2-Way Thermal FSI for an Exhaust Manifold

This tutorial shows how to solve a 2-way steady-state thermal application.

1. Problem Description

This example considers the fluid and thermal solution for an exhaust manifold containing hot exhaust gas flow. The solid temperature field is solved in Mechanical using a Steady State Thermal system, which is coupled to Fluent via System Coupling. This case could be solved entirely in Fluent, but there are times when solving the solid temperature field in Mechanical is desirable. For example the mesh quality requirements are much less stringent in Mechanical, or an existing model may already be available in Mechanical.

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 `Manifold.wbpz` to the working folder.

2.2. Starting Workbench

1. Start ANSYS Workbench and select **File >Restore Archive...** from the menu.

    a. Select `Manifold.wbpz` from your working folder.
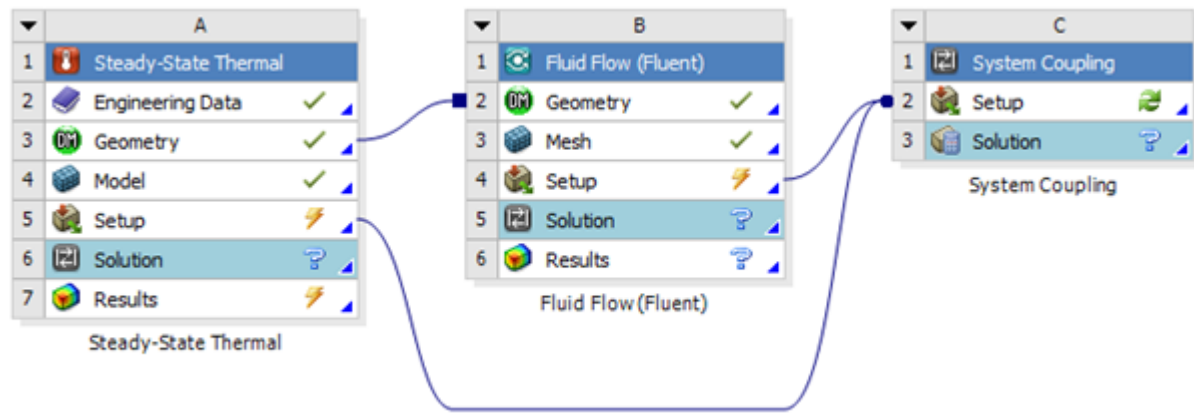
    b. Save to your working folder.

    > **Note**
    >
    > The stand-alone Fluent and Mechanical setups are complete, but no coupling settings have been completed.

2. Drag a **System Coupling** component system onto the **Project Schematic**.

3.   Connect the **Setup** cell (**A5**) of **Steady-State Thermal** system and the **Setup** cell (**B4**) of **Fluid Flow (FLUENT)** system to the **Setup** cell (**C2**) of **System Coupling** system.

2.3. Mechanical Setup

1.   In the **Project Schematic** window double-click on the **Setup** cell (**A5**) of the **Steady State Thermal** system.
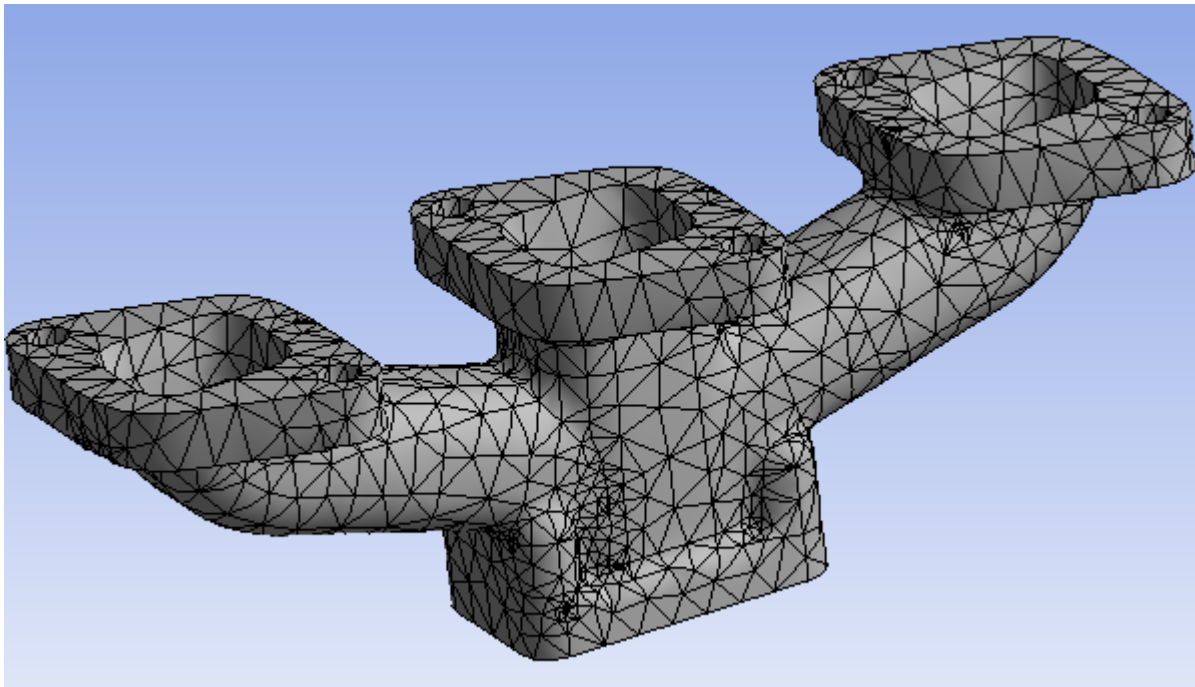
> **Note**
>
> This will launch ANSYS Mechanical.

2.   Check the current setup.

> **Note**
>
> You will notice the following:

- The **fluid** body has been suppressed.

- **Virtual Topology** has been used to simplify the geometry and produce a more uniform mesh.

- A named selection is available for the region that will connect to **System Coupling**.

- **Convection** and **Radiation** conditions have been applied to the outer surfaces of the geometry.
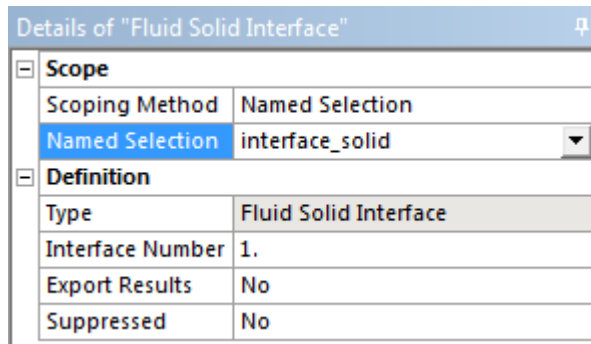
3. In the tree click on **Analysis Settings**.

Details of "Analysis Settings"	⊉
⊟ **Step Controls**	
Number Of Steps	1.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Off
Define By	Substeps
Number Of Substeps	1.
⊟ **Solver Controls**	
Solver Type	Program Controlled
Solver Pivot Checking	Program Controlled
⊞ **Radiosity Controls**	
⊞ **Nonlinear Controls**	
⊞ **Output Controls**	
⊞ **Analysis Data Management**	
⊞ **Visibility**	

a. In the **Details of Analysis Settings** panel select **Off** for **Auto Time Stepping**.

b. Ensure that **Define By** is set to **Substeps**.

c. Ensure that **Number of Substeps** is set to **1**.

> **Note**
>
> These are the standard settings for an FSI analysis. **Auto Time Stepping** or substepping should only be used in an FSI case if the Mechanical solver cannot converge it's solution when reasonable loads are provided.

4. Right-click on **Steady-State Thermal (A5)** in the tree and click on **Insert** > **Fluid Solid Interface** from the context menu.

Details of "Fluid Solid Interface"	📌
Scope	
Scoping Method	Named Selection
Named Selection	interface_solid ▾
Definition	
Type	Fluid Solid Interface
Interface Number	1.
Export Results	No
Suppressed	No

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

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

c. Save the project and close Mechanical.

2.4. Fluent Setup

1. In the **Project Schematic** double-click on **Setup** cell (**B4**).

2. Check the setup in Fluent.

    • If you click on **Models** in the tree you will see that the **Energy** option is **On** and the **Standard k-e** turbulence model is used.

    • Under **Materials** you can see that **air** with the **ideal-gas** equation of state is selected.

    • Under **Boundary Conditions** you can see that three mass flow inlet boundary conditions have been created with a flow rate of **0.15 kg/s** and a temperature of **1200 K**. The outlet is pressure specified with a relative pressure of **0 Pa**.

    • Under **Monitors** an outlet mass flow monitor has been created. The **Residual Monitors** using the local scaling option with a criterion of **5e-5**. The default criterion is 1e-5; this is a tight default convergence tolerance which cannot always be achieved. A criterion of **5e-5** still represents good convergence.

3. In the **Boundary Conditions** task page select **interface_fluid** and click **Edit...** to open the **Wall** dialog box.
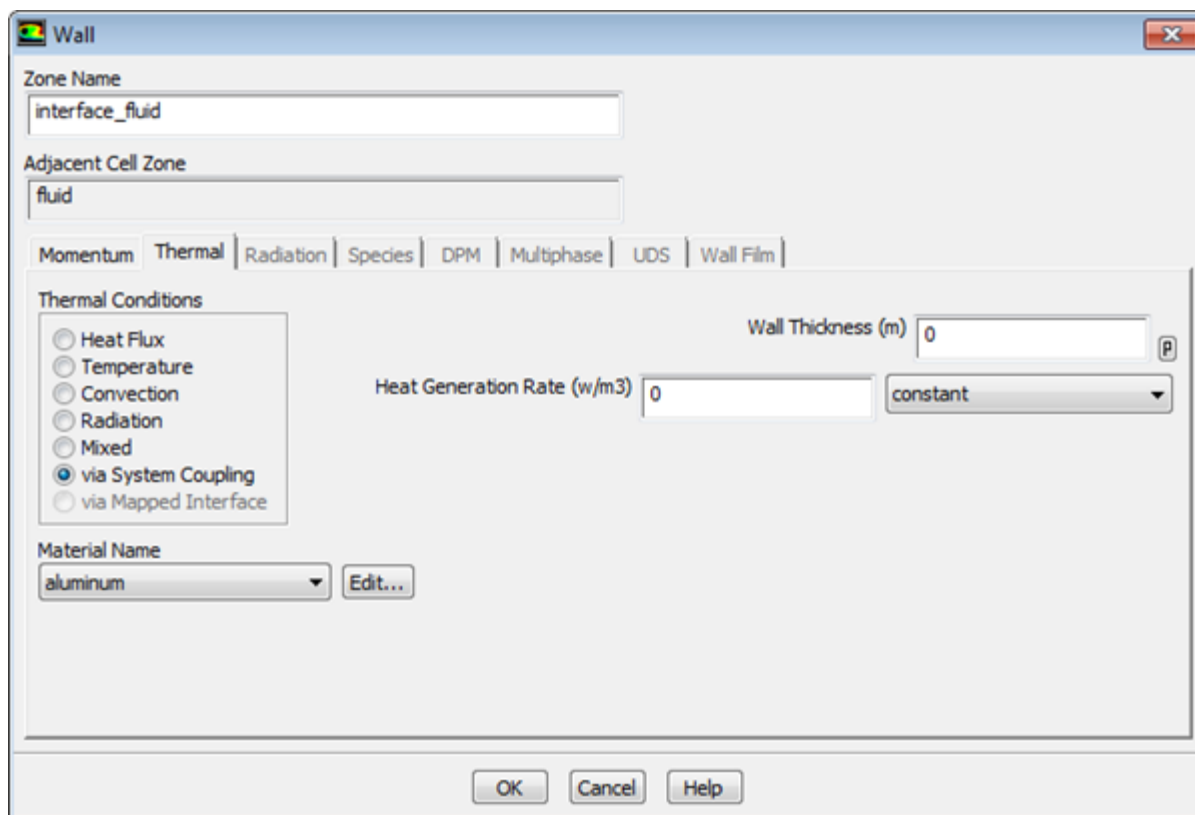
a. In the **Thermal** tab select **via System Coupling** from the **Thermal Conditions** group box.

b. Click **OK** to close the **Wall** dialog box.

4. Select **Run Calculation** in the tree.

• In the **Run Calculation** dialog box enter 25 for **Number of Iterations**.

Note

For a stand-alone steady state Fluent case 500 iterations is a reasonable value. When Fluent is connected to System Coupling this represents the number of Fluent iterations per Coupling Iteration, so a smaller number of iterations is suitable. The goal is not to fully converge Fluent within each Coupling Iteration, but instead enough Fluent iterations should be performed to allow the Fluent fields to react to the new boundary conditions from Mechanical/System Coupling (new wall Temperatures in this case) and move towards sensible values.

---

5. Click the **Synchronize WB cell status** icon [icon] from the toolbar.

2.5. System Coupling Setup

1. Return to the Workbench window.

2. In the **Project Schematic** right-click on the **Setup** cell (**C2**) of the **System Coupling** system and select **Update** from the context menu.

---

3.   Double-click on the **Setup** cell (**C2**) of the **System Coupling** system to open the **System Coupling** tab.

    a.   Click on **Analysis Settings** in the **Outline of Schematic**.

	A	B
	Property	Value
1		
2	Analysis Type	General
3	☐ Initialization Controls	
4	Coupling Initialization	Program Controlled ▼
5	☐ Duration Controls	
6	Duration Defined By	Number Of Steps ▼
7	Number Of Steps	1
8	☐ Step Controls	
9	Minimum Iterations	1
10	Maximum Iterations	30

Properties of Analysis Settings

    •   In the **Properties of Analysis Settings** panel enter `30` for **Maximum Iterations**.

    b.   In **Outline of Schematic** select **Fluid Solid Interface** under **Steady State Thermal** and **interface_fluid** under **Fluid Flow (Fluent)** right-click and select **Create Data Transfer** from the context menu.

    Three **Data Transfer** objects are created.

      i.   One **Data Transfer** is for the **Temperature** transfer from Mechanical to Fluent. Rename it to `Temperature`.

      ii.   The second **Data Transfer** is for the **heat transfer coefficient** transfer from Fluent to Mechanical. Rename it to `HTC`.

      iii.   The third **Data Transfer** is for the **near wall temperature** transfer from Fluent to Mechanical. Rename it to `NWT`.

> **Note**
>
> Renaming the **Data Transfer** objects makes is easier to create charts of these quantities during the solution.

    c.   Save the project.

      File > **Save**

    d.   Click **Update** to start the solution.

2.6. System Coupling Solution

1.   When the solution starts, check the mapping diagnostics in the **System Coupling** log file.
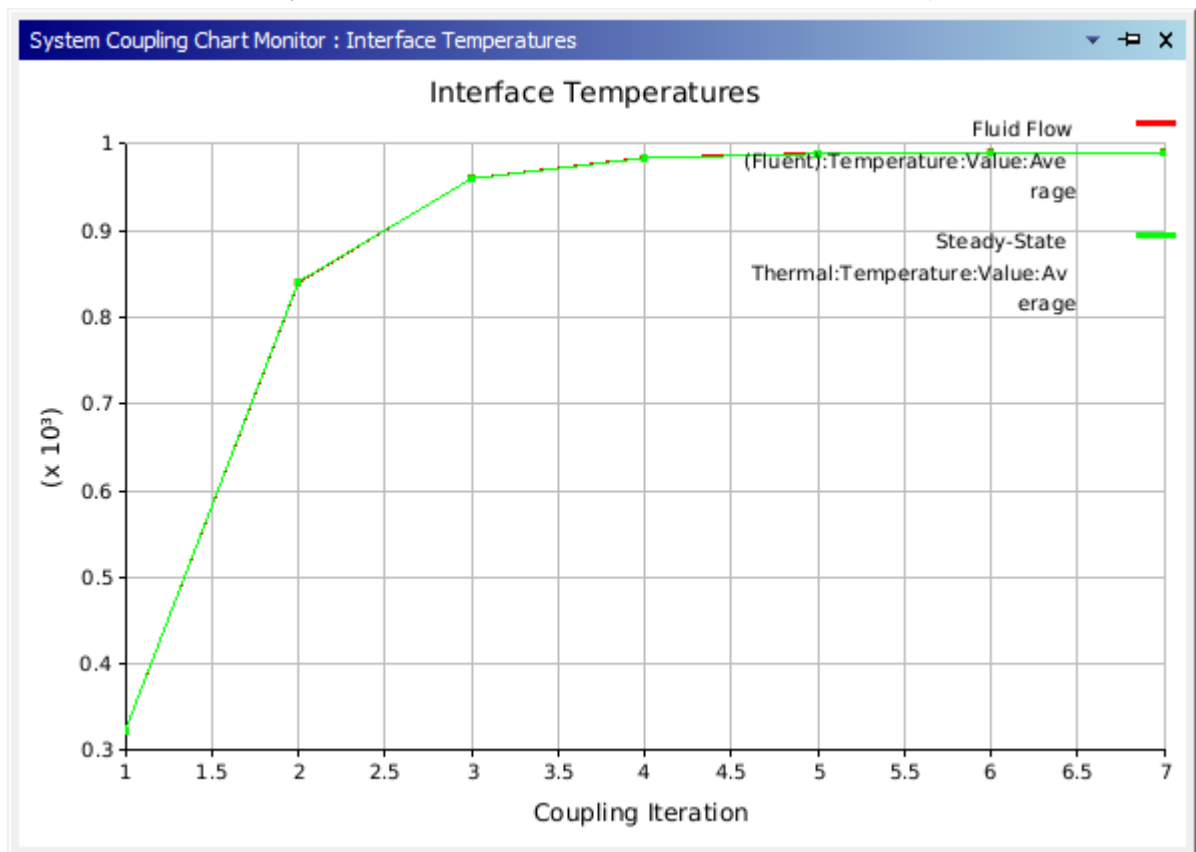
2. Right-click on **Chart Monitors** and select **Create Chart**.

   a. Rename the chart created to `Interface Temperatures`.

   b. Right-click on **Interface Temperatures** and add the average temperature variable by:

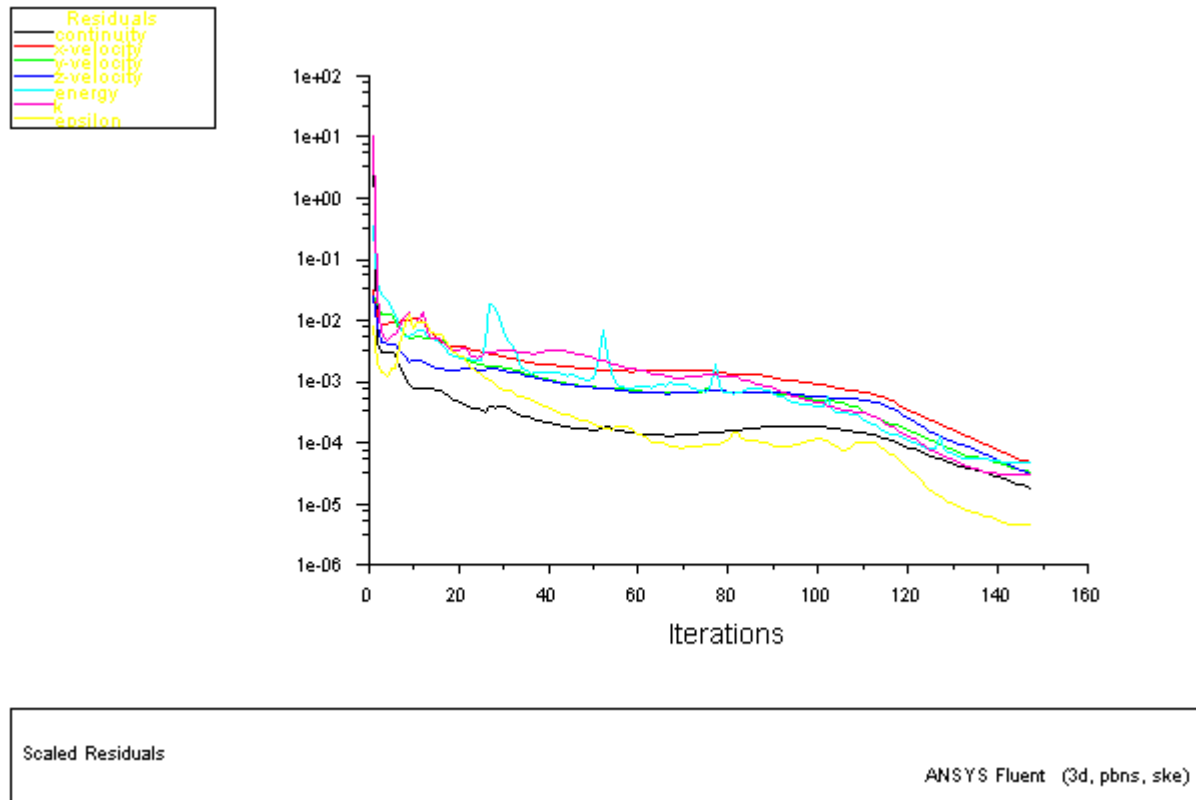   **Add Variable** > **Fluid Flow (Fluent)** > **Temperature** > **Value** > **Average**

      and

      Add Variable > **Steady-State Thermal** > **Temperature** > **Value** > **Average**

The chart shows the wall temperature values reach a steady value after 5 or 6 Coupling Iterations. You can create similar charts for the **HTC** and **NWT** data transfer values if you wish.

3. Return to the Fluent window and check the residuals plot.

Note

Fluent reaches convergence after about 150 iterations. Using 25 Fluent iterations per Coupling Iteration was a good choice, given how long it takes Fluent to converge and how many coupling iterations are needed to converge the Data Transfers.

To optimize solution times it is important to identify if convergence of the Data Transfers or convergence of the fields in Fluent is holding up a solution. If the flow field in Fluent takes a lot of iterations to reach convergence, then using more Fluent iterations per Coupling Iteration would help reduce the number of Coupling Iterations needed. This is particularly important if the Mechanical model is large and takes a long time to solve.

Conversely, if the Data Transfer quantities are very slow to converge, so many Coupling Iterations are required, then there is no need in using many Fluent iterations.

4. Save the project.

Note

Detailed post-processing instructions are not provided here. You can examine the solid temperature field in Mechanical and the fluid solution in Fluent or CFD-Post.

> A thermal stress analysis could easily be performed by dropping a **Static Structural** system onto the **Solution** cell (**A6**) of **Steady-State Thermal** system and then setting some structural constraints.

5. Save and close Workbench when you have finished looking at the solution.

2.7. Summary

This workshop has demonstrated how to perform 2-way thermal coupling between Fluent and Mechanical, which as an alternative to a Fluent CHT solution. The default couplings (HTC/Temperature) worked well for this case and converged quickly. Sometimes the HTC coupling will be slow to convergence and Heat Flow/Temperature coupling should be considered instead.