Thermal Stress Analysis of a T Junction

This example considers a T junction that is subjected to thermal loads due to the mixing of two fluids with different temperatures, as well as fluid pressure.

In this tutorial you will learn how to calculate the stresses in the T junction due to thermal and pressure loads.

1. Problem Description

For calculating thermal and pressure loads on the T junction two approaches are considered:

- Setup and solution of a Steady-State Thermal case in Mechanical using a surface load from a Fluent solution. The thermal solution is then transferred to a Static Structural system and thermal stresses are calculated. This approach should be used when your Fluent solution only includes the fluid zones.
- Setup and solution of the Static Structural case using an imported force and body temperature directly from Fluent. This approach should be used when your Fluent solution includes the fluid and solid zones (CHT).

The Fluent solution provided in this workshop is a CHT case, but it will be used to demonstrate both workflows.

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 `T_Junction_1way.wbpz` to the working folder.

2.2. Starting Workbench

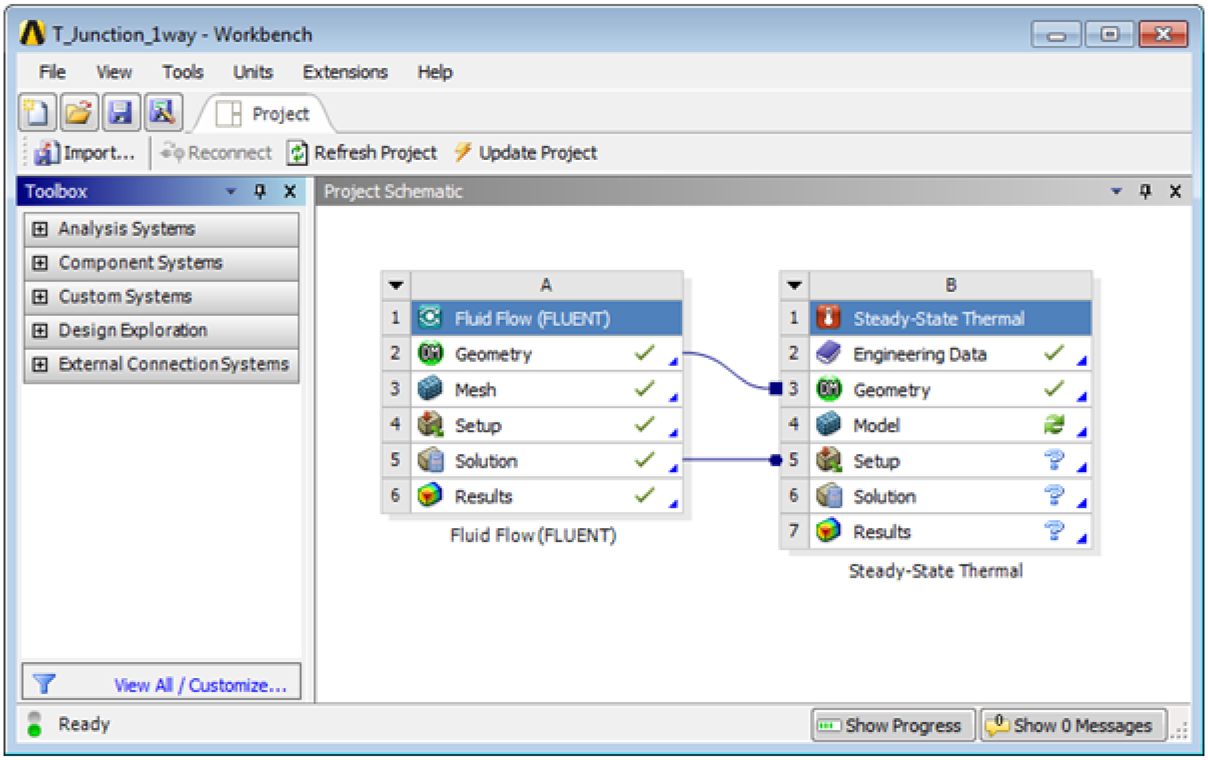
1. Start ANSYS Workbench and select **File >Restore Archive...** from the menu.
   a. Select `T_Junction_1way.wbpz` from your working folder.

b. Save to your working folder.

Note

The fluid flow solution for this exercise has already been completed. Review the solution in Fluent if you wish by editing the **Solution** (**A5**) or **Results** (**A6**) cells.

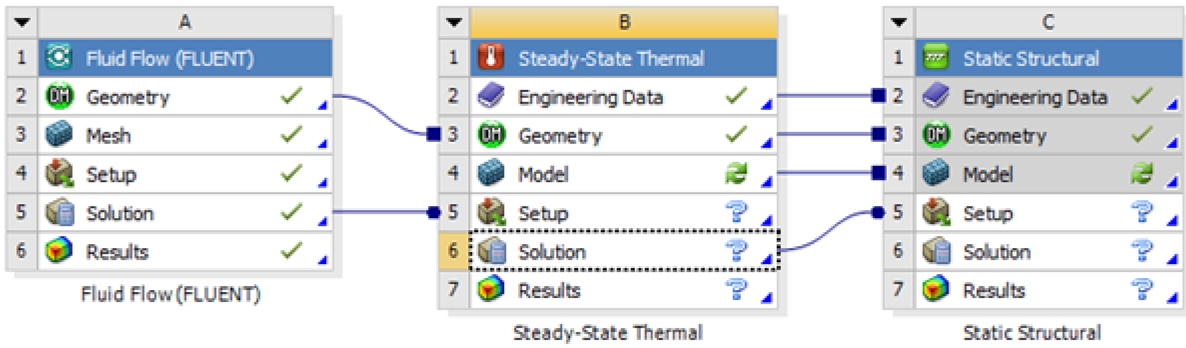
2. Drag a **Steady-State Thermal** analysis system onto the **Project Schematic** and drop it onto the **Solution** cell (**A5**) of the **Fluid Flow (FLUENT)** system.



Note

The links created share a common **Geometry** and transfer the Fluent **Solution** as a load to the **Steady-State Thermal Setup**. Surface temperature and convection coefficient will be available for import into the **Steady-State Thermal** system.

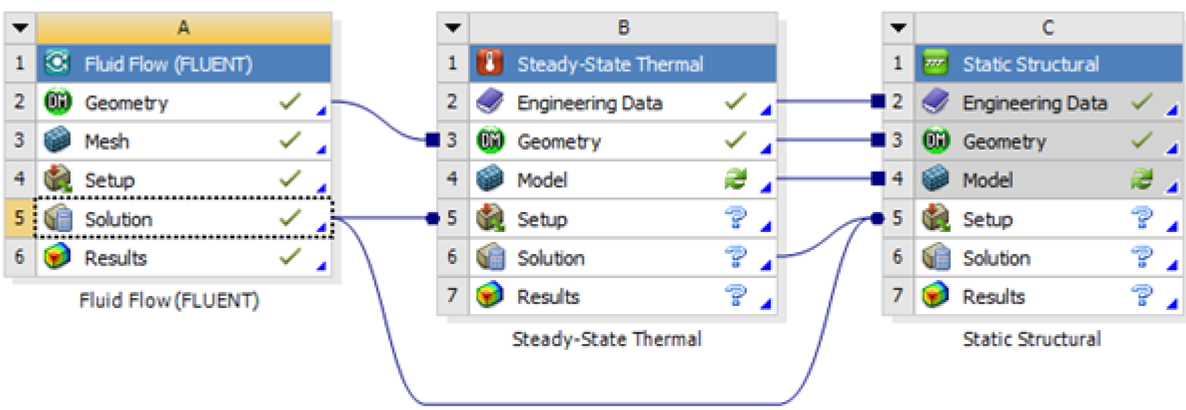
3. Right-click on the **Solution** cell (**B6**) of the **Steady-State Thermal** system and select **Transfer Data To New** > **Static Structural**.



Note

This will transfer the volumetric thermal solution to the structural model as an Imported Body Load. Recall this workflow is used when the Fluent results only contain the flow volume, so the **Steady-State Thermal** system is required to calculate the volumetric temperature in the solid before we can calculate thermal stresses.

4. Connect the **Solution** cell (**A5**) of the **Fluid Flow (FLUENT)** system to the **Setup** cell (**C5**) of the **Static Structural** system.



Note

Now both the fluid pressure from Fluent and the thermal solution from Mechanical are made available to the **Static Structural** model.

2.3. Engineering Data Setup

Here you will create a material for use in Mechanical that has the same physical properties as the material used for the CHT calculation in Fluent.

1. Double-click the **Engineering Data** cell (**B2**) of the **Steady-State Thermal** system.

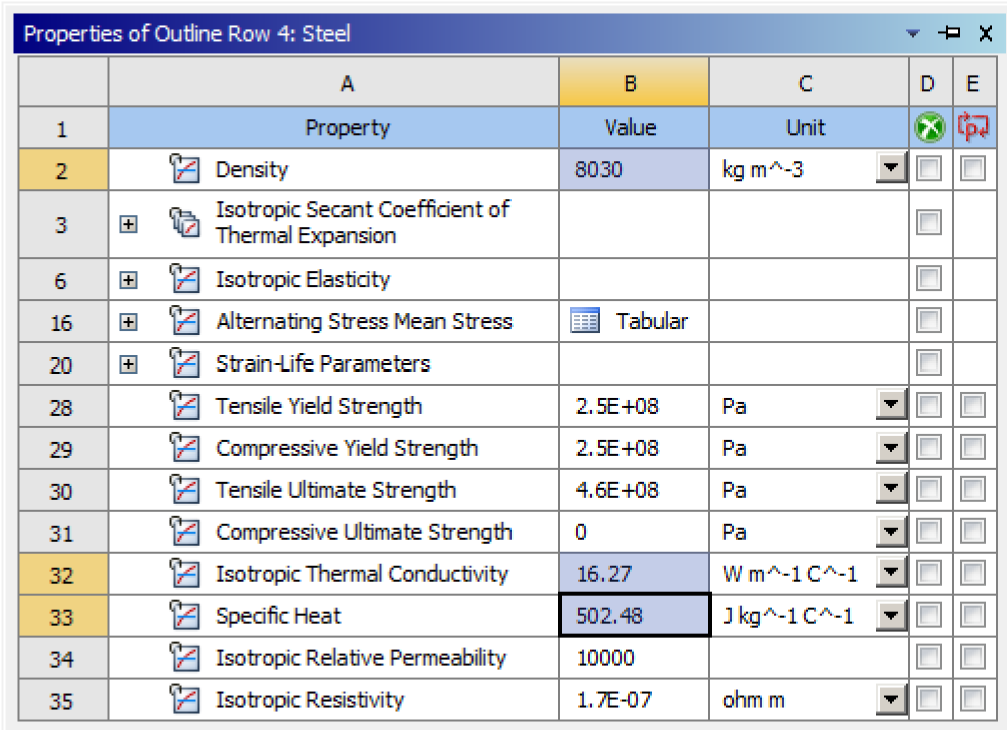
2. In the **Engineering Data** tab that opens right-click on **Structural Steel** in the **Outline of Schematic** and select **Duplicate**.

3. Rename **Structural Steel 2** that is created to `Steel`.

4. Toggle the button **Filter Engineering Data** 

in the toolbar to display all material properties in the **Properties** panel.

5. Select **Steel** in **Outline** and in the **Properties** panel enter `8030` **kg m^–3** for **Density**.



Properties of Outline Row 4: Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	8030	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
16	Alternating Stress Mean Stress	Tabular			
20	Strain-Life Parameters				
28	Tensile Yield Strength	2.5E+08	Pa		
29	Compressive Yield Strength	2.5E+08	Pa		
30	Tensile Ultimate Strength	4.6E+08	Pa		
31	Compressive Ultimate Strength	0	Pa		
32	Isotropic Thermal Conductivity	16.27	W m^-1 C^-1		
33	Specific Heat	502.48	J kg^-1 C^-1		
34	Isotropic Relative Permeability	10000			
35	Isotropic Resistivity	1.7E-07	ohm m		

6. Enter `16.27` **W m^–1 C^–1** for **Isotropic Thermal Conductivity**.

7. Enter `502.48` **J kg^–1 C^–1** for **Specific Heat**.

8. Close the **Engineering Data** tab and return to the **Project Schematic**.

2.4. Thermal Setup

1. Double-click on the **Model** cell (**B4**) of the **Steady-State Thermal** system.

   This will launch Mechanical.

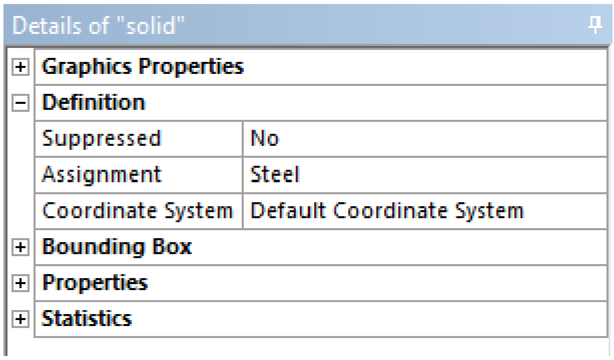
2. In the tree expand **Geometry**.

3. Right-click on **fluid** and select **Suppress Body**.

4. Select **NutsBolts** from the tree under **Geometry**.

   - In the **Details of NutsBolts** panel select **Steel** for **Assignment**.

5. Similarly for **solid** under **Geometry** select **Steel** for **Assignment**.



Details of "solid"	
Graphics Properties	
Definition	
Suppressed	No
Assignment	Steel
Coordinate System	Default Coordinate System
Bounding Box	
Properties	
Statistics	

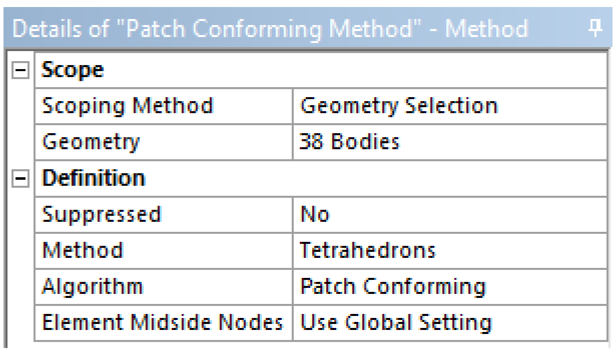
6. Select **Mesh** in the tree.

   a. In the **Details of Mesh** panel select **Medium** from the **Relevance Center** drop-down list under **Sizing**.

   b. Right-click on **Mesh** in the tree and select **Insert** > **Method**.

   c. In the graphics window right-click and click on **Select All**.

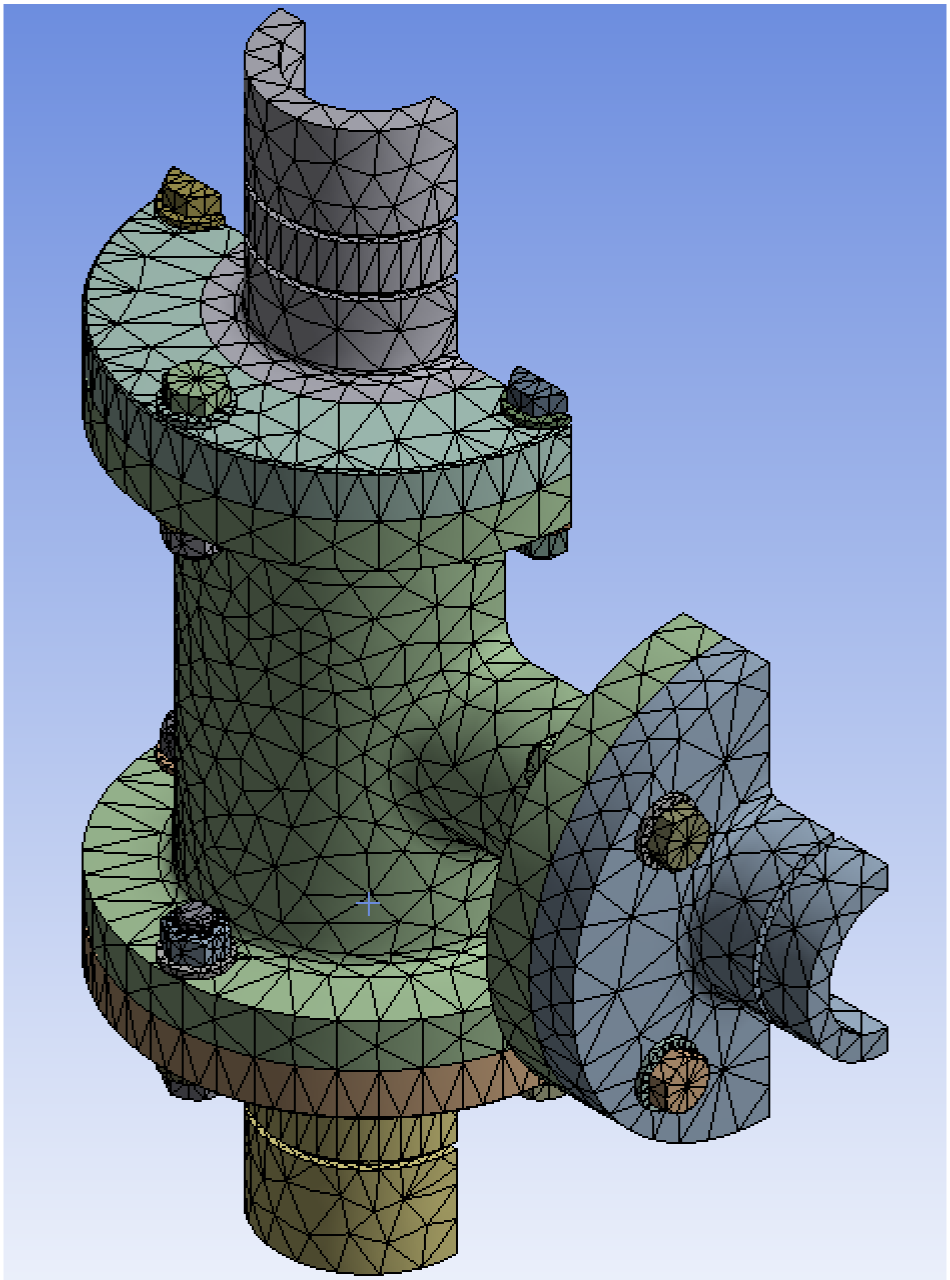
   d. In the **Details** panel click on **Apply** next to **Geometry**.



Details of "Patch Conforming Method" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	38 Bodies
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Conforming
Element Midside Nodes	Use Global Setting

   e. Select **Tetrahedrons** from the **Method** drop-down list.

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



7. In the tree right-click **Steady-State Thermal (B5)** and select **Insert** > **Temperature** from the context menu.

    a. In the **Details of Temperature** panel select **Named Selection** from the **Scoping Method** drop-down list.

    b. Select **wall_cold_end** from the **Named Selection** drop-down list.

    c. Enter `20` °C for **Magnitude**.

    d. Similarly set a temperature of `80` °C for **wall_hot_end**.

Details of "Temperature 2"	
Scope	
Scoping Method	Named Selection
Named Selection	wall_hot_end
Definition	
Type	Temperature
Magnitude	80. °C (ramped)
Suppressed	No

8. In the tree right-click **Steady-State Thermal (B5)** and select **Insert** > **Convection** from the context menu.

Details of "Convection"	
Scope	
Scoping Method	Named Selection
Named Selection	convection
Definition	
Type	Convection
Film Coefficient	10. W/m²·°C (ramped)
Ambient Temperature	15. °C (ramped)
Convection Matrix	Program Controlled
Suppressed	No

   a. In the **Details of Convection** panel select **Named Selection** from the **Scoping Method** drop-down list.

   b. Select **convection** from the **Named Selection** drop-down list.

   c. Enter 10 W/m ^{2}·°C for **Film Coefficient**.

   d. Enter 15°C for **Ambient Temperature**.

9. Right-click on **Imported Load (A5)** in the tree and select **Insert** > **Temperature** from the context menu.

Details of "Imported Temperature"	
Scope	
Scoping Method	Named Selection
Named Selection	interface_solid_side
Definition	
Type	Imported Temperature
Tabular Loading	Program Controlled
Suppressed	No
Transfer Definition	
CFD Surface	interface_solid_side
CFD Data	

   a. In the **Details of Imported Temperature** panel select **Named Selection** from the **Scoping Method** drop-down list.

   b. Select **interface_solid_side** from the **Named Selection** drop-down list.

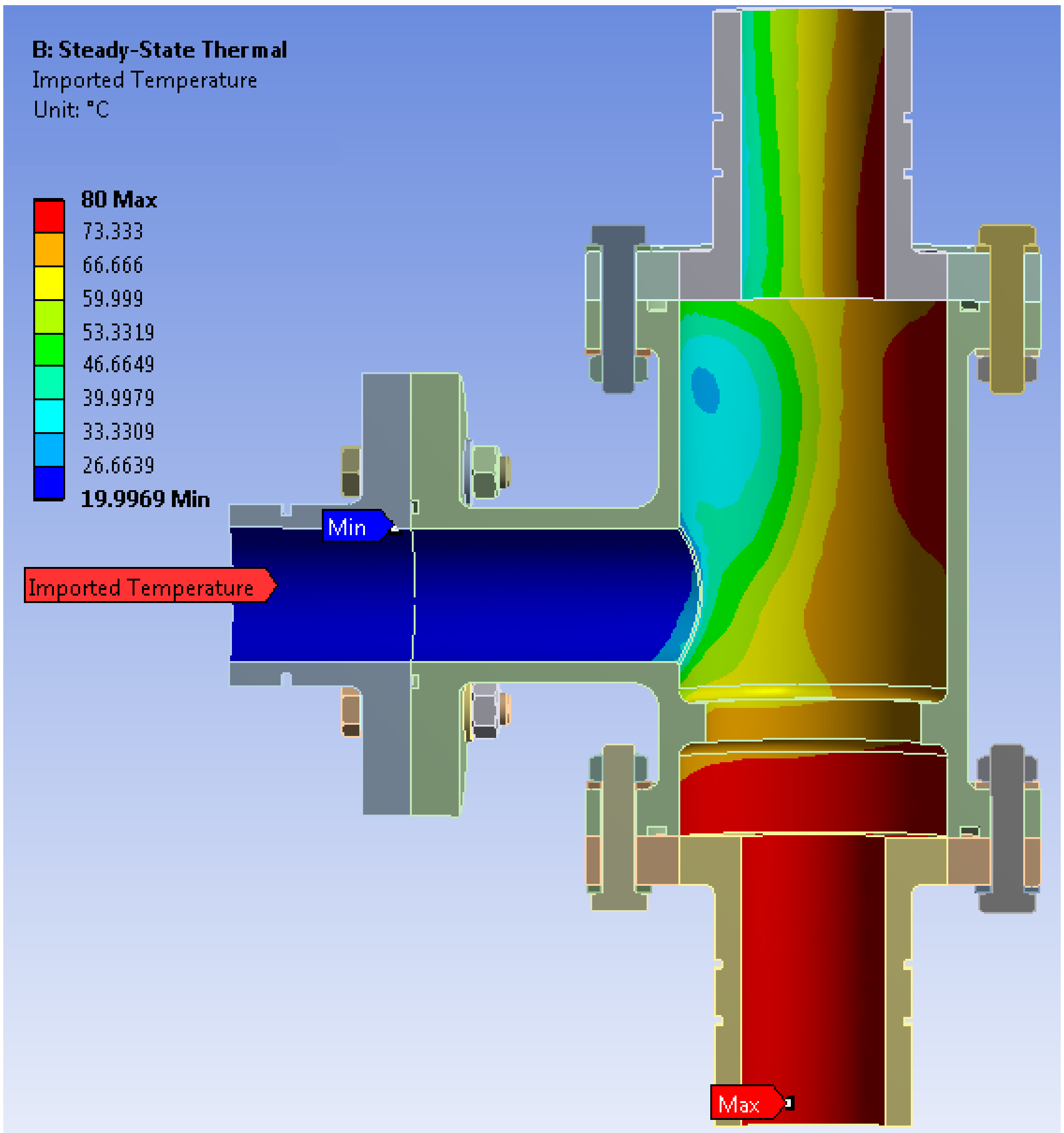
c. Select **interface_solid_side** from the **CFD Surface** drop-down list.

Note

If you only had access to flow region data in Fluent you would normally import a convection coefficient. In this case you have a solid-side temperature field in Fluent, so you can apply the solid surface temperature as a constraint in Mechanical. Be careful to never apply the fluid temperature as a solid temperature constraint.

---

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



The temperature will be shown in the graphics window.

11. Expand **Imported Temperature** in the tree and click on **Imported Load Transfer Summary** below it.

Check that all nodes were mapped and the **Average Temperature** of CFD computed data and Mechanical mapped data are similar.

12. Right-click on **Steady-State Thermal (B5)** in the tree and select **Solve**.
13. Right-click **Solution (B6)** in the tree and select **Insert** > **Thermal** > **Temperature**. See Figure 53: Steady-State Thermal Temperature (p. 9).
14. Right-click **Solution (B6)** in the tree and select **Insert** > **Thermal** > **Total Heat Flux**. See Figure 54: Steady-State Thermal Total Heat Flux (p. 10).
15. Right-click **Solution (B6)** in the tree and select **Evaluate All Results**.

Figure 53: Steady-State Thermal Temperature

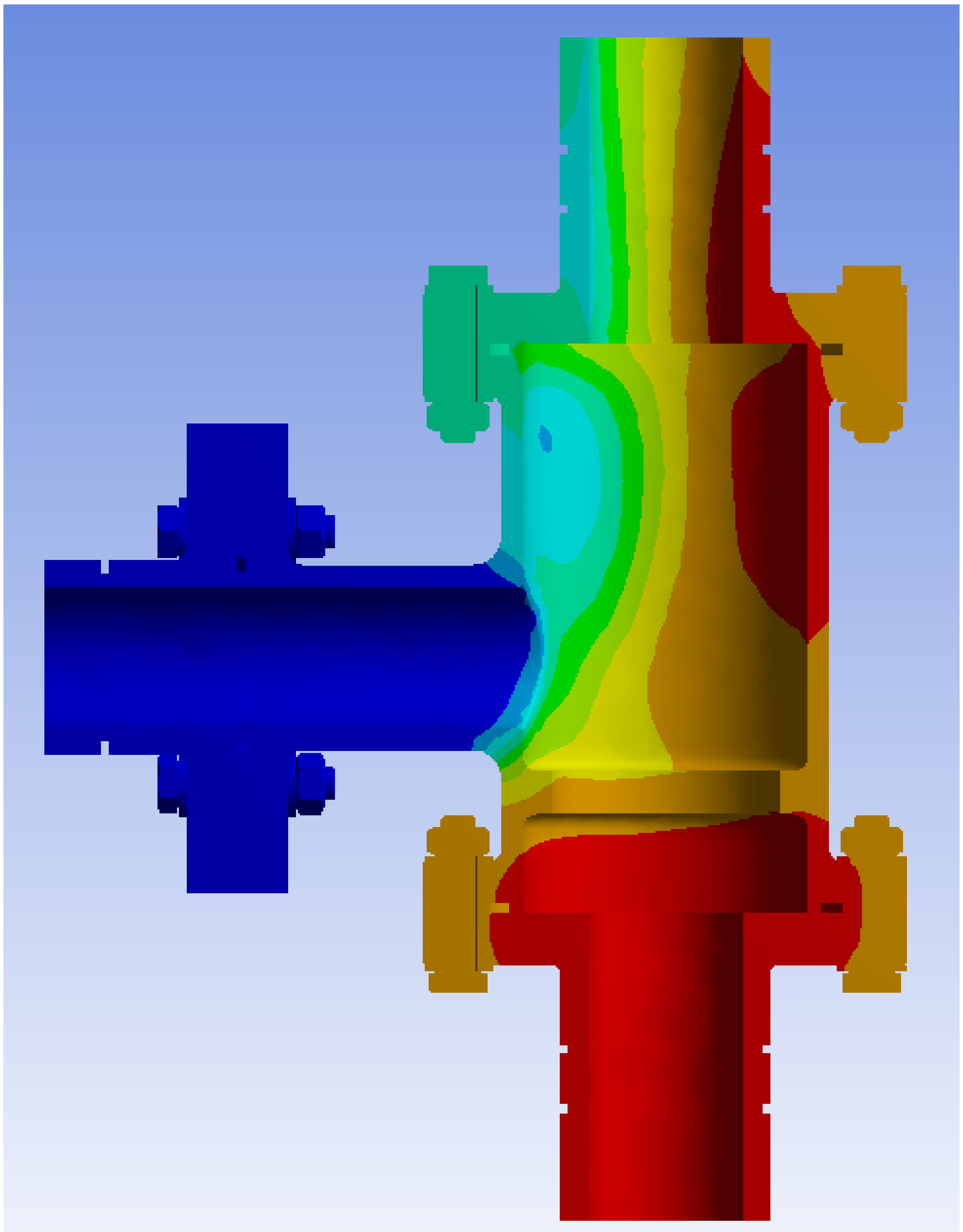
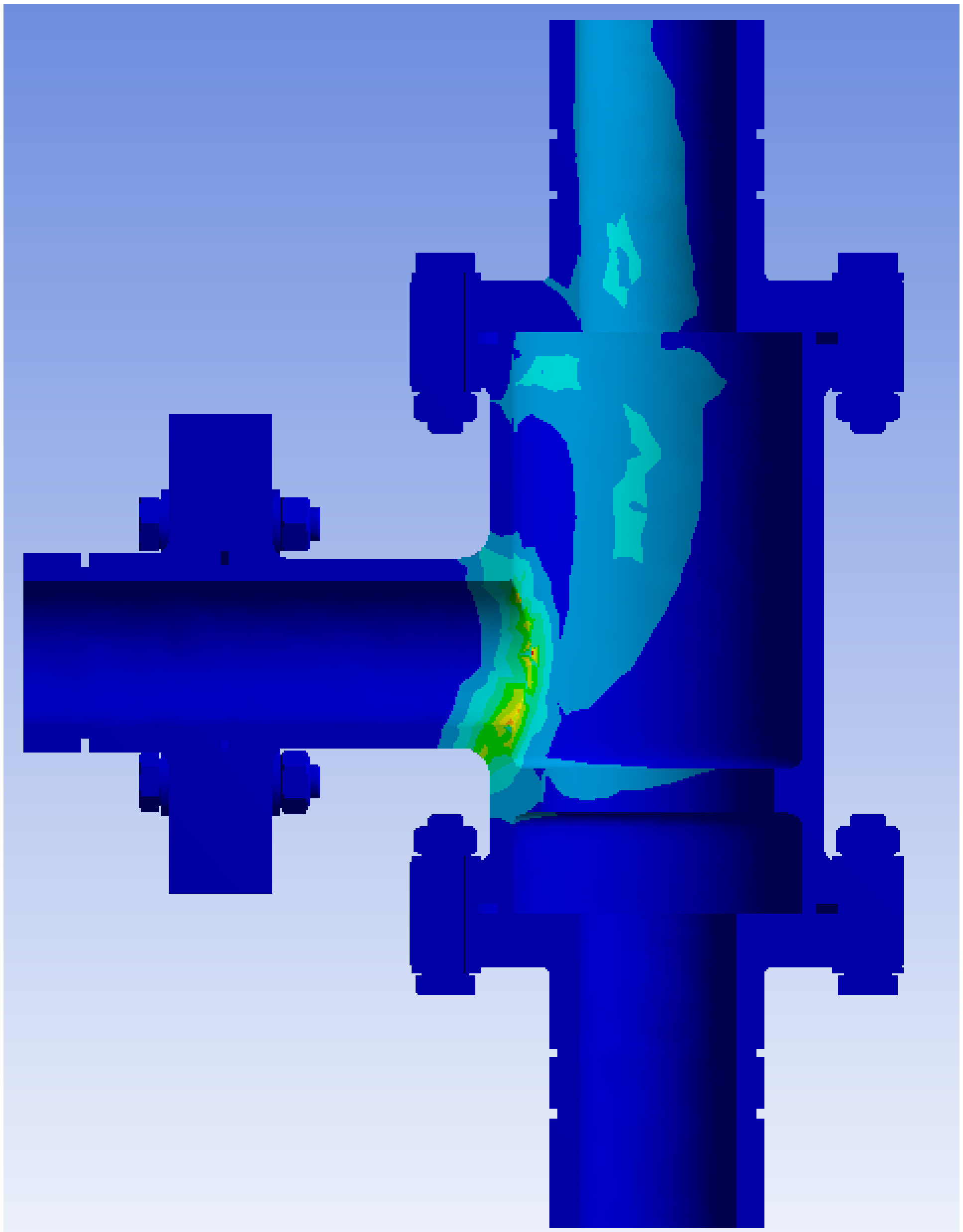


Figure 54: Steady-State Thermal Total Heat Flux



16. Save the project.

2.5. Structural Setup

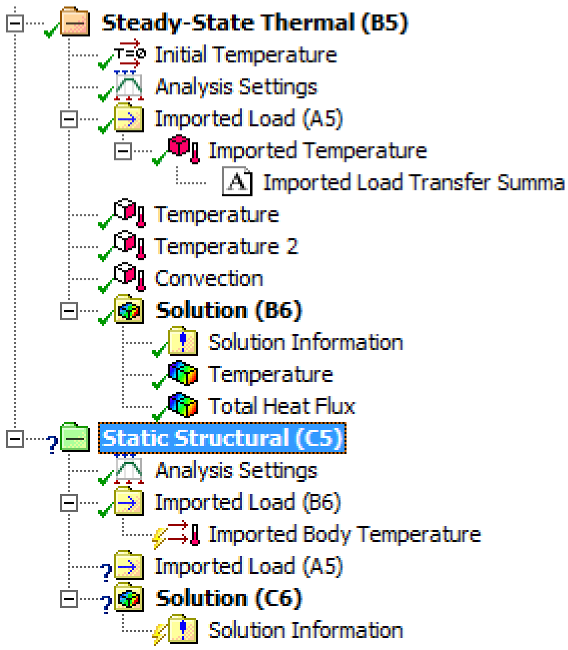
1. Return to the **Project Schematic** and double-click on the **Static Structural** cell (**C5**), in the **Model** tree.

   Note

   The **Steady-State Thermal** system is now fully updated.

   ---

2. In the Mechanical window note the **Imported Load** objects under **Static Structural (C5)**.

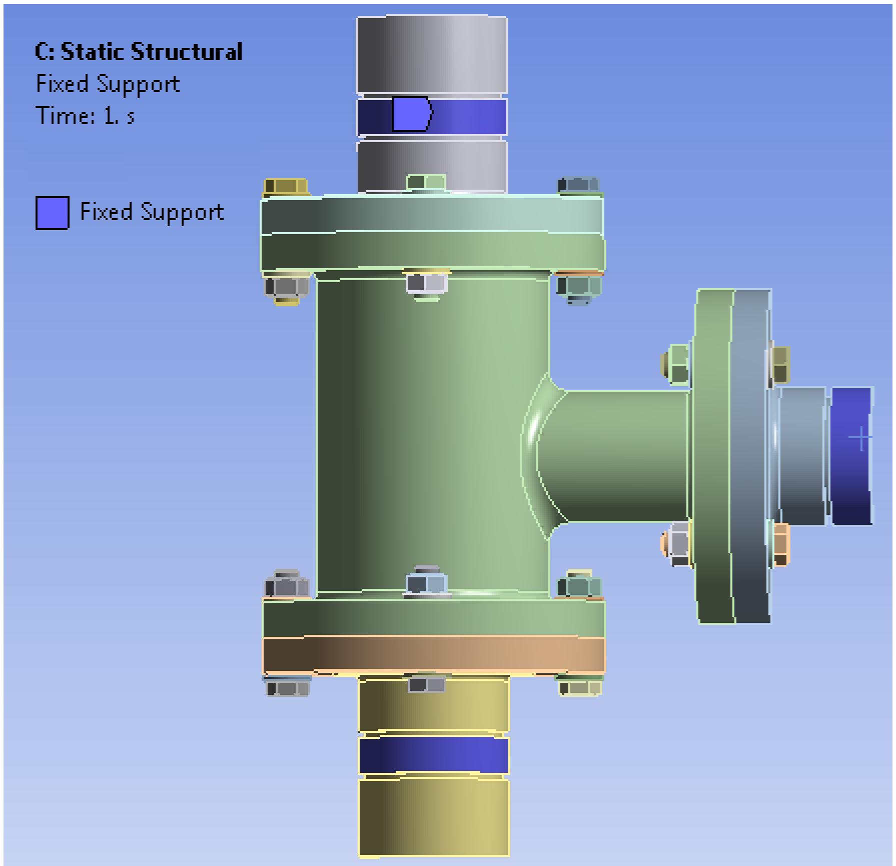


The temperature load from the previously solved thermal model is automatically transferred as **Imported Load (B6)**.

3. In the tree right-click **Static Structural (C5)** and select **Insert** > **Fixed Support**.

   - Select the three surfaces highlighted in Figure 55: Surfaces Selected for Fixed Support (p. 12) and click **Apply** next to **Geometry** in the **Details of Fixed Support** panel.

Figure 55: Surfaces Selected for Fixed Support



4. In the tree right-click **Imported Load (A5)** and select **Insert** > **Pressure**.

   a. In the **Details of Imported Pressure** panel select **Named Selection** from the **Scoping Method** drop-down list.

   b. Select **interface_solid_side** from the drop-down list.

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

   Note

   This is the fluid side of the interface (force does not exist on the solid side of the interface).

   ---

   d. Right-click on **Imported Pressure** and select **Import Load** from the context menu to import the load and then examine the **Imported Load Transfer Summary**.

5. Right-click **Static Structural (C5)** and select **Solve** from the context menu.

6. In the tree right-click **Solution (C6)** and select **Insert** > **Deformation** > **Total**. See Figure 56: Total Deformation (p. 13).

7. In the tree right-click **Solution (C6)** and select **Insert** > **Stress** > **Equivalent (von-Mises)**. See Figure 57: Equivalent Stress (p. 14).

8. In the tree right-click **Solution (C6)** and select **Evaluate All Results**.

Figure 56: Total Deformation

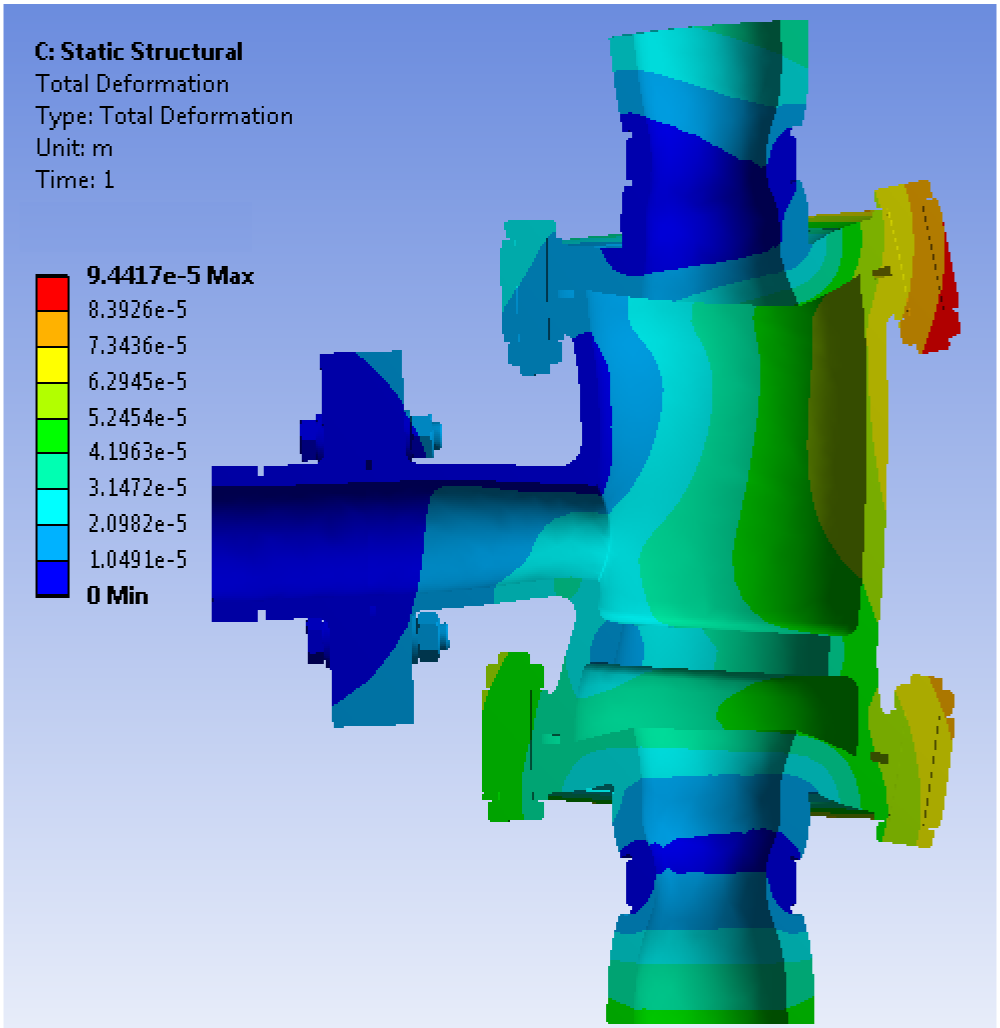
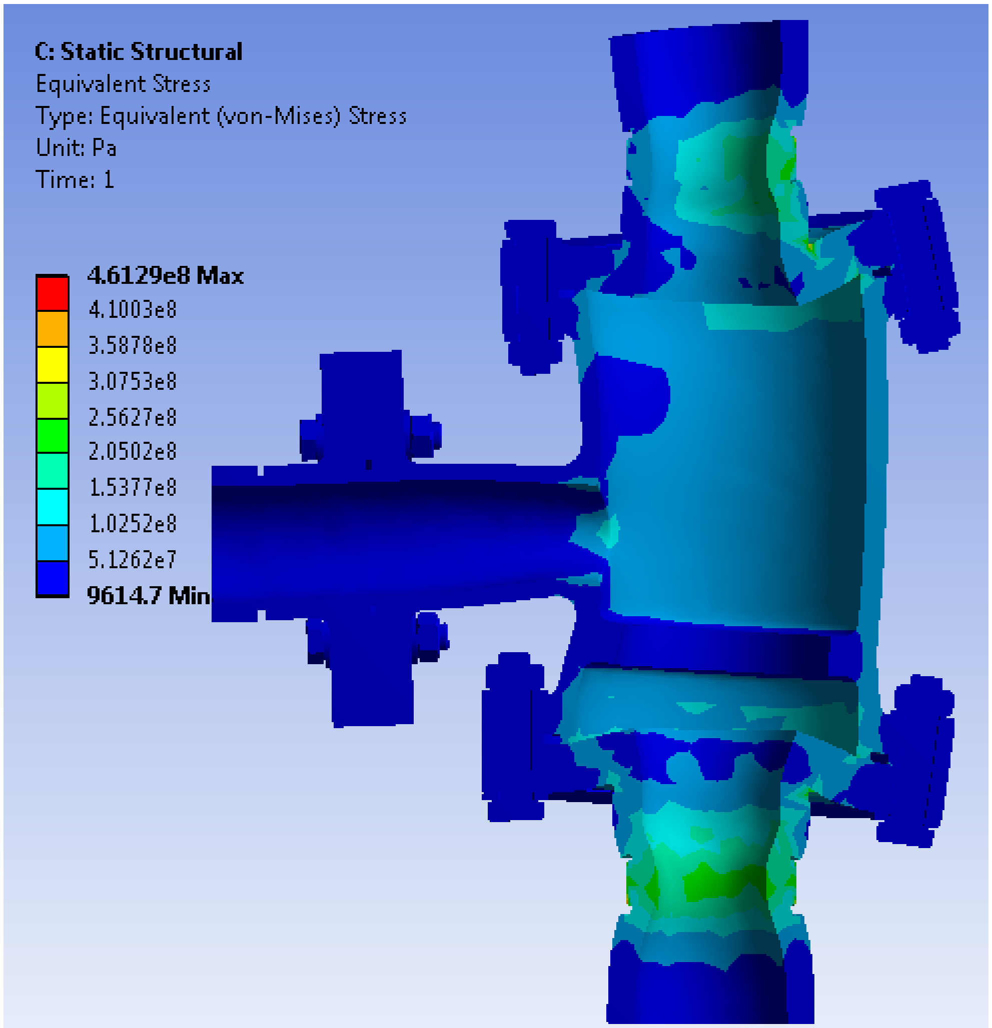


Figure 57: Equivalent Stress

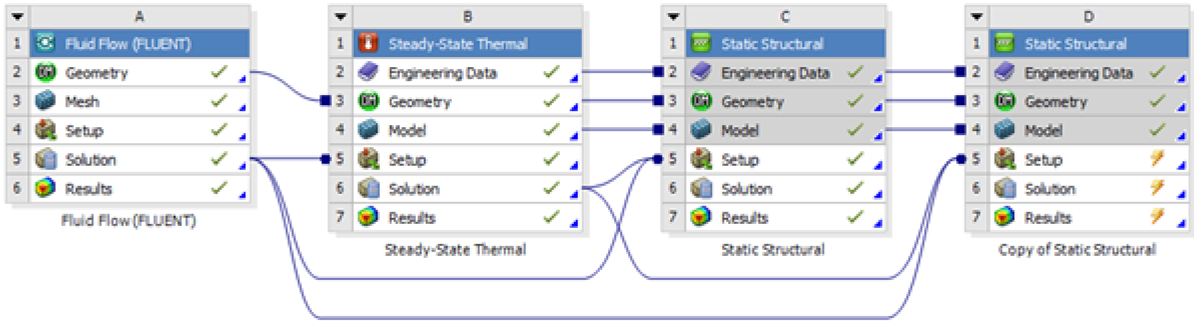


9. After examining the results close Mechanical, return to the **Project Schematic** and save the project.

2.6. Using a Body Temperature Import

Rather than importing a surface temperature from Fluent, re-solving the solid temperature in Mechanical and then transferring the temperature to a structural system, you can import the volumetric temperature directly from Fluent to a structural system (if you solved the solid temperature in Fluent). This avoids the need for a steady-state thermal system. You will complete this alternative workflow next.

1. Right-click the **Setup** cell (**C5**) of the **Static Structural** system and select **Duplicate** from the context menu.

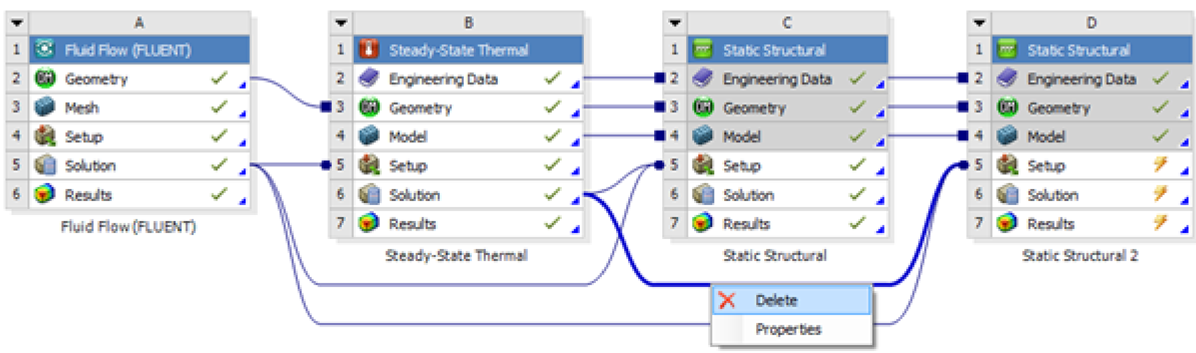


Note

Duplicating from the **Setup** cell retains the upstream links and shares the **Engineering Data**, **Geometry** and **Model** cells.

2. Re-name the new system `Static Structural 2`.

3. Select the link between the **Steady State Thermal** system and the **Static Structural 2**, right-click, and then select **Delete** from the context menu.



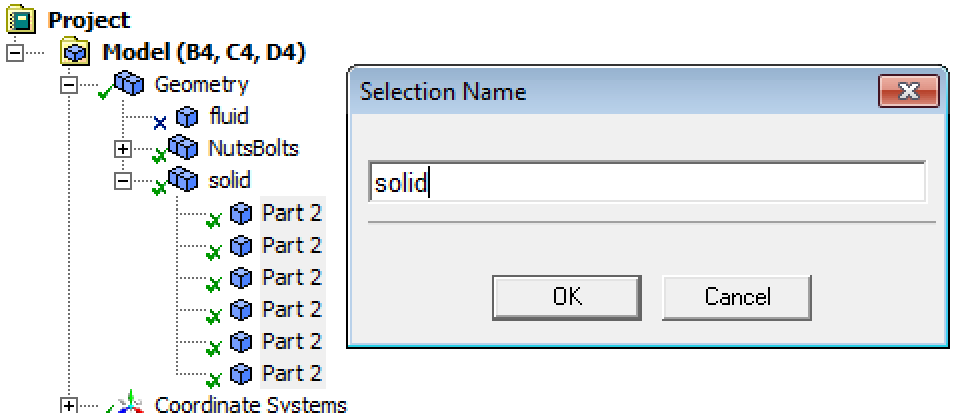
- Click **OK** in the warning dialog box that appears.

4. Double-click the **Setup** cell (**D5**) of the **Static Structural 2** system to open Mechanical, selecting **Yes** when asked if you want to read the upstream data.

Note

The **Imported Pressure** load and **Fixed Support** already exist from the previous structural analysis due to duplication. You must now add the **Imported Body Temperature** load. Start by creating a **Named Selection** for the solid parts. You will use this when importing the temperature load.

a. In the tree, expand **Geometry**, and under that expand **solid**.



i. Select all six bodies named **Part 2**.

ii. Right-click and select **Create Named Selection** from the context menu.

iii. In the **Selection Name** dialog box enter the name `solid`.

Note

Temperature body load will not be applied to the **NutsBolts** part here.

---

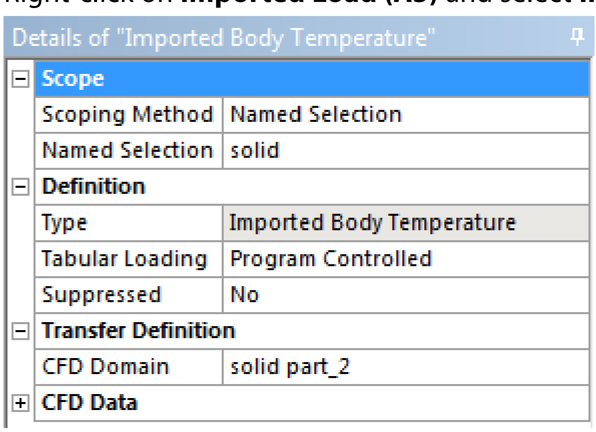
b. Scroll down in the tree to locate **Static Structural 2 (D5)** and click on **Imported Load (A5)** under it.

Note

Under **Imported Load (A5)** you can see that the pressure load is already imported.

---

c. Right-click on **Imported Load (A5)** and select **Insert** > **Body Temperature** from the context menu.

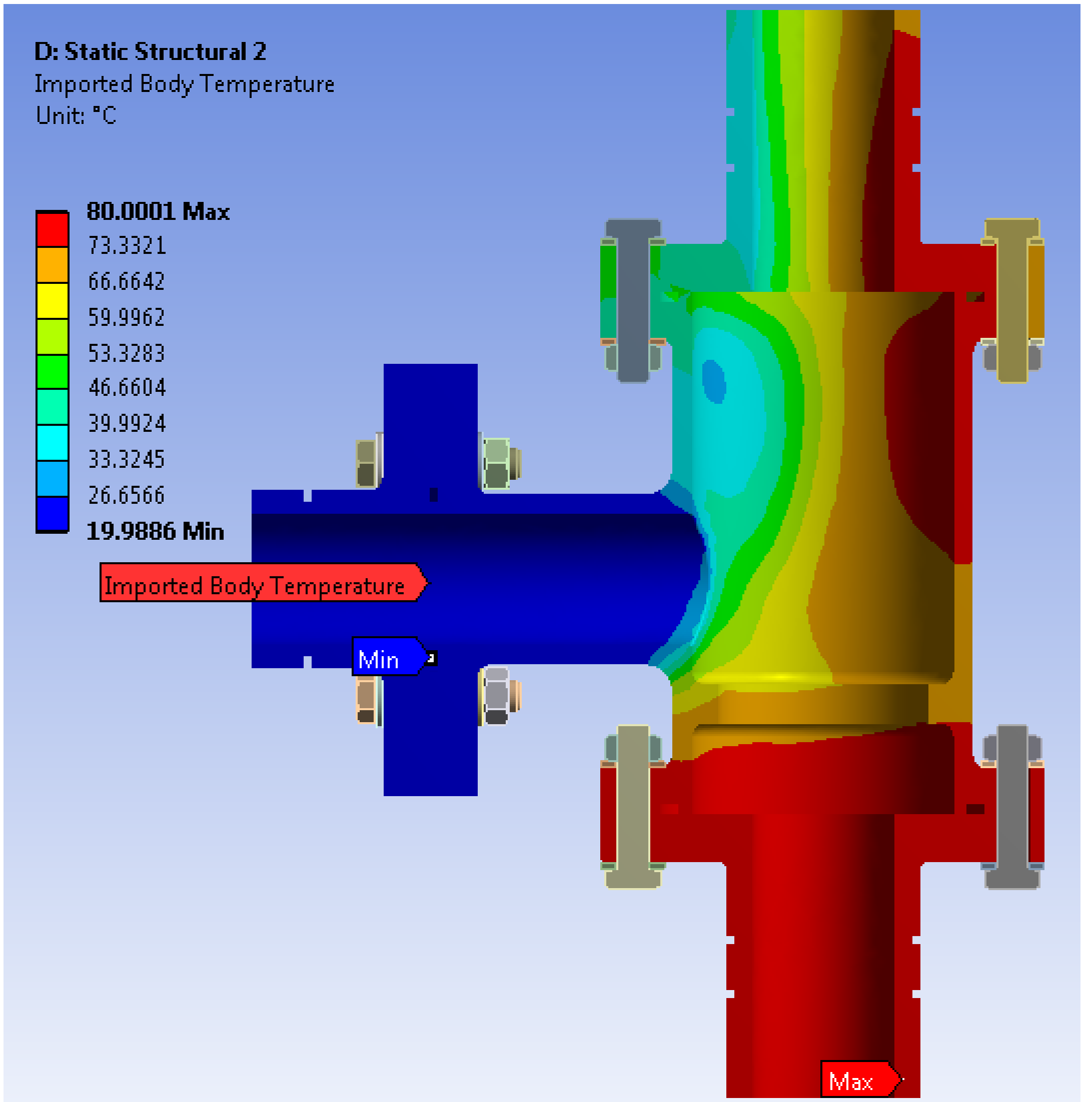


i. In the **Details of Imported Body Temperature** panel select **Named Selection** from the **Scoping Method** drop-down list.

ii. Select **solid** from the **Named Selection** drop-down list.

iii. Select **solid part_2** from the **CFD Domain** drop-down list.

d. Right-click **Imported Body Temperature** and select **Import Load** from the context menu.



e. Examine the **Imported Load Transfer Summary** after the import is complete.

f. Right-click **Static Structural 2 (D5)** and select **Solve** from the context menu.

g. Compare your results with the previous approach, then save and close the project.

2.7. Summary

This workshop has demonstrated two approaches to calculate thermal stresses using CFD results. If a CHT solution is available in Fluent then simply pass the volumetric temperatures to a structural system.

If the solid temperatures are not available in Fluent then thermal data from the fluid wall boundary conditions in Fluent is passed to a thermal system in Mechanical to first calculate the volumetric temperature field in the solid. Typically a heat transfer coefficient is sent from Fluent to the thermal system. Since this is a 1–way thermal coupling then the temperature solution in Mechanical should be compared to the thermal boundary conditions that were assumed in Fluent. If there is a large difference then you should consider an approach that couples the solid and fluid temperature fields more tightly, such as a CHT solution in Fluent or a 2–way thermal coupling between Fluent and Mechanical.