

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**.

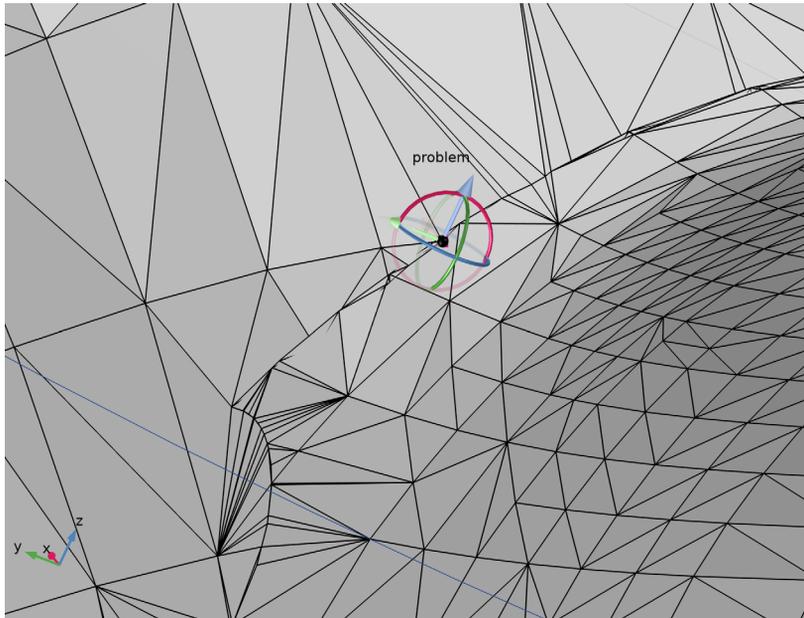
2 In the **Settings** window for **Information**, locate the **Information** section.

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**.