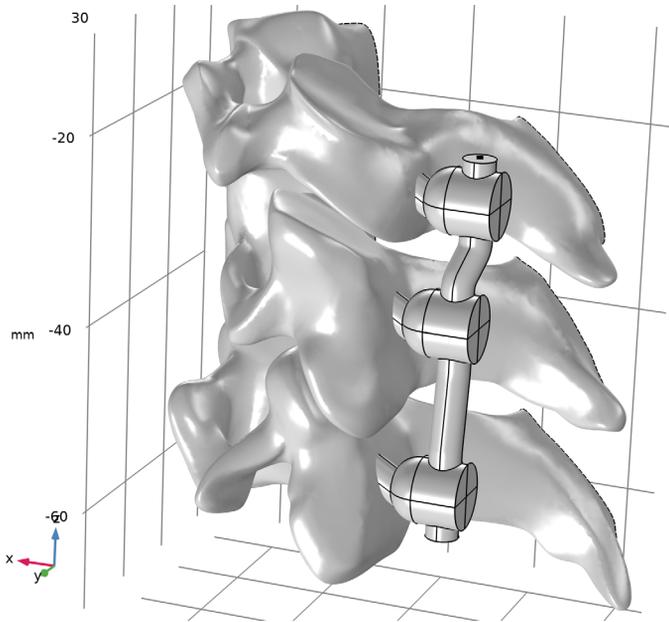
Delete Entities 1 (del1)

The geometry is almost finished, just some cleanup left to do. First, delete the help points as they are no longer needed.

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Object**.

4 From the **Selection** list, choose **Help Points**.

5 Click 🔲 **Build Selected**.

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**.

This removes all construction geometry and you are left with the screws and rod. Check the **Selection List** that there are now four domains, one for each screw and one for the rod.

Simplify the boundaries of the screw threads to one single boundary using virtual operations.

Form Composite Faces 1 (cmf1)

1 In the **Geometry** toolbar, click ⬎ **Virtual Operations** and choose **Form Composite Faces**.

2 In the **Settings** window for **Form Composite Faces**, locate the **Input** section.

3 From the **Faces to composite** list, choose **Simplify threads**.

4 Click ▥ **Build Selected**.

COMBINED VERTEBRAE AND SCREWS

The geometry is now completed. Next, import it into the meshing sequence and combine it with the mesh of the vertebrae and disc.

Import - Geometry

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Meshes** > **Combined Vertebrae and Screws** click **Import - Geometry**.

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

3 From the **Size** list, choose **Finer**.

4 Select the **Resolve geometric details** checkbox to use a finer element size to represent the screws and rod.

This step is done to ensure that the intersecting mesh surfaces of the pedicle screws and the vertebrae have a similar element size. This will help with forming the union of the meshes further ahead.

5 Click **Import**.

This imports the geometry of the screws and rod. Make sure that the screws are indeed fully inside the vertebrae by examining the mesh in the **Graphics** window. If not, go back and check that the geometry is set up correctly.
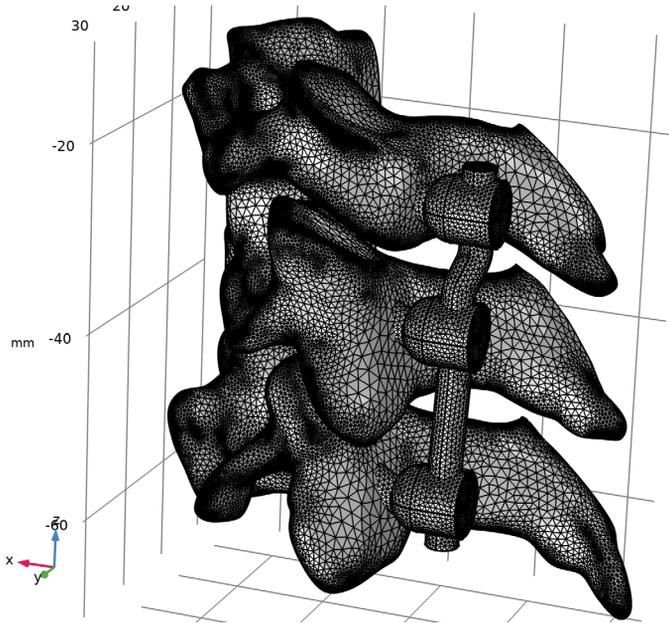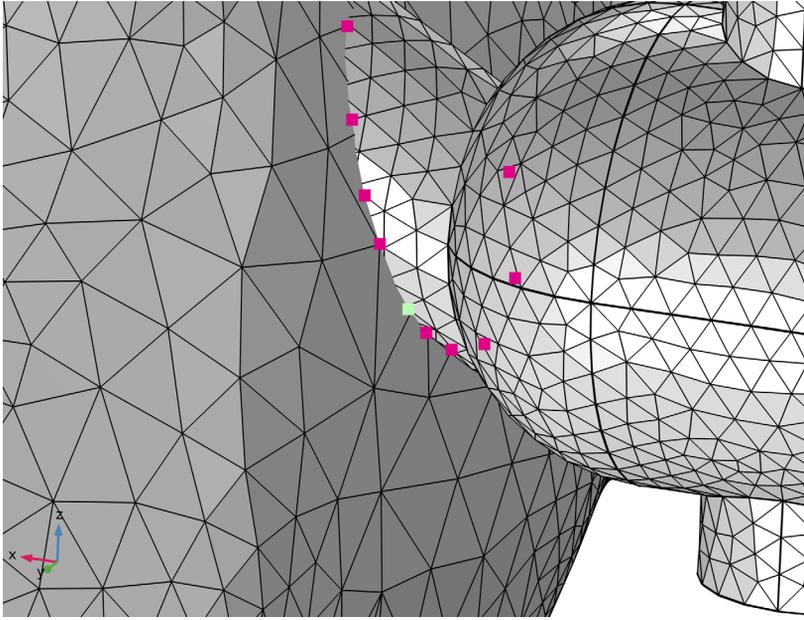

Information

A subnode to the **Import - Pedicle screws and rod** node informs that the meshes are intersecting.

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

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

3 From the **Select location** list, choose **Location 5 (0.4007, 20.37, -55.37)**.

4 Click **Center at Coordinates** to zoom in on a reported problem coordinate to inspect the intersection more closely. If the mesh is more complex, it can help to also clip around the coordinates to remove everything except the mesh closest to the coordinates from the view and you can also rotate the mesh such that you see at least some of the reported

points more clearly. Note that the list of reported points is filtered so that only a smaller number of intersecting elements are listed.
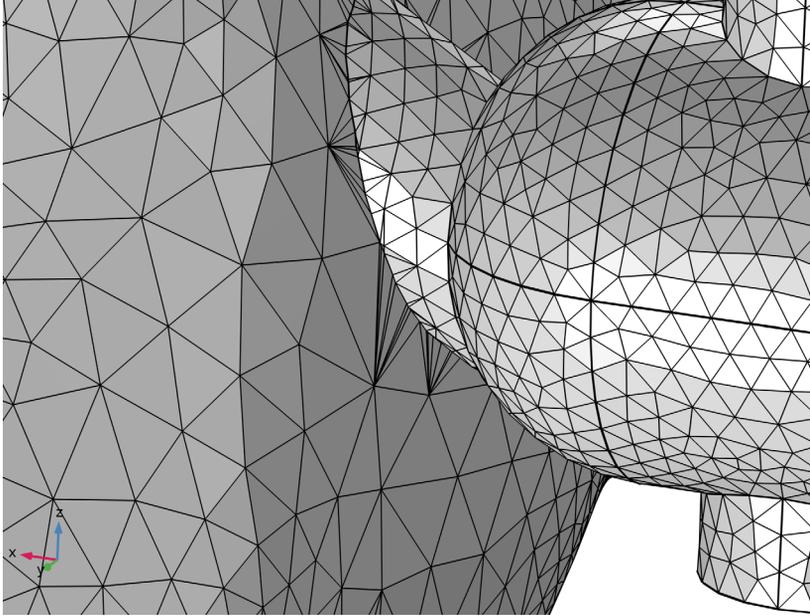

Union 1

As suggested in the Information node, use a mesh Union operation to compute the intersections.

1 In the **Mesh** toolbar, click **Booleans and Partitions** and choose **Union**.

2 In the **Settings** window for **Union**, locate the **Cleanup** section.

3 From the **Placement of mesh vertices** list, choose **Linear**.

**4** Click ⬛ **Build Selected** to unite the entire mesh using the default tolerance.

The union has partitioned the faces and the domains at the intersections. Along the new edges, some sliver triangles have been created. When building the physics-controlled mesh, the quality of the boundary mesh will be improved.

Next, build the physics-controlled mesh.

PHYSICS-CONTROLLED MESH

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Meshes** > **Physics-Controlled Mesh** node.

2 Right-click **Physics-Controlled Mesh** and choose **Build All**.

3 Click the ⬛ **Transparency** button in the **Graphics** toolbar.

4 In the **Model Builder** window, click **Physics-Controlled Mesh**.
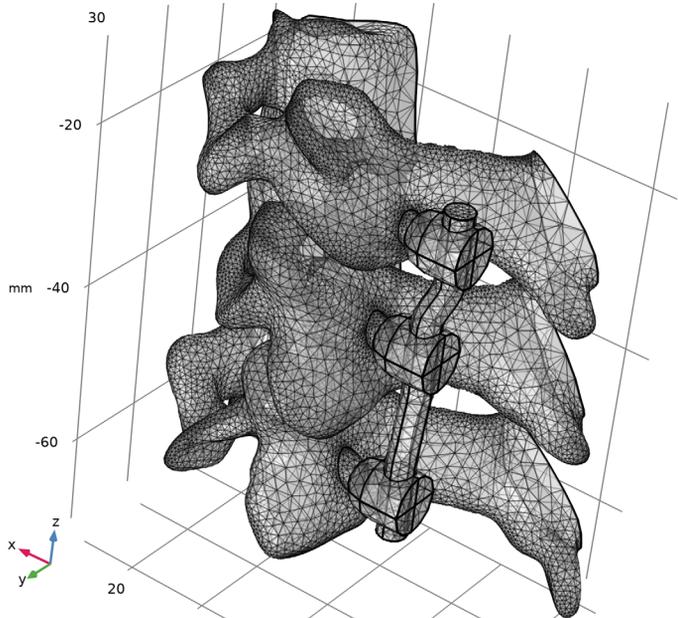
You have now obtained a volume mesh for the combined vertebrae, discs, and pedicle screws.