Created in COMSOL Multiphysics 6.3

STL Import 1 — Repairing and Combining STL Files[1]

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

Introduction

When working with imported STL meshes, there is often a need to repair and edit the mesh, form different computational domains, remesh the faces to improve the element quality, and to fill the domains with a volumetric mesh. In case you have several STL files, you may also need to modify the position of some of the meshes to remove gaps and intersections before combining the meshes.

This tutorial series consists of two parts, where this first part 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. The second part of the series, *STL Import 2 — Combining Geometry with an Imported Mesh*, describes a workflow for combining the mesh obtained in the first part with a parameterized geometry.

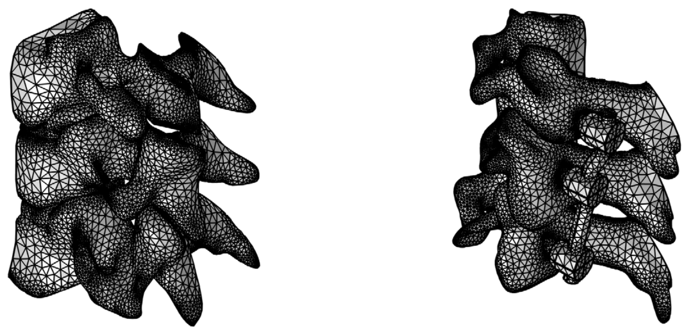


Figure 1: Left: Tetrahedral mesh after repairing, combining and partitioning imported surface meshes in the first part of the tutorial series. Right: Tetrahedral mesh after combining the mesh from the first part with parameterized geometry created in the second part of the tutorial series.

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 second part of this tutorial series, *STL Import 2 — Combining Geometry with an Imported Mesh.*

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

In this tutorial, you will import STL files with surface meshes of the C3,C4 and C5 vertebrae, as well as the C3-C4 and C4-C5 intervertebral discs.

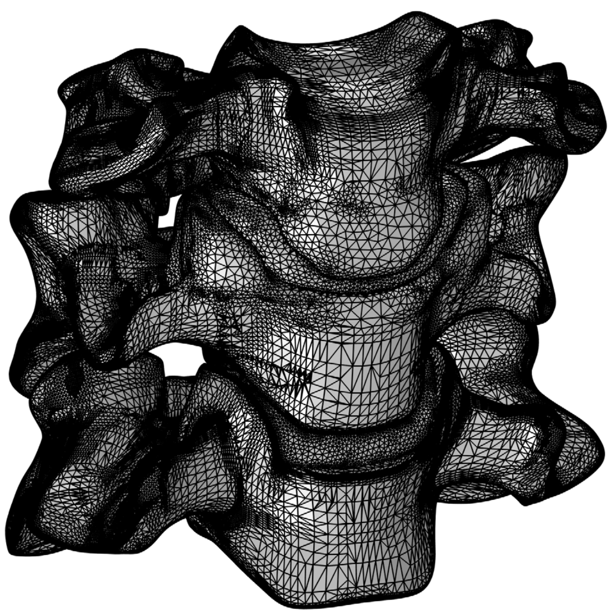
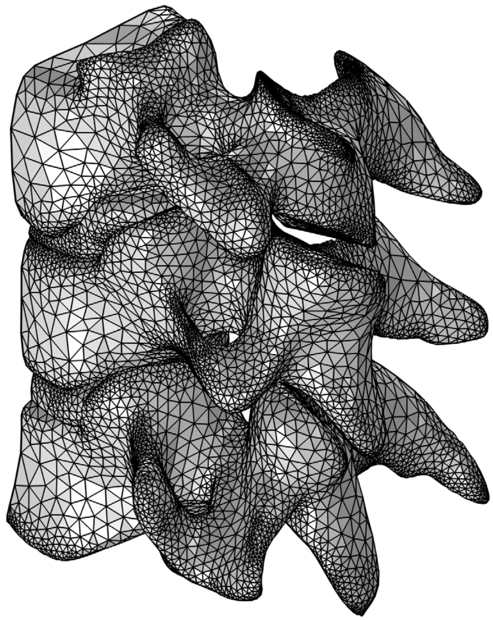


Figure 2: Imported STL meshes of three vertebrae and intervertebral discs that are used as a starting point for this tutorial.

The goal of this tutorial is to demonstrate a workflow for repairing and combining the imported surface meshes such that you can create five connected domains that can be filled with a tetrahedral mesh. The instructions also detail how to further simplify the imported meshes when you can assume that the problem is symmetric, and how to build a physics-controlled mesh.



To achieve this, the tutorial includes the following steps:

- Importing the STL files
- Identifying and repairing small defects such as gaps and intersecting elements to obtain watertight meshes for creating computational domains
- Moving the meshes into position before combining them
- Uniting the meshes
- Resolving intersections and gaps after the union
- Partitioning the domains with a plane and deleting domains you do not need
- Building a physics-controlled mesh

Application Library path: `COMSOL_Multiphysics/Meshing_Tutorials/stl_1_repair_imported_meshes`

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

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**.

This will set the length unit to mm for the Component.

GLOBAL DEFINITIONS

Add a Mesh Part that will be used to import and repair the mesh for the C3 vertebra. It is good practice to repair imported mesh files in separate mesh parts as it makes it easy to ensure that each imported mesh is fully repaired and contains a computational domain before moving on to the next step.

In the **Model Builder** window, right-click **Global Definitions** and choose **Mesh Parts** > **3D Part**.

C3

1 In the **Settings** window for **Mesh Part**, type `C3` in the **Label** text field.

2 Locate the **Units** section. Select the **Use units** checkbox.

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

This will set the length unit to mm for this Mesh Part.

Import 1

1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **C3** click **Import 1**.

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

3 Click **Browse**.

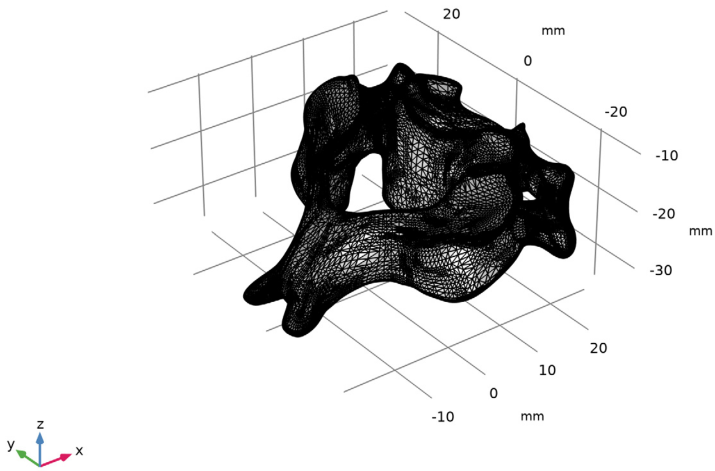
4 Browse to the model's Application Libraries folder and double-click the file `c3_vertebra.stl`.

5 From the **Boundary partitioning** list, choose **Minimal**.

This setting is suitable when importing mesh files that have no obvious boundary partitioning, for example, meshes generated by medical imaging techniques. After the import, the mesh will usually consist of only one boundary that you can partition as needed, using the available tools. Use the **Automatic** or **Detect boundaries** settings when importing meshes that contain planar faces and fillets, which can then be detected to partition the mesh accordingly.

Notice that the **Create domains** checkbox is selected by default. Domains are needed before you can generate a tetrahedral mesh in the vertebra. The import checks if the imported surface mesh forms any watertight regions, and if so, it creates domains.

6 Click **Import**.



The import automatically generates boundary selections based on the content of the STL file. The selections make it easier to select the proper entities in subsequent mesh operations and later when assigning materials and boundary conditions. If it is successful in creating domains, there will also be domain selections. Rename the boundary selection to something more descriptive.

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

Name	SOLID section in file
C3	mesh

Information

An **Information** node appears reporting that a domain could not be created. Expand the node and click on the **Information** subnode to see more details.

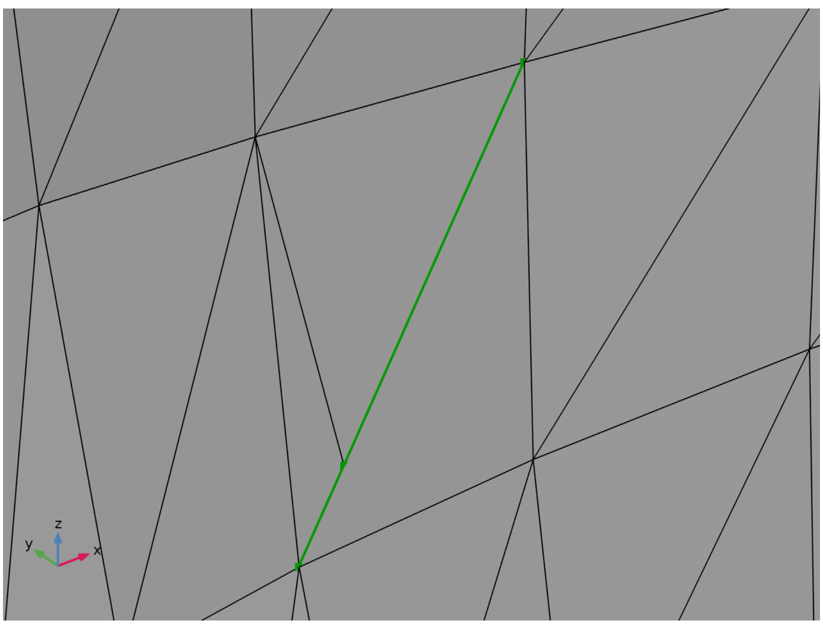
1 In the **Model Builder** window, expand the **Information** node, then click **Information**.

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

3 In the list, select **1**.

This information message indicates that the mesh of the vertebra does not form a watertight component. The edge that appears in the selection is an exterior edge that is adjacent to a hole in the mesh.

4 Click **Zoom to Selection**.



The hole is a slit-like hole with zero or almost zero area. On one side of the slit (on the right side of the green edge in the figure), there is one triangle element, whereas on the other side there are two elements. Since the import functionality cannot partition elements or add new ones, the mesh edges on the two sides of the slit could not be merged even with a larger tolerance.

C3

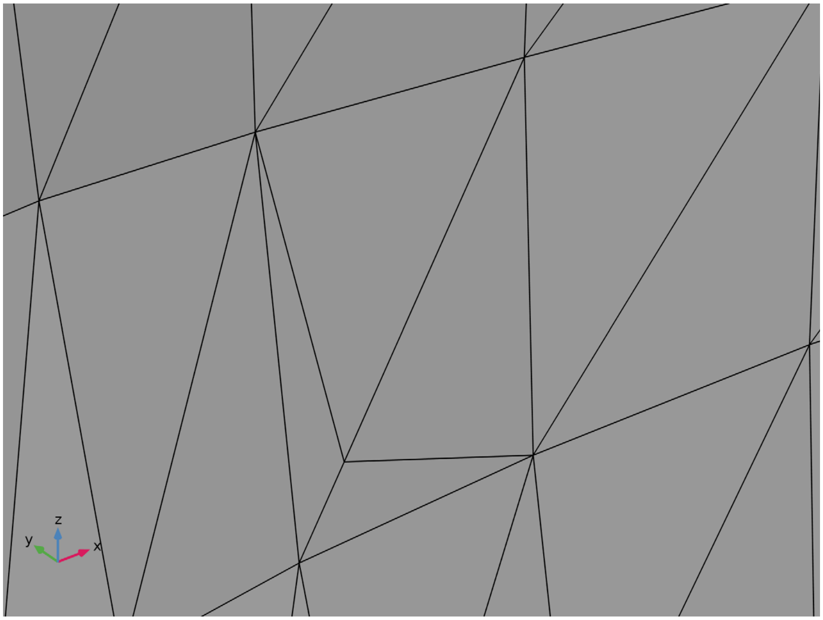
To repair the slit, you will use the Fill Holes operation, which can introduce new mesh edges in order to merge edges. It searches for holes within a selection of boundaries and will automatically fill all holes smaller than a specified tolerance.

1 In the **Mesh** toolbar, click **Cleanup and Repair** and choose **Fill Holes**.
2 In the **Settings** window for **Fill Holes**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **C3**.

Notice that the **Create Domains** checkbox is automatically selected, which means that once all holes are filled and the surface mesh forms a watertight boundary, a domain will be created. Select to create a domain selection and rename the operation to get a more descriptive selection name.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting domains** checkbox.
5 In the **Label** text field, type `C3`.

6 Click **Build Selected**.



The slit has been repaired by introducing a mesh edge on the right side of the slit, merging the two sides of the hole, and removing the edge. This makes the mesh watertight and a domain has been created, which is reported in the **Messages** window.

GLOBAL DEFINITIONS

Next, add a new Mesh Part and import the mesh of the C3-C4 intervertebral disc.

In the **Model Builder** window, under **Global Definitions** right-click **Mesh Parts** and choose **3D Part**.

C3-C4 DISC

1 In the **Settings** window for **Mesh Part**, type `C3-C4 Disc` in the **Label** text field.

2 Locate the **Units** section. Select the **Use units** checkbox.

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

Import 1

1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **C3-C4 Disc** click **Import 1**.

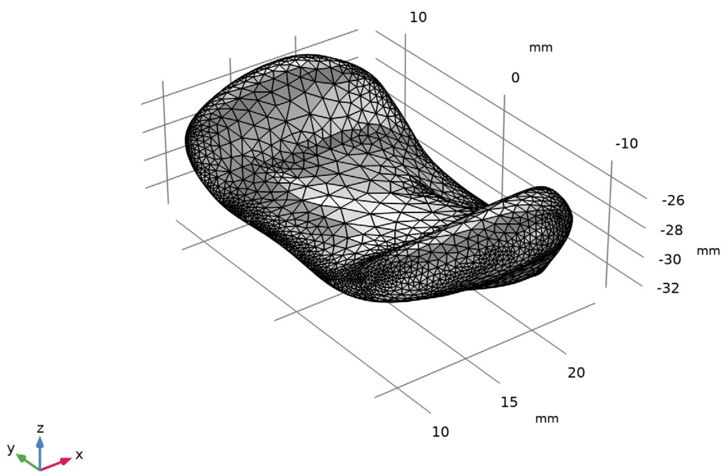
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `intdiscC_3-4.stl`.

5 From the **Boundary partitioning** list, choose **Minimal**.

6 Click **Import**.



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

Name	SOLID section in file
C3-C4	mesh

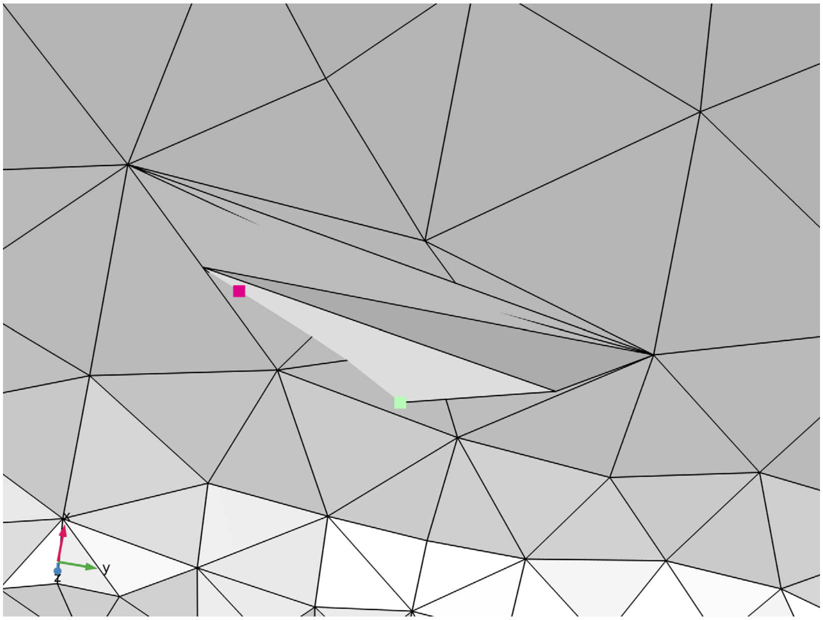
Information

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

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

3 Click **Center at Coordinates**.

Rotate the mesh and zoom in further to see the view below. For more complex meshes when the coordinates are hidden behind mesh faces, you can also click the **Clip Around Coordinates** button to remove everything except the mesh closest to the coordinates in the view. If you do, click the **Remove Clipping** button to return to the view with the full mesh.



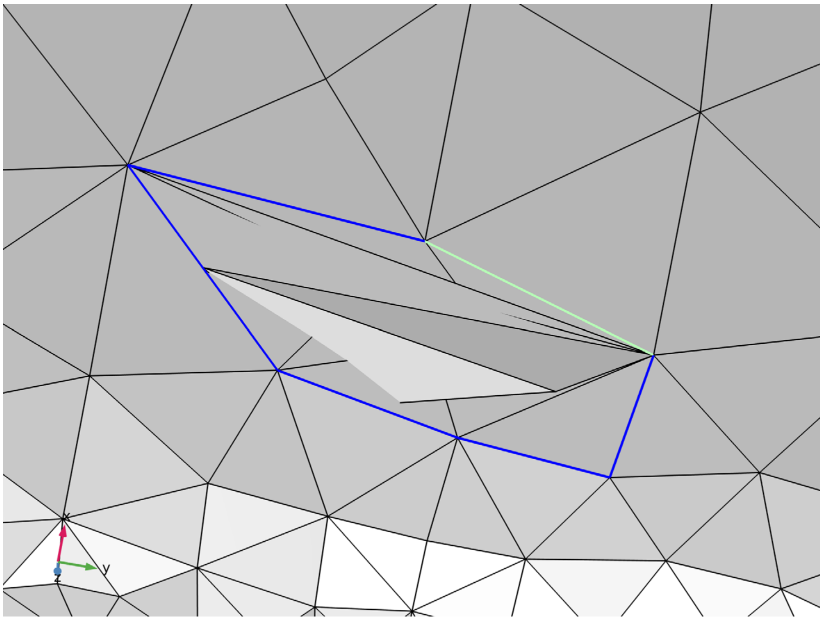
Also in this case, a domain could not be created, this time because there are intersecting mesh elements. To repair the mesh, partition it to create a boundary containing the intersecting elements, delete this boundary, and finally, generate a new mesh face to cover the resulting hole. Here, do the partitioning manually using the Create Edges operation. For a faster workflow, you can also try to use the add-in `mesh_partition_with_ball` available in the COMSOL Multiphysics Add-in Libraries. See the model *Spray Particle Deposition in Human Airways* in the Application Gallery on the COMSOL website for an example of how to use the Add-In to clean up an imported STL mesh.

Create Edges 1

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

2 Select mesh edges around the intersecting elements by clicking them in the **Graphics** window, similar to what is shown in the image below. The selection is most easily done with transparency off. The selected edges can differ from the ones selected in the figure, as long as the edges form a closed loop, include the intersecting elements, and delimit

only a small region around the intersecting elements. 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**, locate the **Selections of Resulting Entities** section.

4 Select the **Resulting small faces** checkbox to create a selection containing the boundary with the intersecting elements.

5 Click **Build Selected** to generate the edges and partition the boundary.

Delete Entities 1

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

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

3 From the **Selection** list, choose **Create Edges 1 (Small faces)** (the boundary with the intersecting elements).

4 Click **Build Selected**.

There is now a hole in the surface mesh, so before continuing, the hole must be sealed. 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. Here, use the Fill Holes operation. Note that the mesh faces generated by both of these operations will be minimal faces that strive to be as planar as possible.

C3-C4

1 In the **Mesh** toolbar, click **Cleanup and Repair** and choose **Fill Holes**.

2 In the **Settings** window for **Fill Holes**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **C3-C4**.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting domains** checkbox.

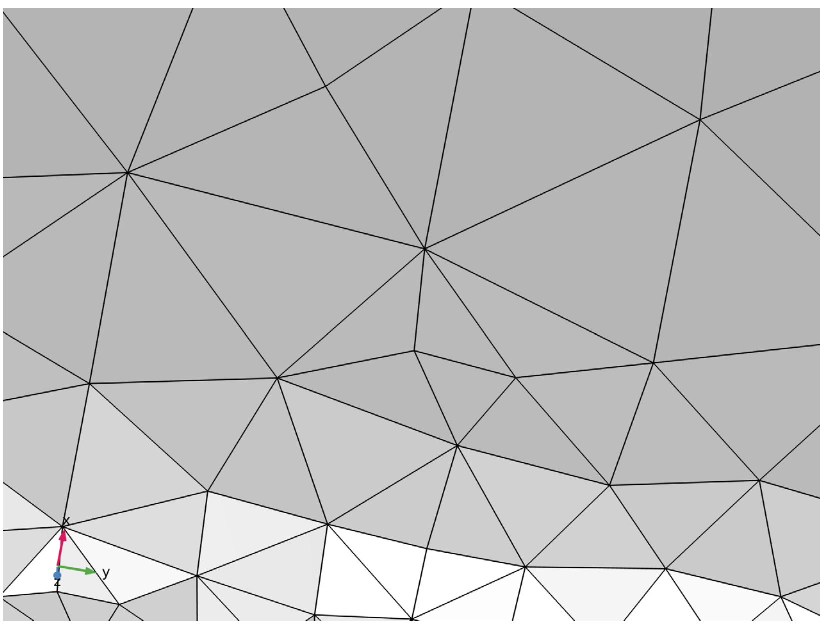
5 In the **Label** text field, type `C3-C4`.

6 Click **Build Selected**.

The hole is sealed with a new mesh face and the result is a smooth mesh containing a domain. Check that the domain selection **C3-C4 Disc** has been added in the Selection List.

SELECTION LIST

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



2 In the **Model Builder** window, click **C3-C4**.

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

GLOBAL DEFINITIONS

Add a third Mesh Part for the C4 vertebra.

In the **Model Builder** window, under **Global Definitions** right-click **Mesh Parts** and choose **3D Part**.

C4

1 In the **Settings** window for **Mesh Part**, type `C4` in the **Label** text field.

2 Locate the **Units** section. Select the **Use units** checkbox.

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

Import 1

1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **C4** click **Import 1**.

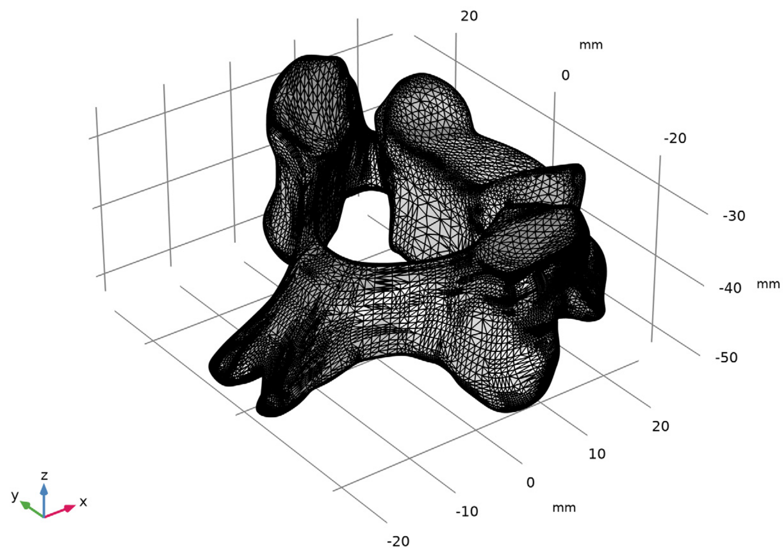
2 In the **Settings** window for **Import**, locate the **Import** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `c4_vertebra.stl`.

5 From the **Boundary partitioning** list, choose **Minimal**.

6 Click **Import**.

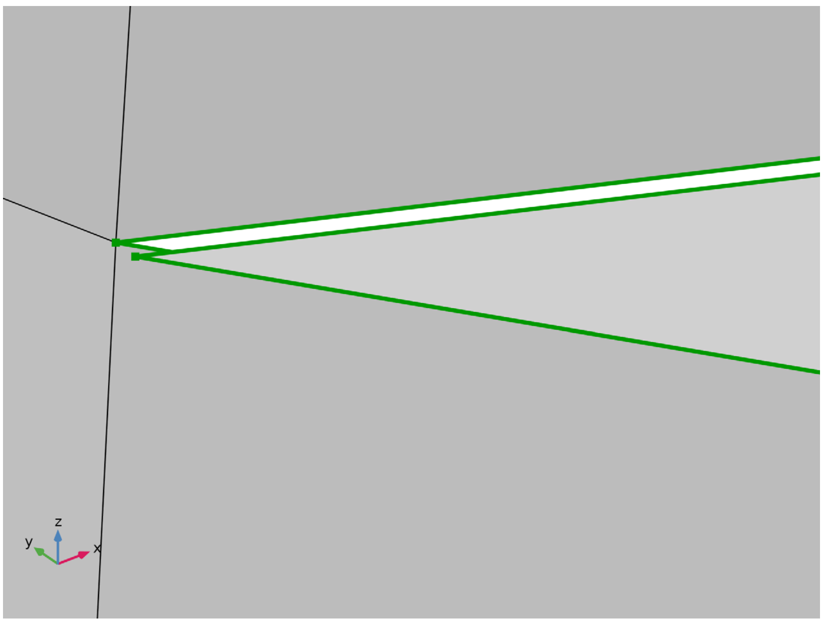


Information

1. In the **Model Builder** window, expand the **Global Definitions** > **Mesh Parts** > **C4** > **Import 1** > **Information** node, then click **Information**.
2. In the **Settings** window for **Information**, locate the **Geometric Entity Selection** section.
3. In the list, select **1**.

   Once again, the import failed to create a domain due to a hole in the mesh.
4. Click **Zoom to Selection**.

   Zoom in as much as possible on the tip of the v-shaped edge to see that the hole is caused by a mismatch in coordinates for one of the triangles.



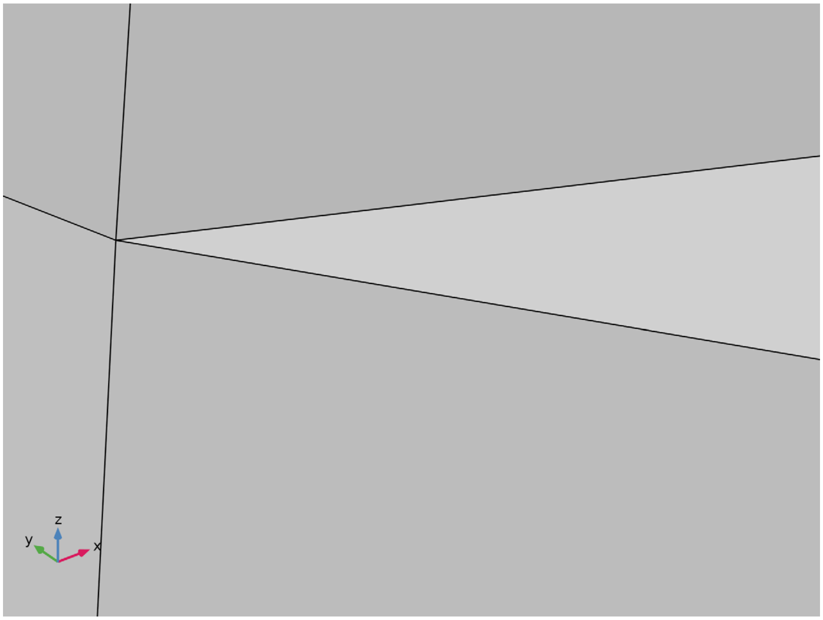
STL files contain a set of triangles that need to be stitched together within a tolerance. In this case, the automatic tolerance is not large enough to connect the vertices of the triangles. This means that this type of hole can be repaired by increasing the Repair tolerance setting for the import.

Import 1

1. In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **C4** click **Import 1**.
2. In the **Settings** window for **Import**, locate the **Import** section.
3. From the **Repair tolerance** list, choose **Absolute**.

4 In the **Absolute tolerance** text field, type `5e-4[mm]`.

5 Click **Import**.



Using the larger repair tolerance, the element edges are now merged. As a result, the imported mesh is watertight and contains a domain with a corresponding domain selection. Rename the selections to something more descriptive.

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

Name	SOLID section in file
C4	mesh

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

Name	SOLID section in file
C4	mesh

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

GLOBAL DEFINITIONS

Add another mesh part for the C4-C5 intervertebral disc.

In the **Model Builder** window, under **Global Definitions** right-click **Mesh Parts** and choose **3D Part**.

C4-C5 DISC

1 In the **Settings** window for **Mesh Part**, type `C4-C5 Disc` in the **Label** text field.

2 Locate the **Units** section. Select the **Use units** checkbox.

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

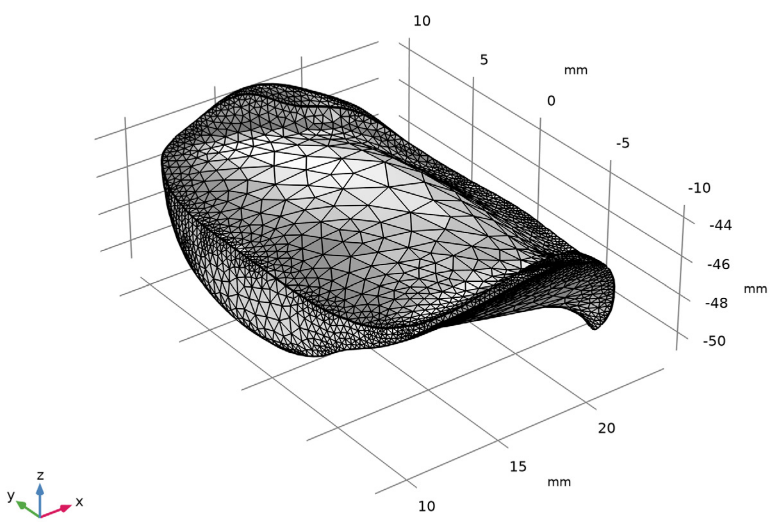
1 In the **Model Builder** window, under **Global Definitions** > **Mesh Parts** > **C4-C5 Disc** click **Import 1**.

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 `intdiscC_4-5.stl`.

5 Click **Import**.



This mesh has been prepared to not contain anything that needs repairing.

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

Name	SOLID section in file
C4-C5	mesh

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

Name	SOLID section in file
C4-C5	mesh

GLOBAL DEFINITIONS

Next, add a mesh part for the C5 vertebra.

In the **Model Builder** window, under **Global Definitions** right-click **Mesh Parts** and choose **3D Part**.

C5

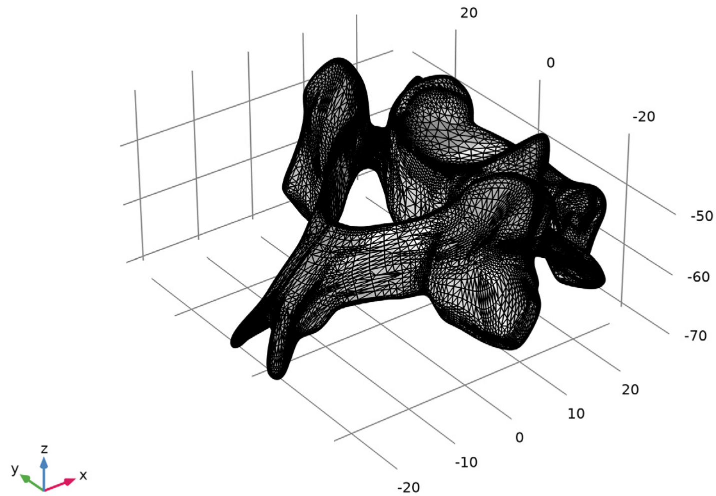
1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Browse**.

3 Browse to the model's Application Libraries folder and double-click the file `c5_vertebra.stl`.

4 From the **Boundary partitioning** list, choose **Minimal**.

5 Click **Import**.



This mesh has been prepared to fit perfectly with the adjacent vertebrae so here we want the import to partition the boundaries automatically. It is also prepared not to need repair.

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

Name	SOLID section in file
C5	mesh

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

Name	SOLID section in file
C5	mesh

8 In the **Model Builder** window, click **Mesh Part 5**.

9 In the **Settings** window for **Mesh Part**, type `C5` in the **Label** text field.

10 Locate the **Units** section. Select the **Use units** checkbox.

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

GLOBAL DEFINITIONS

Next, add a Mesh Part where the meshes will be combined into one.

In the **Model Builder** window, under **Global Definitions** right-click **Mesh Parts** and choose **3D Part**.

COMBINED MESH

1 In the **Settings** window for **Mesh Part**, type `Combined Mesh` in the **Label** text field.

2 Locate the **Units** section. Select the **Use units** checkbox.

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

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

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

3 From the **Source** list, choose **Meshing sequence**.

4 From the **Mesh** list, choose **C3**.

5 Select the **Import unmeshed domains** checkbox to also import the domain you created earlier for the volume enclosed by the mesh. Domains are needed for most physics interfaces when running simulations in the volumes inside.

6 Click **Import**.

Import 2

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

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

3 From the **Source** list, choose **Meshing sequence**.

4 From the **Mesh** list, choose **C3-C4 Disc**.

5 Select the **Import unmeshed domains** checkbox.

6 Click **Import**.

Information

The information message states that the mesh has intersecting elements and gives the suggestion to use the Union operation to compute the intersection. Rotate the mesh and zoom in on the coordinates to see where they are located.

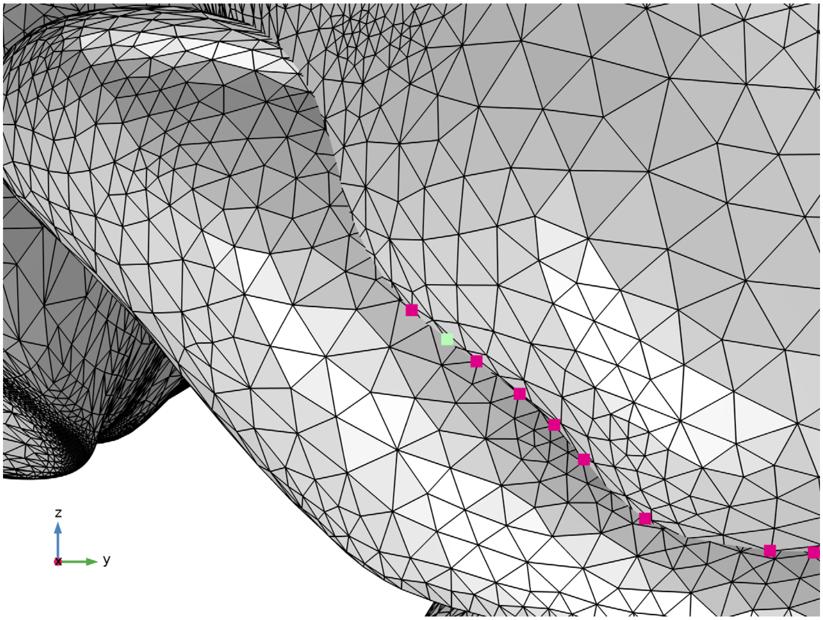
1 Click the **Go to YZ View** button in the **Graphics** toolbar.

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

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

4 From the **Select location** list, choose **Location 11 (23.45, -6.925, -29.37)**.

5 Click **Center at Coordinates**.



Zooming in on the intersecting elements confirms that the mesh of the vertebra intersects that of the intervertebral disc. Before resolving the intersection with a Union operation, you will also import the rest of the meshes into this mesh part.

Import 3

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

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

3 From the **Source** list, choose **Meshing sequence**.

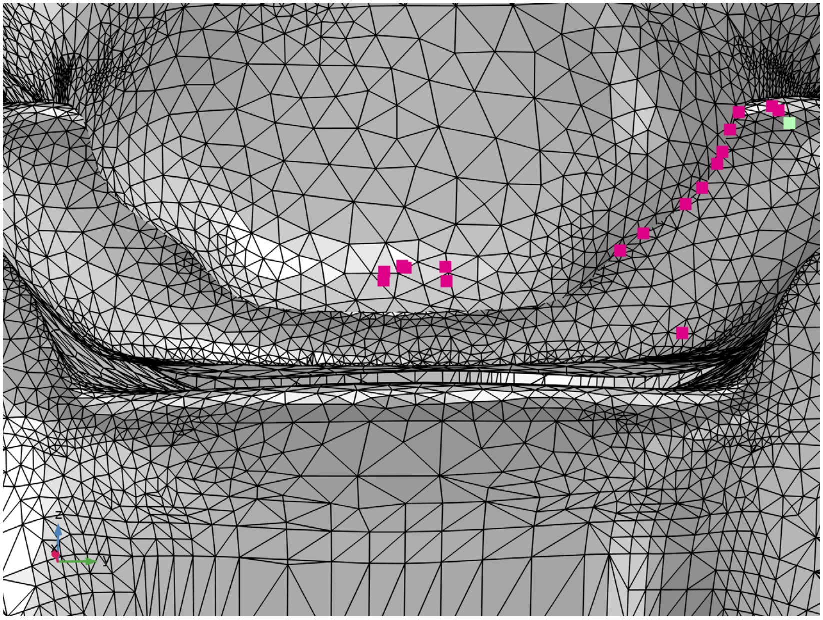
4 From the **Mesh** list, choose **C4**.

5 Select the **Import unmeshed domains** checkbox.

6 Click **Import**.

Information

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



The C4 vertebra does not intersect the intervertebral disc, in fact there is a gap between the two.

Import 3

To adjust the position of the C4 vertebra you can use a Transform attribute to the Import node. This allows for moving, rotating, and scaling an imported mesh. You can add several Transform attributes to an Import node, to, for example, rotate the mesh around more than one axis.

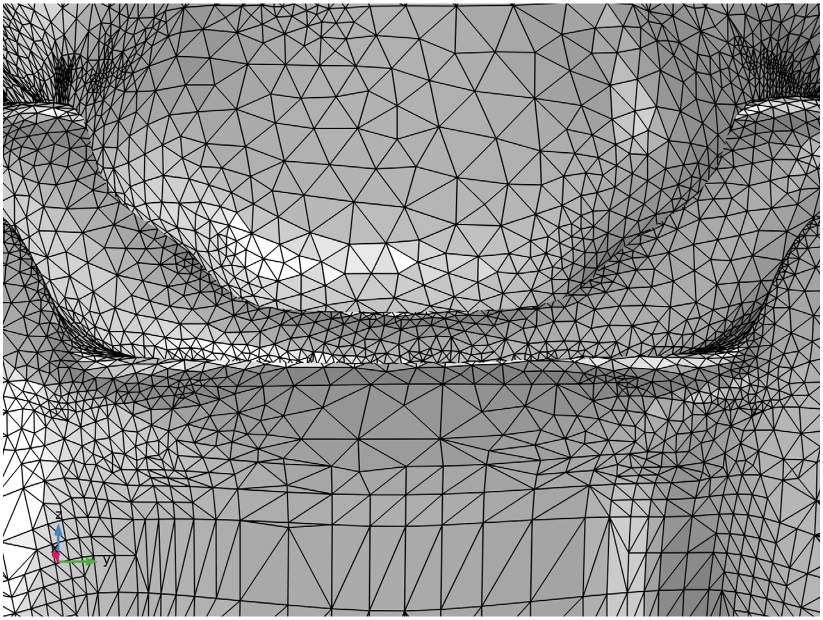
Transform 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Transform**.

2 In the **Settings** window for **Transform**, locate the **Displacement** section.

3 In the **z** text field, type `1[mm]`.

4 Click **Build Selected**.



Import 4

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

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

3 From the **Source** list, choose **Meshing sequence**.

4 From the **Mesh** list, choose **C4-C5 Disc**.

5 Select the **Import unmeshed domains** checkbox.

6 Click **Import**.

Information

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

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

3 Click **Center at Coordinates**.

The C4-C5 disc and the C5 vertebra have been prepared with identical meshes on the touching boundaries. These boundaries only need to be merged.

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

Import 5

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

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

3 From the **Source** list, choose **Meshing sequence**.

4 From the **Mesh** list, choose **C5**.

5 Select the **Import unmeshed domains** checkbox.

6 Click **Import**.

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

Add a Clip Plane to see inside the mesh and inspect how the meshes are intersecting.

8 In the **Graphics** window toolbar, click ▾ next to **Clipping**, then choose **Add Clip Plane**.

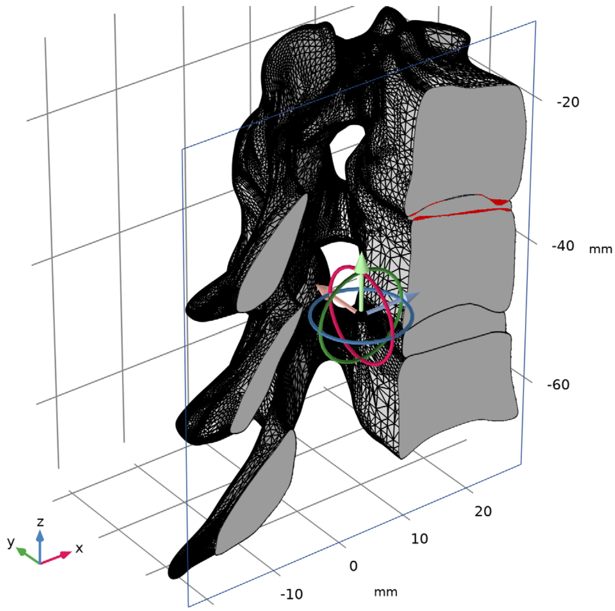
9 In the **Graphics** window, hover over the frame for **plane1** and right-click to open up the Graphics context menu. Select **Align to y-Axis** to rotate **plane1** to be parallel with the xz plane.

10 In the **Graphics** window, hover over the frame for **plane1** and right-click to open up the Graphics context menu. Select **Invert Clipping** to view the part of the mesh that is positioned on the positive side of the y axis.

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

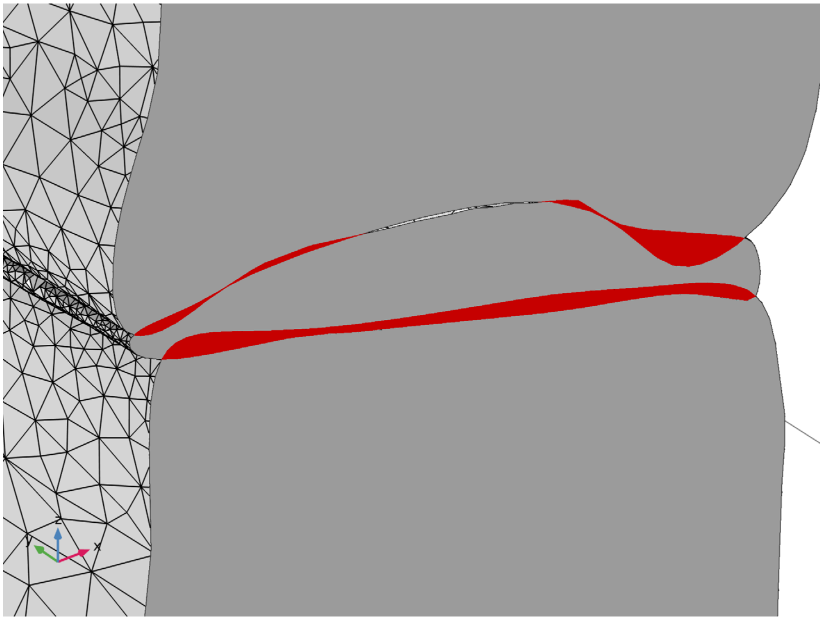
12 In the **Graphics** window toolbar, click ▾ next to **Clipping Active**, then choose **Show Cross Section** to show the cross-sectional faces of the domains that are cut with the Clip Plane. Since the cross-sectional faces can only be visualized where domains exists, this is one way to check if the mesh operations were actually able to create domains after repairing the holes. Note that the cross-sectional faces are not shown, even if there are domains, if the **Wireframe** option is enabled in the **Graphics** toolbar, so remember to check the status of that option if you do not see any cross-sections.

13 In the **Graphics** window toolbar, click ▾ next to **Clipping Active**, then choose **Colorize** to disable the colors of the domains. This gives a more clear view of the overlaps between the imported meshes.



It is possible to click and drag the frame of the Clip Plane in the graphics to pan through the volumes. If you do, move the frame back to somewhere close to y=0 before continuing.

Zoom in on the cross section of the C3-C4 intervertebral disc.



The red areas show where the domains of the imported meshes intersect. In this view you can see that these meshes are not even close to match up no matter how careful you are at positioning the meshes in relation to each other. The black edges on top of the disc indicate that there is a gap between the disc and the C3 vertebra. First unite the meshes and then resolve the intersections and the gap.

Union 1

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

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

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

4 Select Domains 1–3 only.

5 Locate the **Cleanup** section. From the **Repair tolerance** list, choose **Absolute**.

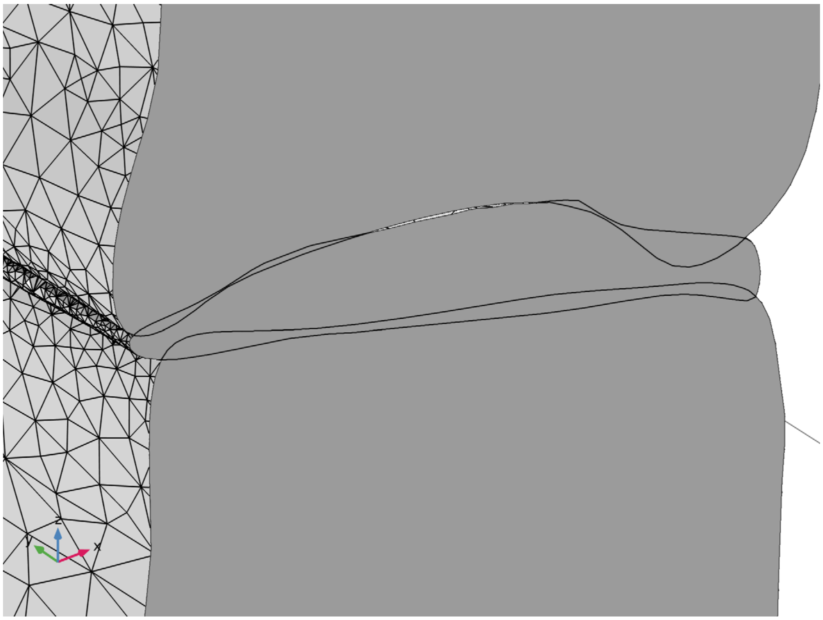
6 In the **Absolute tolerance** text field, type `6e-4[mm]`.

Lowering the tolerance usually helps when calculating a complex intersection. In some cases, when the element size differs too much between the input meshes, it can be necessary to remesh the boundaries before the Union operation to obtain similar sized elements. This can help with a successful computation of a mesh union, but is not needed for our vertebra mesh.

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

Switching to a linear placement of the mesh vertices helps here as the curved surfaces almost coincide at some locations.

8 Click **Build Selected**.



The red areas are no longer visible as the Union operation has formed separate domains for each intersection region. You can also confirm this using the **Selection List** window.

SELECTION LIST

1 Go to the **Selection List** window to verify that there are 7 domains in the mesh, one large domain for each of the imported meshes and two that have been created for the intersecting regions. Keep the window open as you will use it more later on.

In the bottom of the **Selection List** window, the available named selections are also listed for easy access.

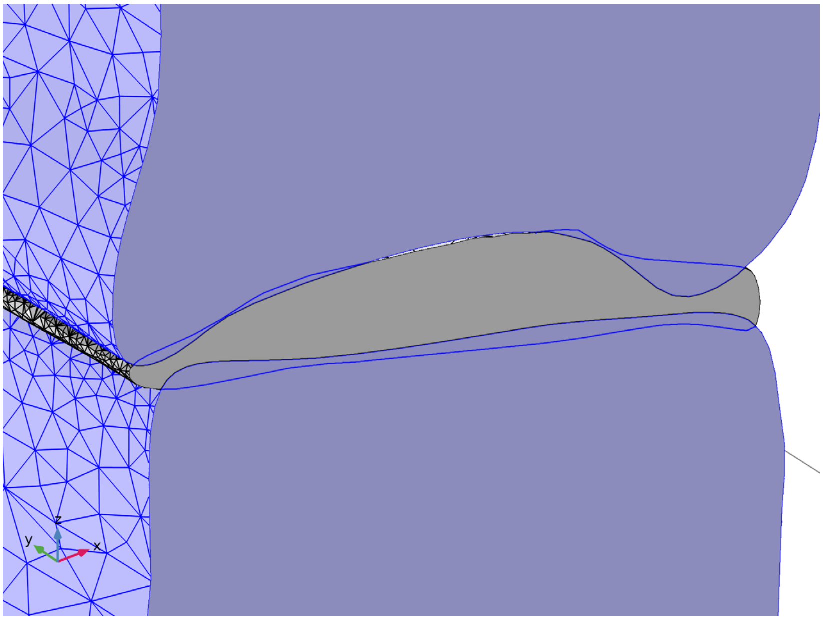
COMBINED MESH

Assuming that you are only interested in the five connected domains for the vertebrae and discs, continue with adding first a Join Entities operation to eliminate the additional domains, and then a Merge Entities operation to collapse the void regions.

Join Entities 1

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

2 Select Domains 1, 3, 6, and 7 only.



Note that you can just as well add the overlap domains to the domain of the disc, but in this case add them to the domains of the vertebrae.

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

Check in the **Selection List** window that there are now five domains, one for each vertebra and two for the discs.

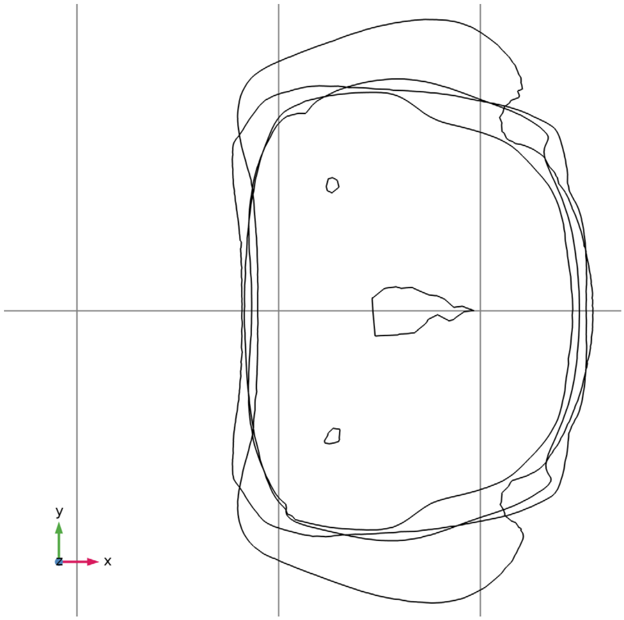
By switching to wireframe rendering, you can easily inspect the edges formed by the union to see if any edge loops were created around void regions between the domains.

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 Click the **Mesh Rendering** button in the **Graphics** toolbar.

6 Click the **Clipping Active** button in the **Graphics** toolbar.

7 Click the **Go to XY View** button in the **Graphics** toolbar. Zoom in further to see the view below.



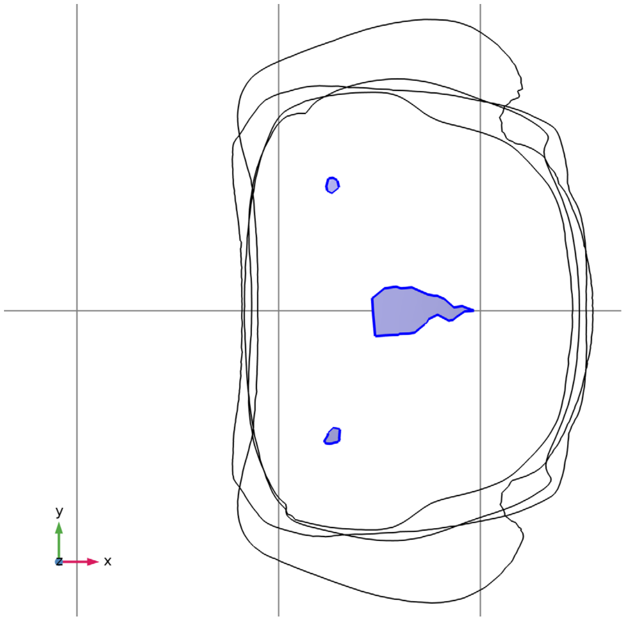
The large loops are the expected intersection edges where the boundaries of the vertebrae and discs intersect. The smaller edge loops are the bounding edges of thin void regions between the C3 vertebra and the disc. These are the result of the gaps you could see when looking at the cross section with clipping applied in the view.

Merge Entities 1

1 In the **Mesh** toolbar, click **Cleanup and Repair** and choose **Merge Entities**.

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

3 Draw a box that includes the smaller edge loops only. This selects the faces of the voids that to merge.



4 In the **Settings** window for **Merge Entities**, click **Build Selected**.

This still leaves one set of boundaries for each of the voids. Add another Join Entities operation to join these boundaries with the adjacent boundary.

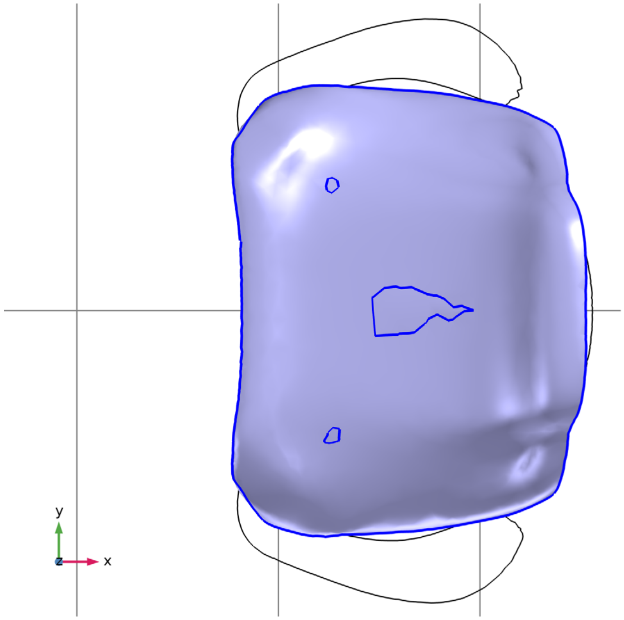
Join Entities 2

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

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

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

4 Select Boundaries 7 and 10–12 only.



5 Click **Build Selected**.

In the **Graphics** window, you should now only see the larger edge loops in the intersection between the discs and vertebrae.

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

Merge Entities 2

Next, Add two Merge Entities operations to merge the boundaries adjacent to the C4-C5 disc.

1 In the **Mesh** toolbar, click **Cleanup and Repair** and choose **Merge Entities**.

2 Select Boundaries 5 and 6 only (the boundary of the C5 vertebra and the bottom boundary of the adjacent disc).

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

Merge Entities 3

1 In the **Mesh** toolbar, click **Cleanup and Repair** and choose **Merge Entities**.

2 Select Boundaries 4 and 7 only (the boundary of the C4 vertebra and the top boundary of the C4-C5 disc).

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

Next, restore the mesh rendering before continuing the work on the mesh.

4 Click the **Mesh Rendering** button in the **Graphics** toolbar.

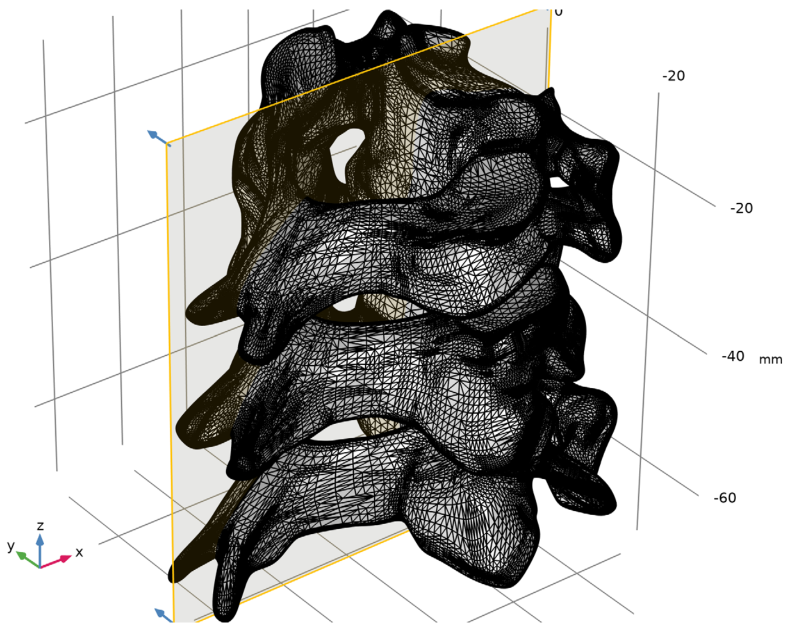
Intersect with Plane 1

It is often possible to simplify a simulation by exploring symmetries in the geometry and physics. Assuming that the xz-plane is a symmetry for the vertebrae and disc, in the following you will partition and keep only half of the domains.

1 In the **Mesh** toolbar, click **Booleans and Partitions** and choose **Intersect with Plane**.

2 In the **Settings** window for **Intersect with Plane**, locate the **Plane Definition** section.

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



Create a selection for the part of the mesh that is below the plane. Selections can be useful during the physics setup, for example to assign materials and boundary conditions. If you, for example, will assign a symmetry boundary condition for the planar faces of the intersection, make sure to also select the checkbox for **Intersection faces**.

4 Locate the **Selections of Resulting Entities** section. Select the **Entities below** checkbox.

5 From the **Show outside part** list, choose **Off**.

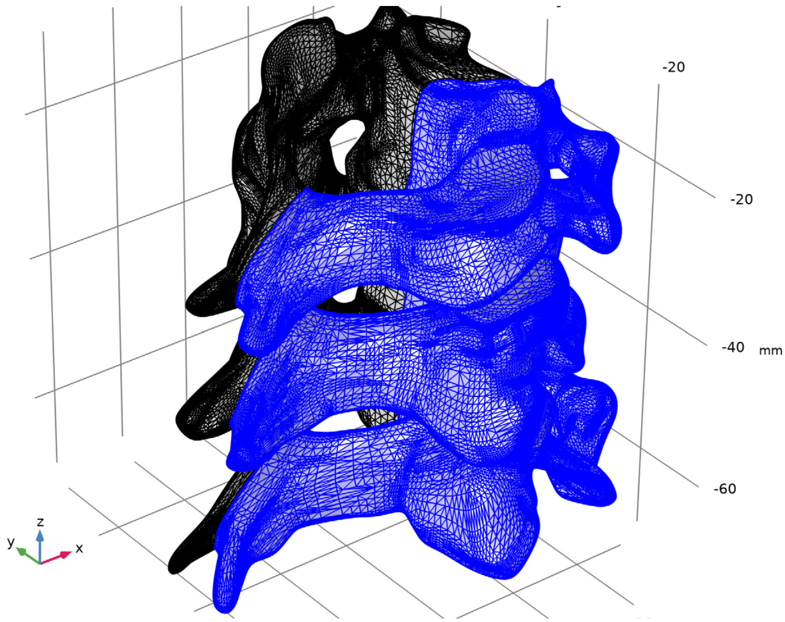
6 Click **Build Selected**.

Delete Entities 1

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

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

3 From the **Selection** list, choose **Intersect with Plane 1 (Below)**.



4 Click **Build Selected**.

Remesh Faces 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Remesh Faces**.

Remesh the surfaces to prepare for part 2 where the surfaces of the vertebrae will be intersected the geometry of the screws.

2 In the **Settings** window for **Remesh Faces**, locate the **Boundary Selection** section.

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

Size

1 In the **Model Builder** window, expand the **Remesh Faces 1** node, then click **Size**.

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

3 From the **Predefined** list, choose **Extra fine**.

4 Click **Build Selected**.

Information

According to the information messages, the element size used does not resolve narrow regions in the ridges of the vertebrae. It is good practice to investigate if you need to refine the mesh further to better resolve geometric details and the solution field in such regions. However, if you know that the mesh is good enough for your purposes, you can safely ignore this information.

You are now done with preparing the geometric model for the vertebrae and discs. The remaining steps will generate a physics-controlled mesh that can be used for computation. You do this in the Component.

COMPONENT 1 (COMP1)

First, import the combined mesh of the vertebrae and discs into the Mesh 1 node, then build the physics-controlled mesh of Mesh 2 that operates on the geometric model of Mesh 1. In the second part of this tutorial series, Mesh 1 will be used to combine the vertebrae with the geometry of three pedicle screws.

MESH 1

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

Import - Combined mesh

1. In the **Mesh** toolbar, click **Import**.
2. In the **Settings** window for **Import**, type `Import - Combined mesh` in the **Label** text field.
3. Locate the **Import** section. From the **Source** list, choose **Meshing sequence**.
4. From the **Mesh** list, choose **Combined Mesh**.
5. Select the **Import unmeshed domains** checkbox to also import the domains formed in the Mesh Parts.
6. Click **Import**.

   Add another meshing sequence to generate a physics-controlled mesh.

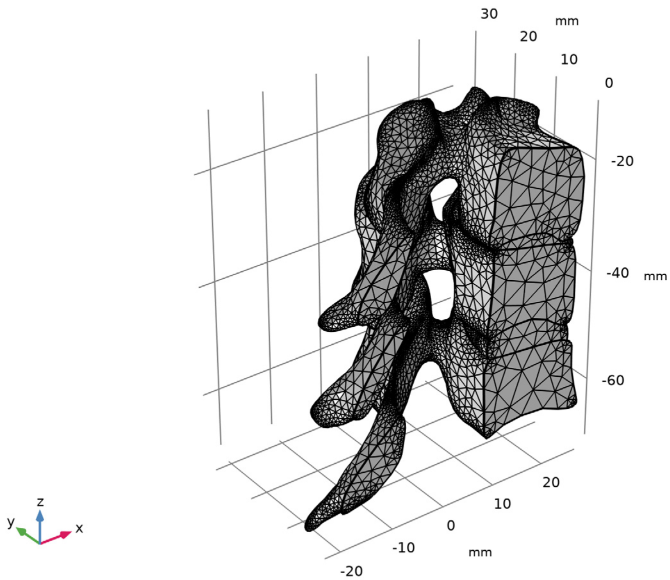
PHYSICS-CONTROLLED MESH

1. In the **Mesh** toolbar, click **Add Mesh** and choose **Add Mesh**.
2. In the **Settings** window for **Mesh**, type `Physics-Controlled Mesh` in the **Label** text field.
3. In the **Model Builder** window, click **Physics-Controlled Mesh**.
4. Locate the **Physics-Controlled Mesh** section. From the **Element size** list, choose **Coarser**.

5 Click **Build All**.

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

7 In the **Model Builder** window, click **Physics-Controlled Mesh**.



The volume mesh is now ready and can be used to set up a simulation. This concludes the first part of this tutorial. In the second part, this mesh will be combined with a geometry for three pedicle screws inserted into the vertebrae.