1–way Structural FSI for a Flow Probe

This tutorial shows how to prepare a 1-way steady force transfer from Fluent to Mechanical in ANSYS Workbench.

In this tutorial you will learn how to do:

- Setup of the Static Structural case given a completed Fluent solution.
- Solution of the Static Structural case using an imported load.
- Analysis of the results of a 1-way FSI simulation within Mechanical.

1. Problem Description

This example considers a probe that is protruding into a flow field and subjected to a high flow velocity (100 m/s). You will determine how the flow affects the probe structurally.

2. Setup And Solution

The following sections describe the setup and solution steps for this tutorial:

2.1. Preparation

1. Create a working folder on your computer.
2. Copy the file `Probe_1way.wbpz` to the working folder.

2.2. Starting Workbench

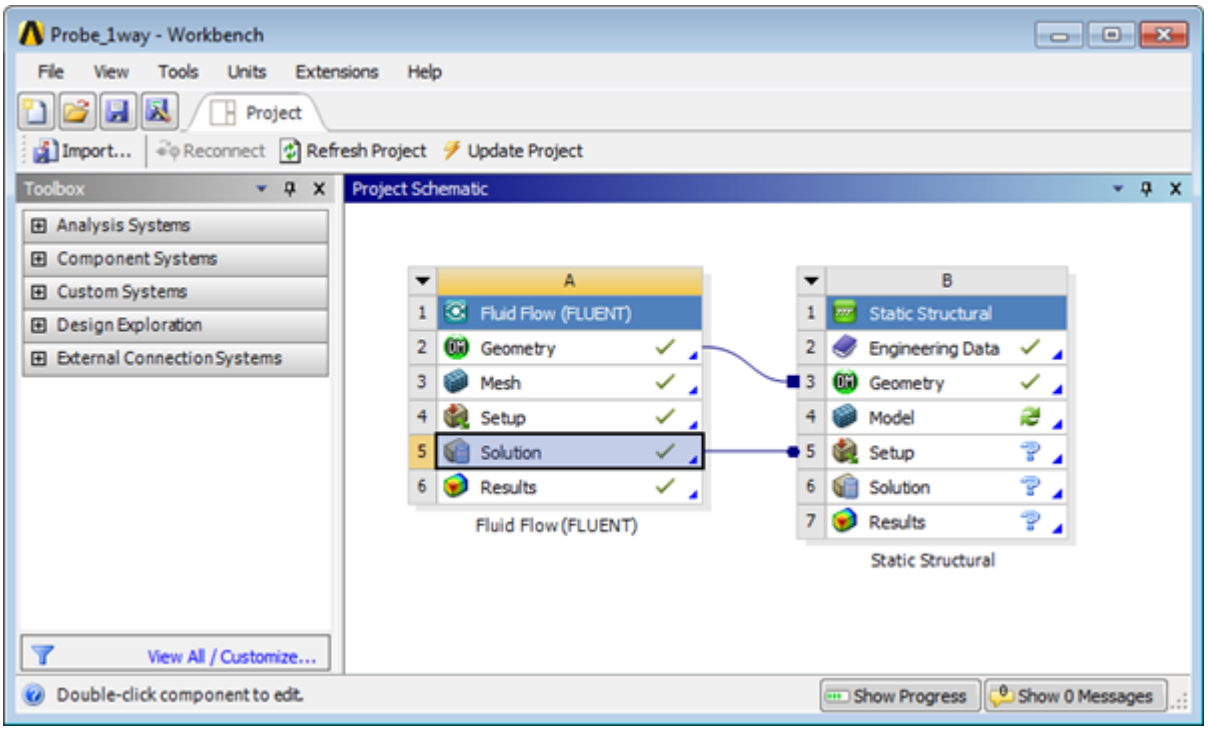
1. Start ANSYS Workbench and select **File >Restore Archive...** from the menu.
   a. Select `Probe_1way.wbpz` from your working folder.
   b. Save to your working folder.

> **Note**
>
> The fluid flow solution for this exercise has already been completed.

2. Drag a **Static Structural** analysis system onto the **Project Schematic** and drop it onto the **Solution** cell (**A5**) of the **Fluid Flow (FLUENT)** system.

Note

This shares the **Geometry** cells and links the **Solution** cell of the **Fluid Flow (FLUENT)** system to the **Setup** cell of the **Static Structural** system.



Note

The links created will allow for a common geometry, as well as transfer the **Fluid Flow** solution to the **Static Structural** setup. The solution data will be used as a pressure load in the **Static Structural** system.

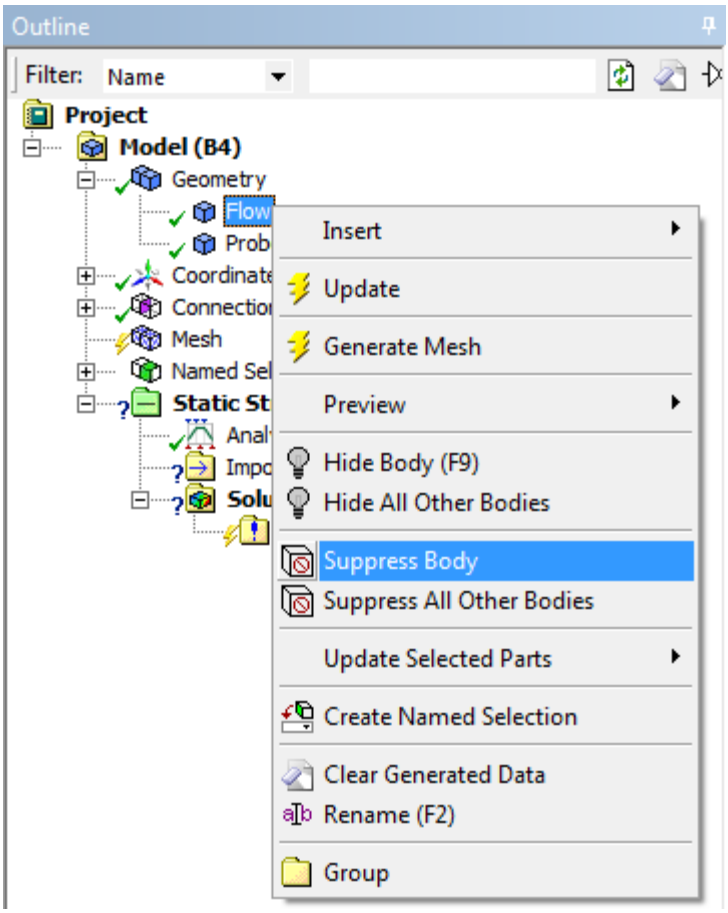
2.3. Mesh

1. In the **Project Schematic** window double-click on the **Model** cell, **B4**.

Note

This will launch ANSYS Mechanical.

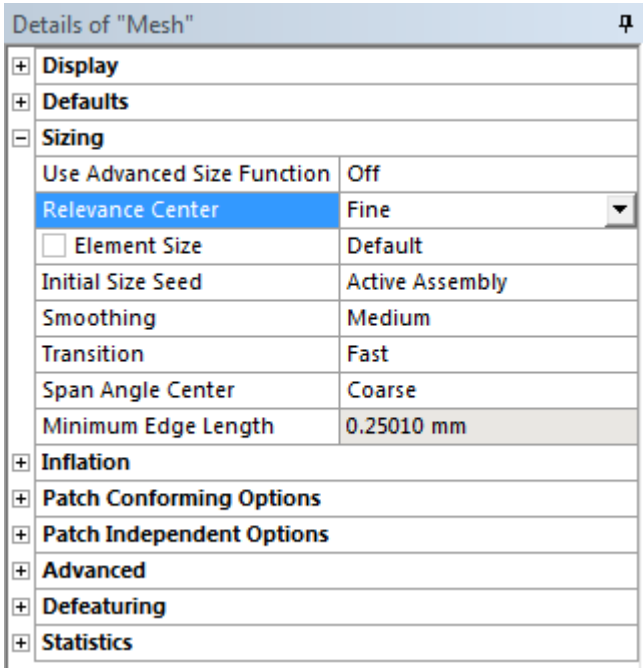
2. In the model tree, expand **Geometry**.



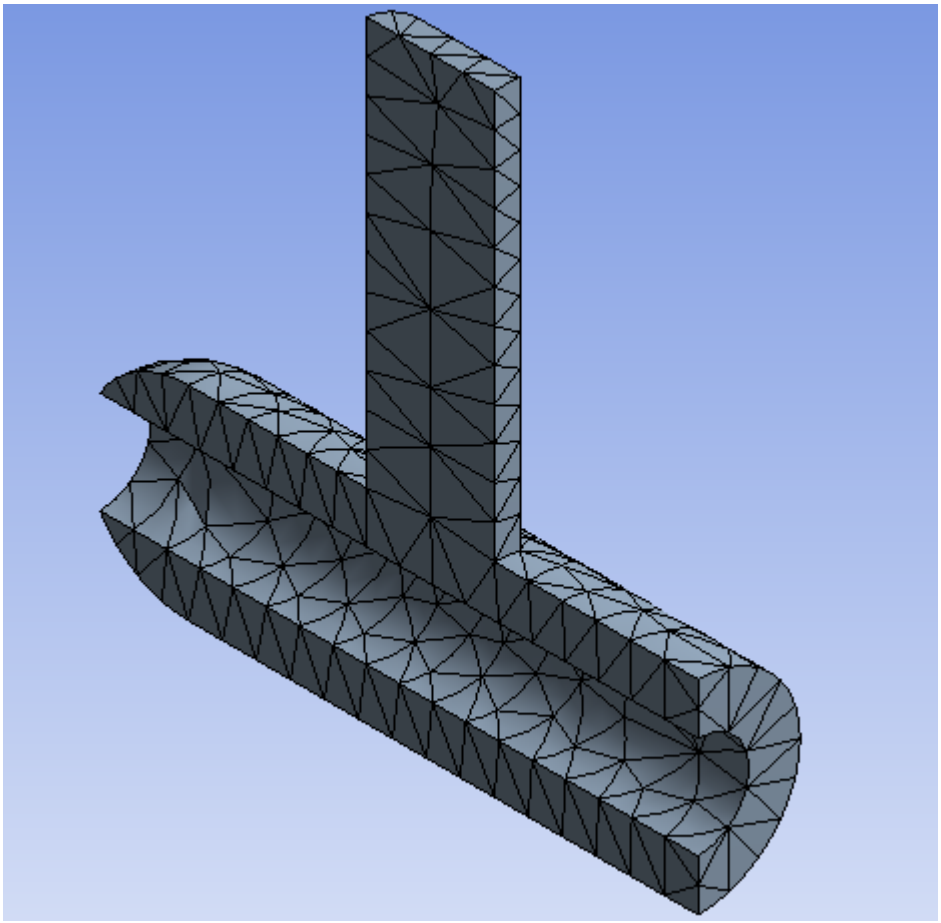
- Right-click **Flow** and select **Suppress Body** from the context menu.

3. Select **Mesh** in the tree.

- In **Details of Mesh**, expand **Sizing** and select **Fine** from **Relevance Center** drop-down list.



4. Right-click **Mesh** in the tree and select **Generate Mesh**.



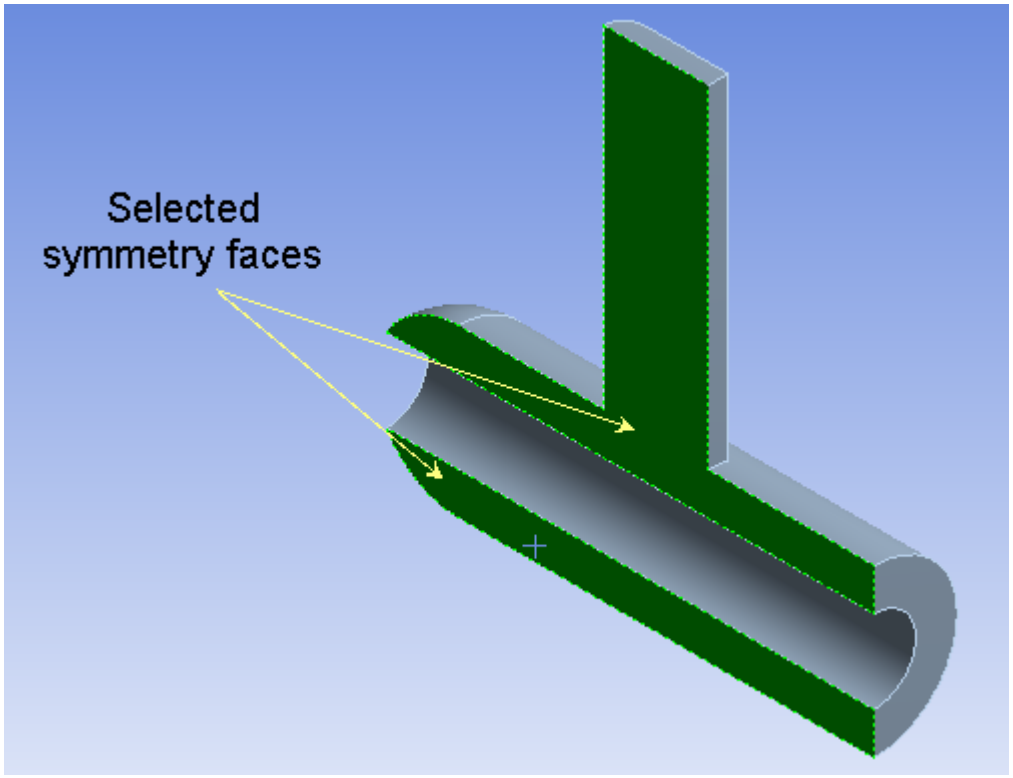
2.4. Setup

1. Right-click on **Static Structural (B5)**, in the **Model** tree.

   a. From the context menu select **Insert** > **Frictionless Support**.

   b. Select the two symmetry faces of the geometry as shown in Figure 45: Selected Faces for Frictionless Support (p. 5) and click **Apply** in the **Details of Frictionless Support** panel.

Figure 45: Selected Faces for Frictionless Support

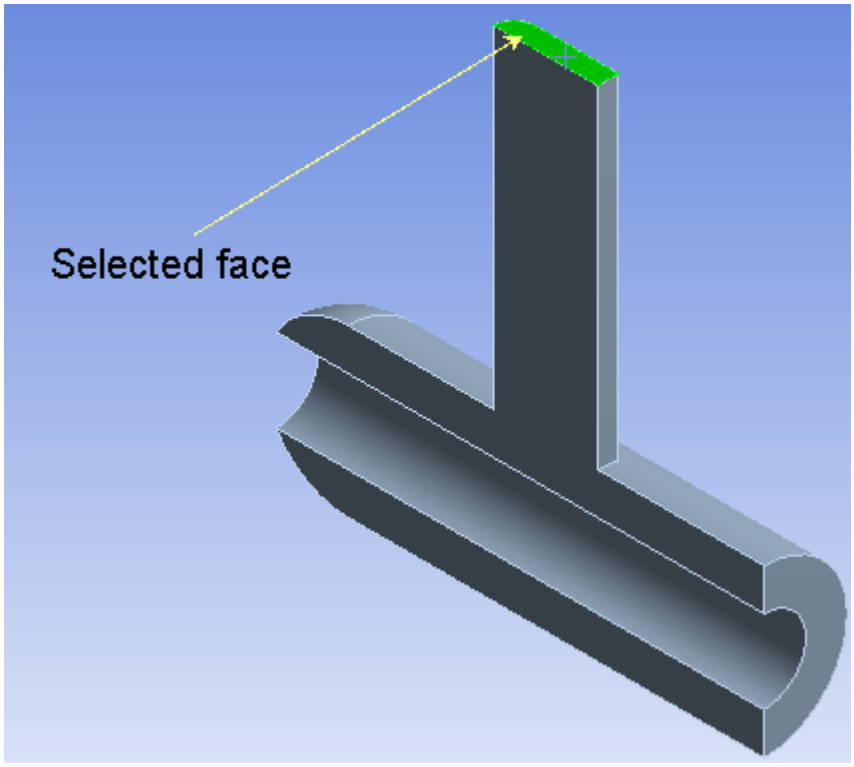


2. Right-click on **Static Structural (B5)**, in the **Model** tree.

   a. From the context menu select **Insert** > **Fixed Support**.

   b. Select the small face at the top of the geometry as shown in Figure 46: Selected Face for Fixed Support (p. 6) and click **Apply** in the **Details of Fixed Support** panel.

Figure 46: Selected Face for Fixed Support



3. In the tree expand **Static Structural (B5)** and right-click on **Imported Load (A5)** in the tree below it.

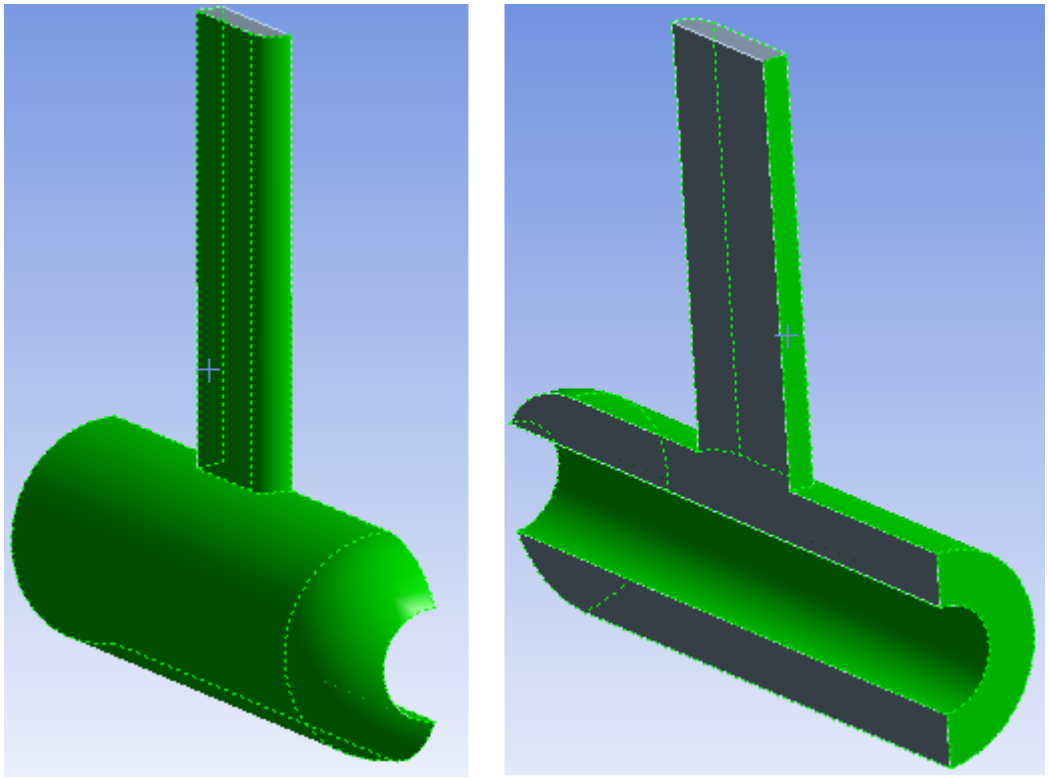
   a. From the context menu select **Insert** > **Pressure**.

   Note

   This is the load transferred from the Fluent solution.

   b. Select the remaining 7 surfaces of the probe as shown in Figure 47: Faces Selected for Imported Pressure (p. 7) and click **Apply** next to **Geometry** in the **Details of Imported Pressure** panel.

Figure 47: Faces Selected for Imported Pressure



c. Select **fsisurface** from the **CFD Surface** drop-down list.

d. Right-click on **Imported Pressure** in the tree and select **Import Load** from the context menu.

e. Expand the **Imported Pressure** entry in the tree and select the **Imported Load Transfer Summary** entry to review the mapping details.

CFD Load Transfer Summary

All values correspond to the CFD results before the application of any Scale or Offset operations set in Mechanical.

CFD Computed Forces from CFD Results File
E:\tutorials\FluentFSI\01_Probe_1way\Probe_1way_files\dp0\SYS\MECH\Solution\FFF-3-00081.dat.gz
X-component = 9.3962e-003 N
Y-component = 8.5928e-005 N
Z-component = -2.1648e-002 N

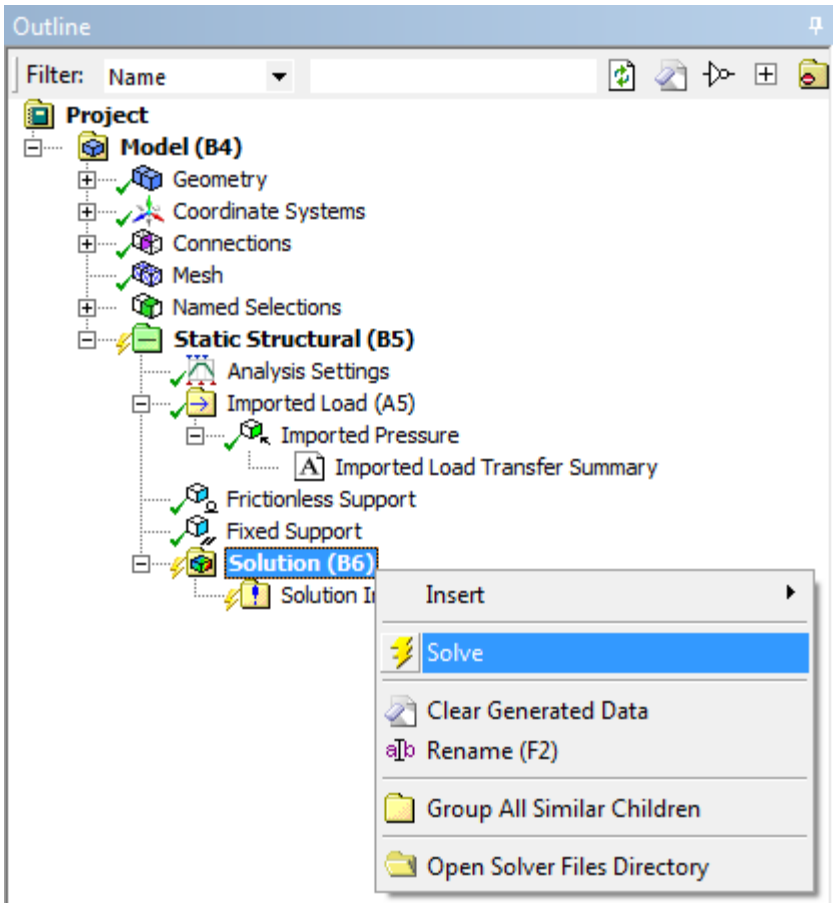
Mechanical Mapped Forces for Mechanical Surface File
E:\tutorials\FluentFSI\01_Probe_1way\Probe_1way_files\dp0\SYS\MECH\Import_ANSYS_50.cdb
X-component = 7.9227e-003 N
Y-component = 6.758e-004 N
Z-component = -2.4455e-002 N

100% of Mechanical nodes were mapped to the CFD surface.

Note

As you can see from the report due to the coarse meshes the CFD and mapped Mechanical forces do not accurately match.

f. Right-click **Solution (B6)** in the tree and select **Solve** from the context menu.



2.5. Results

1. Right-click **Solution (B6)** in the tree and select **Insert** > **Stress** > **Equivalent (von-Mises)** from context menu. See Figure 48: Equivalent (von-Mises) Stress Contours (p. 10).

2. Right-click **Solution (B6)** in the tree and select **Insert** > **Deformation** > **Total** from context menu. See Figure 49: Total Deformation Contours (p. 10).

3. Right-click **Solution (B6)** in the tree and select **Evaluate All Results** from context menu.

4. Examine the stress (Figure 48: Equivalent (von-Mises) Stress Contours (p. 10)) and deformation (Figure 49: Total Deformation Contours (p. 10)) results.

Figure 48: Equivalent (von-Mises) Stress Contours

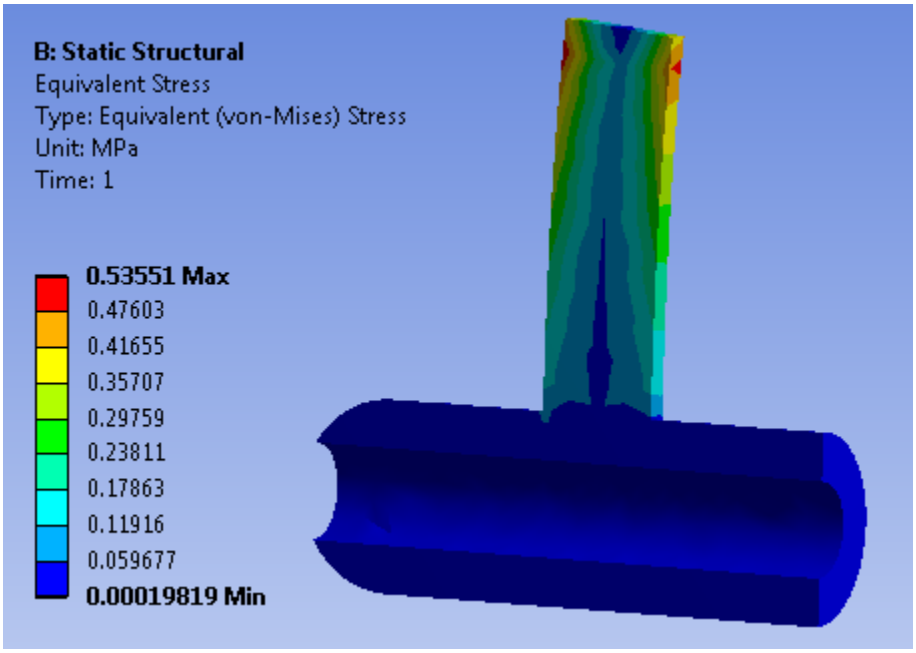
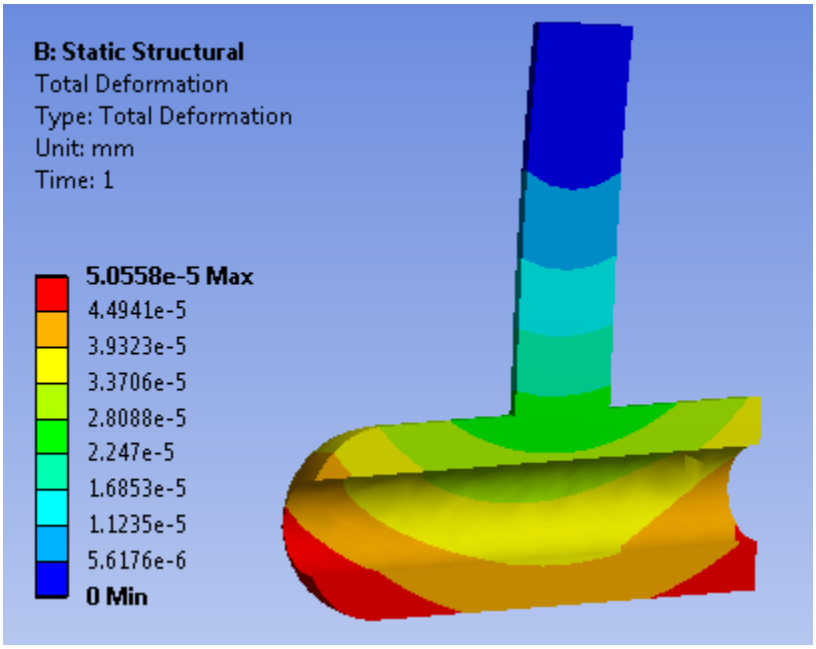


Figure 49: Total Deformation Contours



Note

The small deformations confirm that 1-way FSI is suitable. Given the small deformations, this case could have been solved with **Large Deflection = Off** in the **Mechanical Analysis Settings**.

3. Summary

You learned to set up a static structural case of a given a completed Fluent solution. You also learned to solve the case using an imported load.