# Thermal-Structural 2-way FSI in a Leakage Path

This tutorial shows how to perform a 2-way FSI analysis that combines force/displacement coupling and thermal coupling. You will learn how to:

- Use a Static Structural system to define the structural-thermal model, including the use of Command objects to select coupled field elements and set the thermal boundary conditions.

- Expose thermal variables to System Coupling from the Static Structural system.

- Create a steady-state Fluent model that uses System Coupling to set thermal boundary conditions and dynamic mesh zones.

- Create dynamic mesh zones in Fluent that are consistent with the motion from an adjacent FSI interface.

- Define appropriate Analysis and Data Transfer settings in System Coupling.

- Monitor convergence in System Coupling.

- Change the Data Transfer variables in System Coupling to improve convergence.

1. Problem Description

This example considers a narrow leakage path that is representative of the gap between the plunger and barrel in a fuel injector. Upstream of the plunger pressurized fuel is supplied to the fuel injector. During operation the plunger will move to open/close the flow path supplying fuel to the engine cylinder.
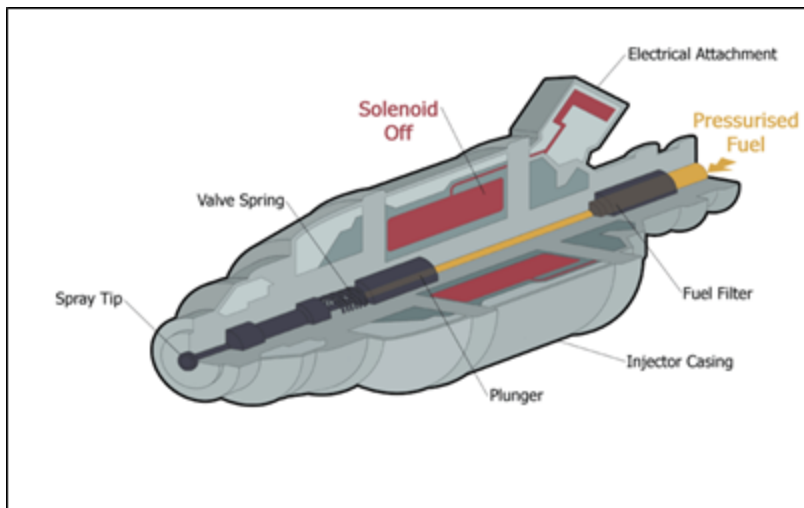
Since the plunger can move, a small clearance gap exists between the plunger and the fuel injector housing (the barrel). The fuel will leak through this gap. Due to the high fuel pressure, the gap can deform affecting the leakage flow rate. Furthermore, viscous heating of the fuel is significant and will cause heating in the plunger and barrel producing thermal stresses. The deformations due to thermal stresses will also affect the gap size, which will in turn affect the leakage flow rate and the viscous fluid heating.
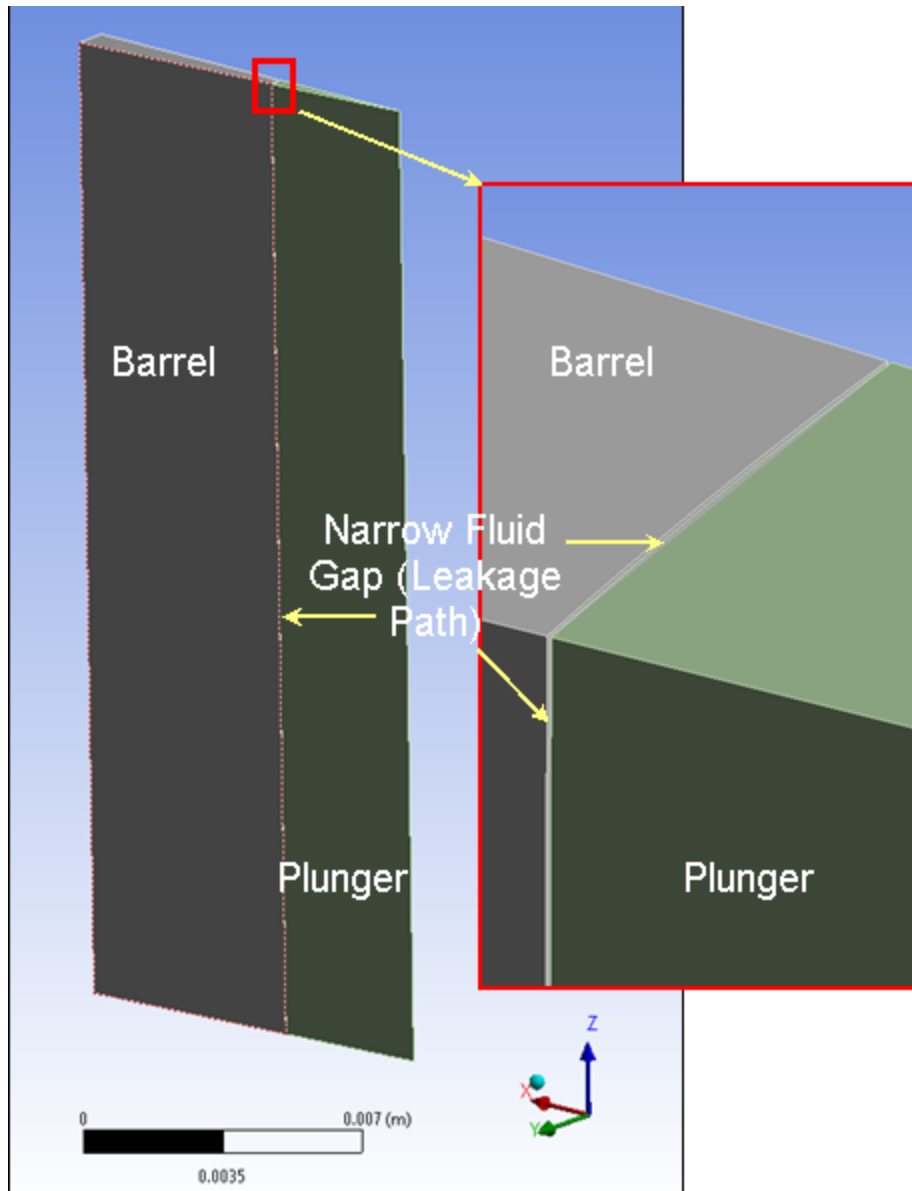
A coupled 2-way thermal-structural FSI solution is needed to solve this case. A steady-state approach is used with a stationary plunger.

A 5 degree slice of the full 3D model is used. The inlet side of the plunger is exposed to a pressure of 2500 bar (250 MPa) and a fuel temperature of 80° C. Fuel is forced through the narrow leakage path between the barrel and plunger which has an initial gap size of 5 microns. The outlet side is assumed to be at atmospheric pressure.

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 `Leakage_Path.wbpz` to the working folder.

2.2. Starting Workbench

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

    a. Select `Leakage_Path.wbpz` from your working folder.

    b. Save to your working folder.
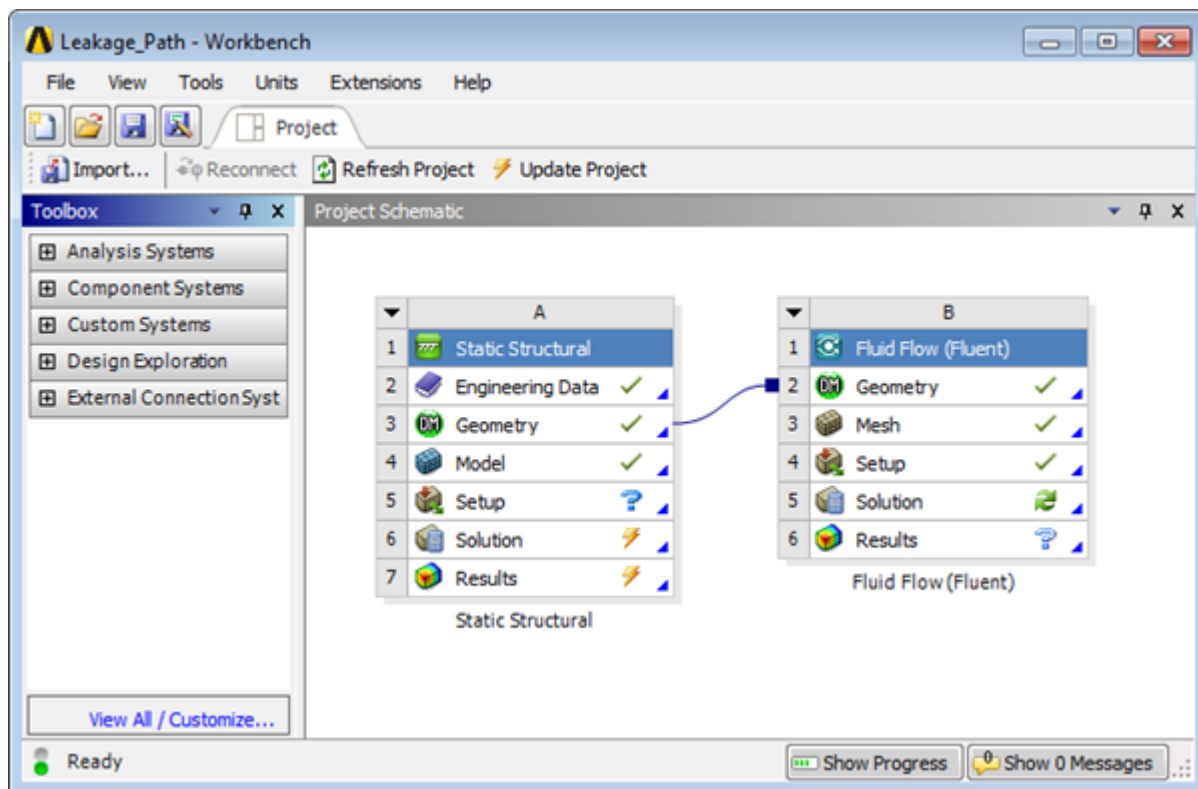
Note

The **Geometry** and **Mesh** for the **Static Structural** and **Fluent** systems has already been created. A basic **Fluent** setup without FSI has also been defined. Before completing the Structural and FSI setup we'll first review the existing mesh and Fluent setup.

2.3. Fluent Setup

1. Double-click **Setup** cell (**B4**) to open Fluent.

Ensure that **Double Precision** is enabled in the **Fluent Launcher** dialog box and click **OK**.

Note

The Fluent mesh for the leakage path contains hex elements with 8 elements through the gap thickness.

---

2. Click on **Models** in the tree.

    a. In the **Models** task page you can see that **Energy** is enabled, which shows that the energy equation is solved.

    b. **Laminar** is selected from the **Viscous Model**.

        i. Double-click **Viscous** in the **Models** task page the **Viscous Model** dialog box opens and you can see that **Viscous Heating** is enabled.

        ii. Close the **Viscous Model** dialog box.

3. Click on **Materials** in the tree.

    • In the **Materials** task page double-click on **diesel-fuel** in the list of **Materials**.

        i. In the **Create/Edit Materials** dialog box you can see that the **Density**, **Cp (Specific Heat)**, and **Thermal Conductivity** of the **diesel-fuel** material have **constant** values.

        ii. **Viscosity** is defined as function of temperature.

        iii. Close the **Create/Edit Materials** dialog box.

4. Click on **Cell Zone Conditions** in the tree.

    • In the **Cell Zone Conditions** task page double-click on **fluid** in the list of **Zone**.

        i. In the **Fluid** dialog box ensure that **diesel-fuel** is selected from the **Material Name** drop-down list.

        ii. Close the **Fluid** dialog box.

5. Click on **Boundary Conditions** in the tree and review each boundary condition.

    • Two wall boundary conditions are defined for the wetted surfaces of the plunger and barrel with an adiabatic thermal condition. These will be used for FSI interfaces later.

    • A pressure inlet boundary condition sets the pressure to 2.5e8 Pa and the temperature to 353.15 K.

    • A pressure outlet is defined at 0 Pa.

    • Two symmetry planes for the axi-symmetric surfaces complete the boundary conditions.

---

6. Click on **Monitors** in the tree.

   Three surface monitors have been defined to track the inlet mass flow, the outlet mass flow and the outlet temperature.

   a. Select the **Residuals** in the **Monitors** task page and click **Edit....**

      In the **Residual Monitors** dialog box you can see that —

      · **Compute Local Scale** is enabled.

      · **local scaling** is selected from the **Reporting Option** drop-down list.

      > ### Note
      >
      > Using local residual scaling gives a more consistent representation of the residuals.

      · The default value of **1e-05** for **Absolute Criteria** is retained for all equations. This is quite a tight convergence tolerance.

   b. Close the **Residual Monitors** dialog box.

7. Click on **Solution Methods** in the tree.

   You can see in the **Solution Methods** task page that **Coupled** is selected from the **Scheme** drop-down list, with the **Pseudo Transient** option and **High Order Term Relaxation**. These settings give good convergence in Fluent for this case.

8. Click on **Run Calculation** in the tree.

   You can see that a **Timescale Factor** of **10** is used for the pseudo transient timescale. This acceler-ates convergence for this case. For the rest the default settings are used.

   > ### Note
   >
   > Next you'll review and complete the Structural Setup. You can leave Fluent open.

2.4. Structural Setup

Return to the Workbench window and in the **Project Schematic** double-click on **Setup** cell (**A5**) of **Static Structural** system.

1. In the Mechanical window once the geometry is loaded you can review the structural geometry and mesh.

   · In the tree under **Geometry** you can see that the **Fluid** body has been suppressed leaving a small gap between the **Plunger** and **Barrel** bodies.

- Under **Connections** you can see that **Contacts** was automatically created between the **Plunger** and **Barrel** bodies but has been suppressed.
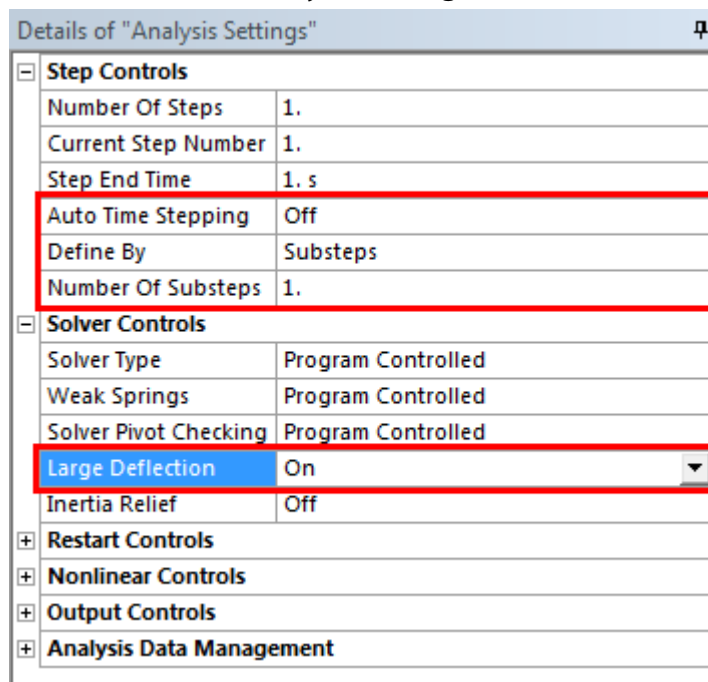
> **Note**
>
> It is important to always check the contact regions to make sure they are suitable.

- The mesh has been created using Sweep methods.

- You can see the **Named Selections** in the tree, which have been imported from the geometry model.

2. Make sure the **Units** in Mechanical are set to **Metric (m, kg, N, s, V, A)** and **Celsius** is enabled from the **Units** menu.
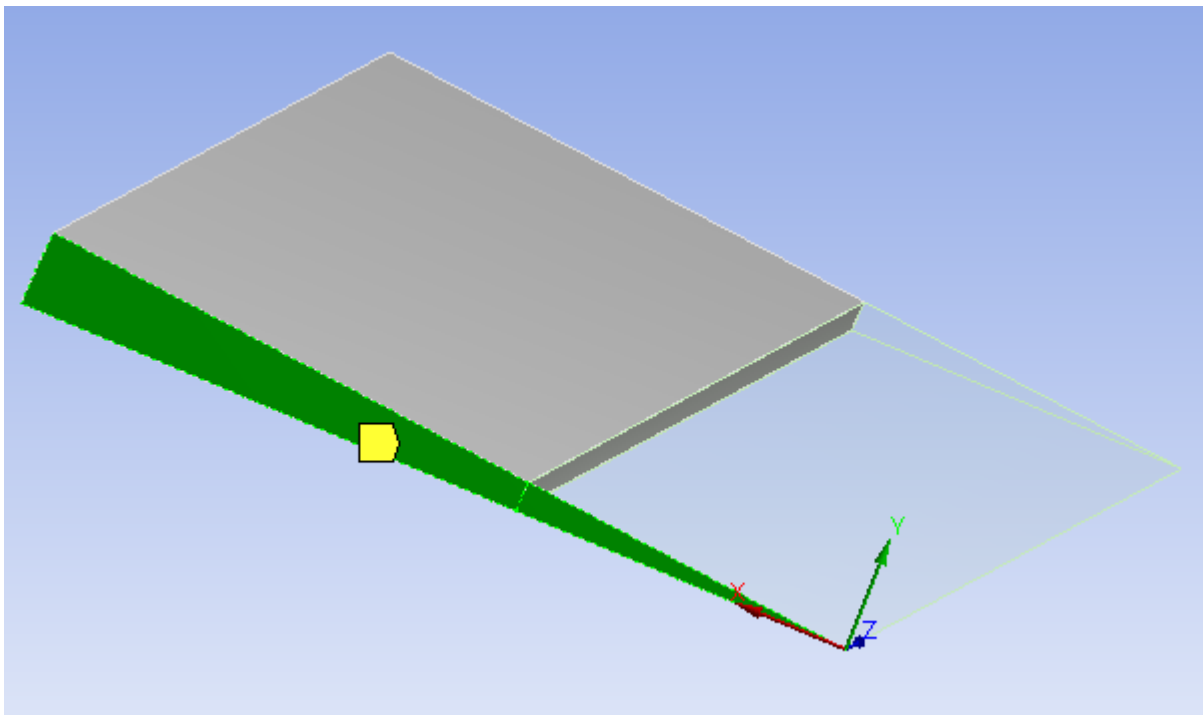
3. In the tree click on **Analysis Settings**.

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
Weak Springs	Program Controlled
Solver Pivot Checking	Program Controlled
Large Deflection	On
Inertia Relief	Off
Restart Controls	
Nonlinear Controls	
Output Controls	
Analysis Data Management	

a. In the **Details of Analysis Settings** panel select **Off** from the **Auto Time Stepping** drop-down list.

b. Retain the selection of **Substeps** from the **Define By** drop-down list.

c. Retain **1** for **Number Of Substeps**.

d. Under **Solver Controls** select **On** from the **Large Deflection** drop-down list.

4. Right-click on **Analysis Settings** in the tree and select **Insert** > **Pressure** from the context menu.

Details of "Pressure"		무
⊟ **Scope**		
Scoping Method	Named Selection	
Named Selection	plunger_inlet	
⊟ **Definition**		
Type	Pressure	
Define By	Normal To	
☐ Magnitude	2.5e+008 Pa (ramped)	►
Suppressed	No	

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

    b.   Select **plunger_inlet** from the **Named Selection** from down list.

    c.   Enter 2.5e+008 for **Magnitude**.

5.   Right-click on **Analysis Settings** in the tree and select **Insert** > **Displacement** from the context menu.

    a.   In the **Details of Displacement** panel for **Geometry** select the two faces at the outlet (low-Z) side of the model as shown in Figure 52: Faces Selected for Displacement (p. 7).
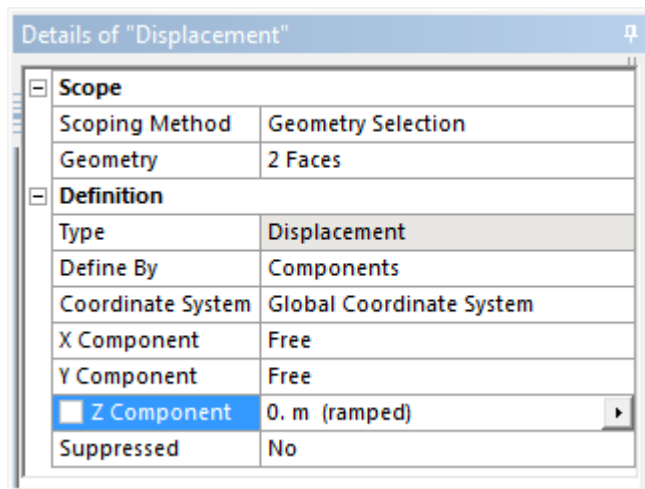
Figure 52: Faces Selected for Displacement
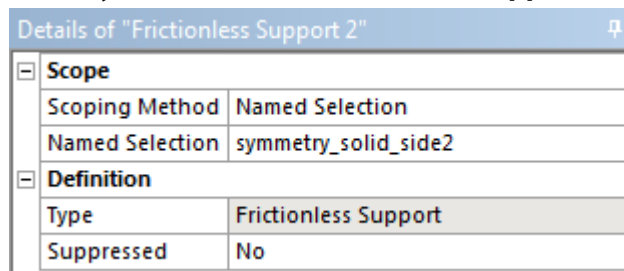

> **Note**
>
> Before selecting the faces make sure that the **Face** button in the toolbar 🔲 is pressed.

b.   Retain the selection of **Free** for **X Component** and **Y Component**.

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	Free
Y Component	Free
☐ Z Component	0. m (ramped) ▸
Suppressed	No

c.   Enter 0 for **Z Component**.

6.   Right-click on **Analysis Settings** in the tree and select **Insert** > **Frictionless Support** from the context menu.

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

b.   Select **symmetry_solid_side1** from the **Named Selection** from down list.

7.   Similarly insert a second **Frictionless Support** using **symmetry_solid_side2**.

Details of "Frictionless Support 2"	
Scope	
Scoping Method	Named Selection
Named Selection	symmetry_solid_side2
Definition	
Type	Frictionless Support
Suppressed	No

> **Note**
>
> The displacement support constrains the model in the z direction while the frictionless support will prevent out-of-plane motion in the x and y directions. A fixed support should not be used because it sets all element degrees of freedom to zero, including temperature.

8.   Right-click on **Analysis Settings** in the tree and select **Insert** > **Fluid Solid Interface** from the context menu.

Details of "FSI Barrel"	
Scope	
Scoping Method	Named Selection
Named Selection	interface_solid_barrel_side
Definition	
Type	Fluid Solid Interface
Interface Number	1.
Data to Transfer [Expert]	All System Coupling Data Transfers
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_barrel_side** from the **Named Selection** from down list.

c. Select **All System Coupling Data Transfers** from the **Data to Transfer [Expert]** drop-down list.

d. Rename the interface to `FSI Barrel`.

Note

This will tell **System Coupling** that this FSI interface is able to couple thermal quantities as well as force/displacement. You still need to switch from structural elements to thermal-structural elements in Mechanical and define the thermal boundary conditions using **Commands** objects.

---

9. Similarly insert a second **Fluid Solid Interface** using the **Named Selection interface_solid_plunger_side**. Select **All System Coupling Data Transfers** from the **Data to Transfer [Expert]** drop-down list and rename the interface to `FSI Plunger`.

10. Right-click on **Analysis Settings** in the tree and select **Insert** > **Commands** from the context menu.

a. Enter the commands as shown below.

Note

To save time you can copy and paste the first section from the `Commands.txt` file provided with this tutorial.

```
/prep7

! Get max element type number
*get,etype_num,etyp,0,num,max

! Define coupled field elements with thermal-structural DOF
et,etype_num+1,solid226,11
et,etype_num+2,solid227,11

! Change solid187 to solid227
esel,s,ename,,187
emodif,all,type,etype_num+2
```

```
! Change solid186 to solid226
esel,s,ename,,186
emodif,all,type,etype_num+1

! Select all elements
esel,all

/solu
```

> **Note**
>
> These commands will change all SOLID186/187 structural elements to SOLID226/227
> coupled field elements with thermal-structural degrees of freedom. These commands
> can be applied to any structural case that was meshed with SOLID186/187 elements.

    b.   Rename the **Commands (APDL)** object to `Commands – Switch Element Types.`

11.   Similarly insert another **Commands** object and enter the commands as shown. You can copy and paste
from the `Commands.txt` file again.

```
! Set initial temperature
ic,all,temp,40.0

! Apply a convection load on the barrel external wals
sf,barrel_external_walls,conv,500,40

! Apply a temperature constraint on the plunger inlet side
d,plunger_inlet,temp,80
```
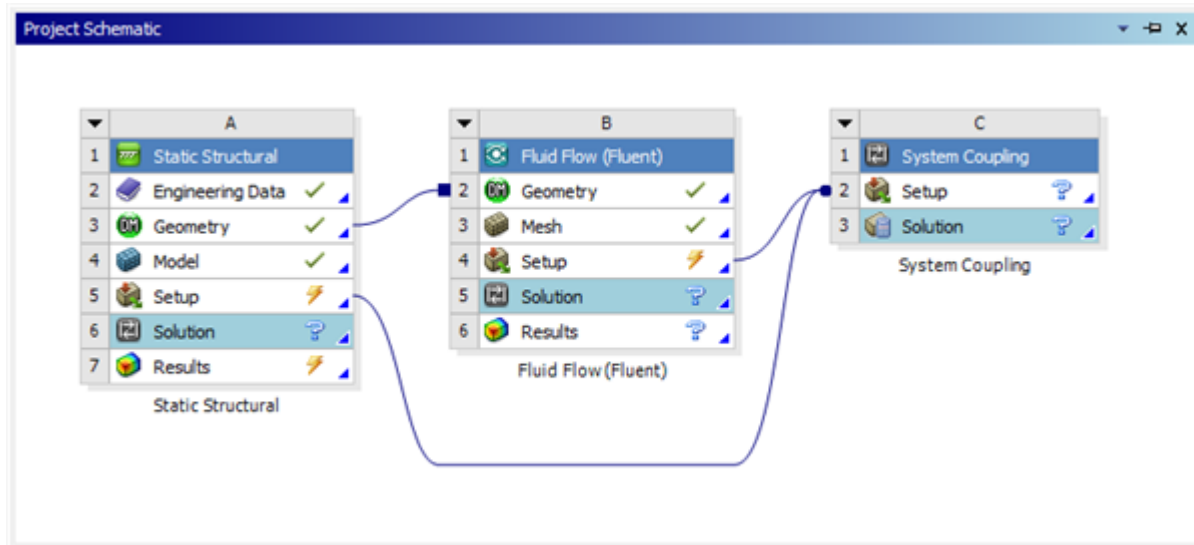
> **Note**
>
> These commands will set the initial temperature to 40° C.If you select **Static Structural
> (A5)** from the tree, it shows the **Environment Temperature** is **22° C**. Zero thermal
> stresses are assumed at the **Environment Temperature**.
>
> The commands also define a convection coefficient of 500 W/m^2 C with a bulk temper-
> ature of 40° C applied to the named selection **barrel_external_walls** and a temperature
> constraint of 80° C on the named selection **plunger_inlet**.

    •   Rename the **Commands (APDL)** object to `Commands – Thermal/Initial Boundary Con-`
`ditions.`

12.   On the Workbench **Project Schematic** drag-and-drop a **System Coupling** component system onto the
Setup cell (**A5**) of the **Static Structural** system.

a. Also connect the **Setup** cell (**B4**) of the **Fluid Flow (Fluent)** to the **Setup** cell (**C2**) of the **System Coupling** system.

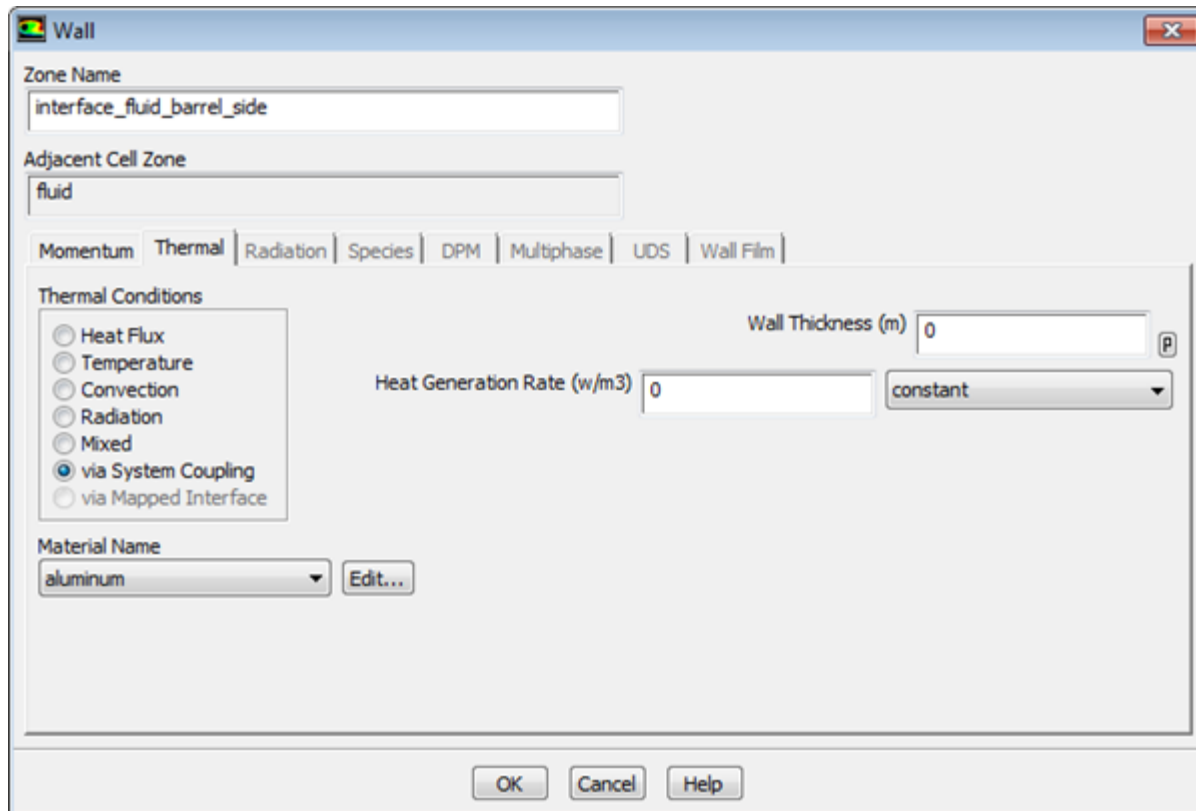b. Update the **Setup** cell (**A5**) of **Static Structural** system.

> **Note**
>
> Next you will complete the Fluent setup by defining thermal and structural FSI coupling regions, dynamic mesh settings and solver controls.

13. Save the project.

2.5. Fluent FSI Setup

Return to the Fluent window.

1. In Fluent in the **Boundary Conditions** task page double-click **interface_fluid_barrel_side**.

a. In the **Thermal** tab of the **Wall** dialog box select **via System Coupling** from the **Thermal Conditions** group box.
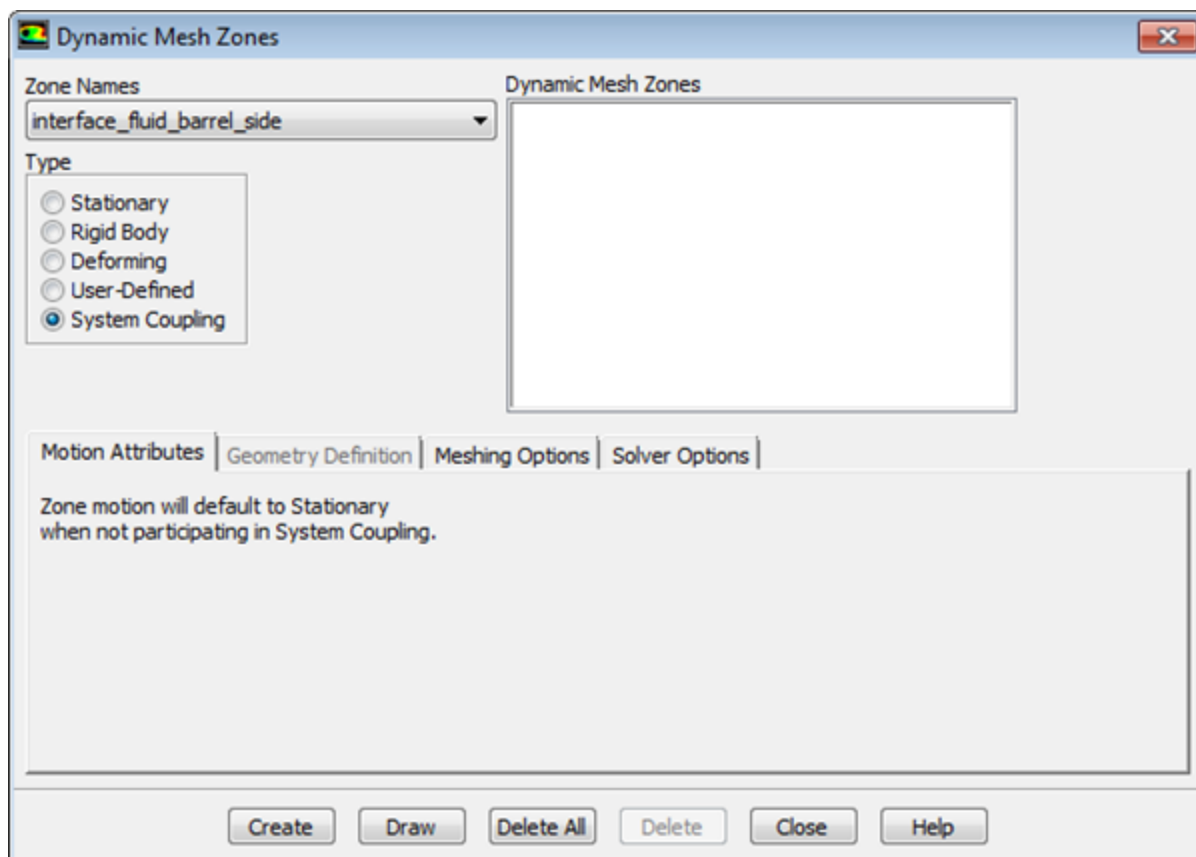
> **Note**
>
> The type of thermal data to transfer will be set in **System Coupling**.

b. Close the **Wall** dialog box.

2. Similarly set **via System Coupling** as the **Thermal Conditions** for **interface_fluid_plunger_side**.

3. From the tree select **Dynamic Mesh** and in the **Dynamic Mesh** task page enable **Dynamic Mesh**.

a. Under **Dynamic Mesh Zones** click **Create/Edit...**.

i. In the **Dynamic Mesh Zones** dialog box select **interface_fluid_barrel_side** from the **Zone Names** drop-down list.

ii. Select **System Coupling** in the **Type** group box and click **Create**.

iii. Similarly select **interface_fluid_plunger_side** from **Zone Names** and selecting **System Coupling** create another FSI interface region.


Note

It is always important to consider the dynamic mesh settings for boundary zones adjacent to moving FSI interfaces. In this case the two symmetry boundaries, the inlet and the outlet are adjacent to the FSI interfaces.

On the symmetry planes the nodes that are shared with the FSI interfaces may move, but they will remain planar due to the structural constraints. So the symmetry planes can be set to deforming using the planar option.

The outlet will also remain planar since the structural motion is constrained in the z direction at the outlet, but radial motion could occur. Therefore the outlet can also be set to deforming using the planar option.

The inlet may move in the radial direction and also in the z-direction, so the planar option is not valid here. In fact nothing can be said about the inlet motion since it will depend on how the adjacent FSI interfaces move. In this case you can use the unspecified option, which is only available with the **Linearly Elastic Solid** smoothing method. The inlet will be free to float anywhere and its position will be determined by the motion of adjacent boundaries and the interior mesh.

---

iv. Close the **Dynamic Mesh Zones** dialog box.

b. In the **Dynamic Mesh** task page click **Settings** to open the **Mesh Method Settings** dialog box.
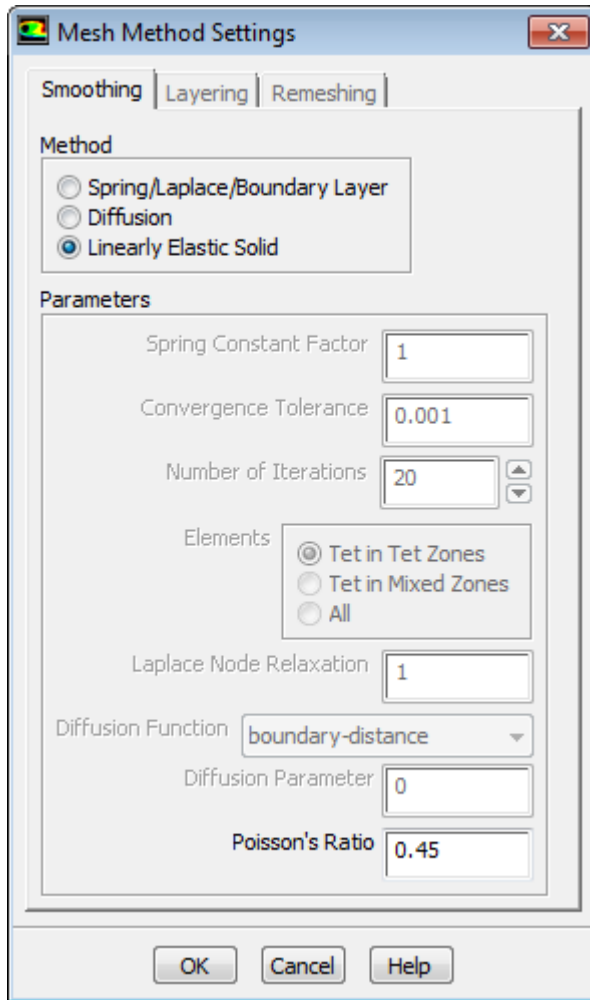
i. Select **Linearly Elastic Solid** from the **Method** group box in the **Smoothing** tab.

ii. Click **OK** to close the **Mesh Method Settings** dialog box.

c. In the **Dynamic Mesh** task page click **Create/Edit...**.

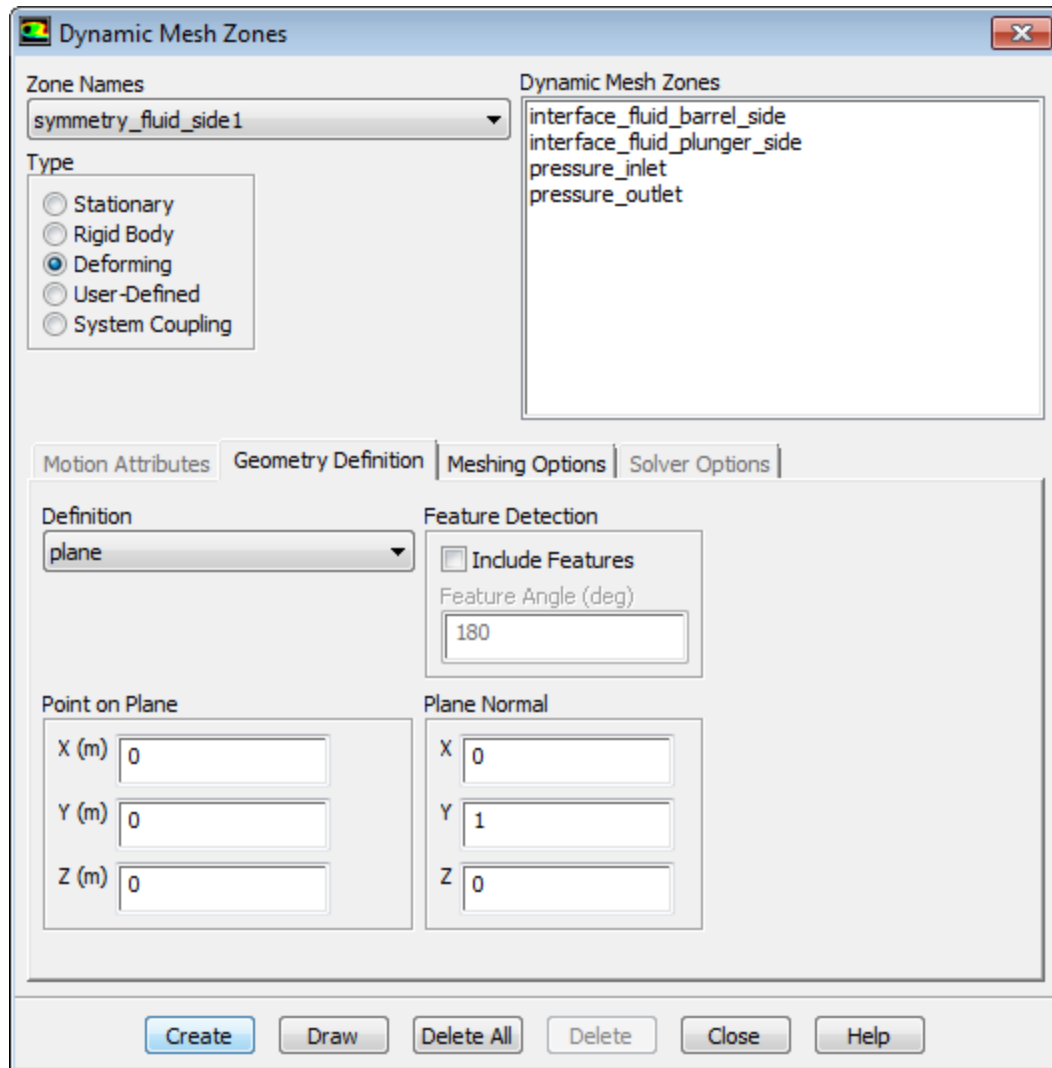i. Select **pressure_inlet** from the **Zone Names** drop-down list.

14
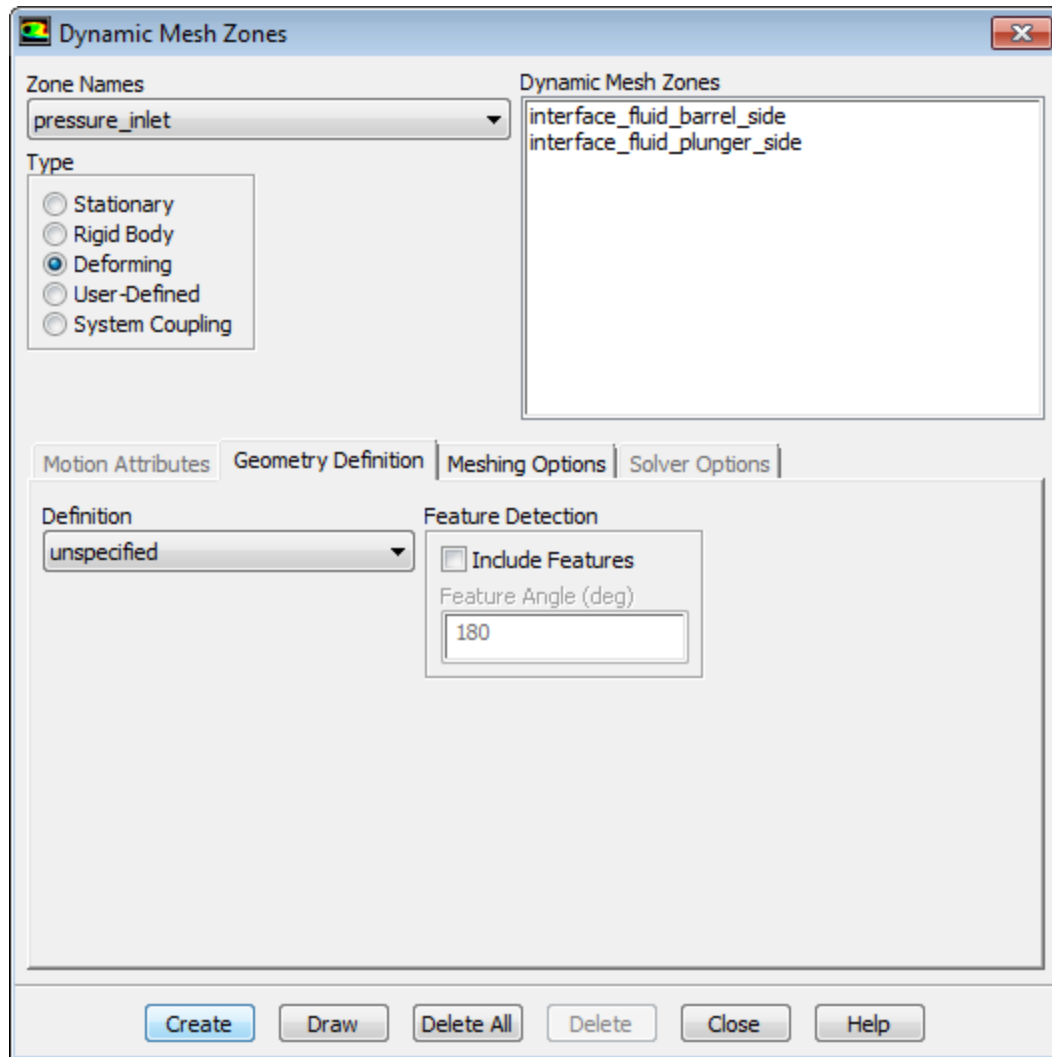
A. Select **Deforming** from the **Type** group box.

B. In the **Geometry Definition** tab select **unspecified** from the **Definition** drop-down list.

C. Click **Create**.

ii. Select **pressure_oulet** from the **Zone Names** drop-down list.
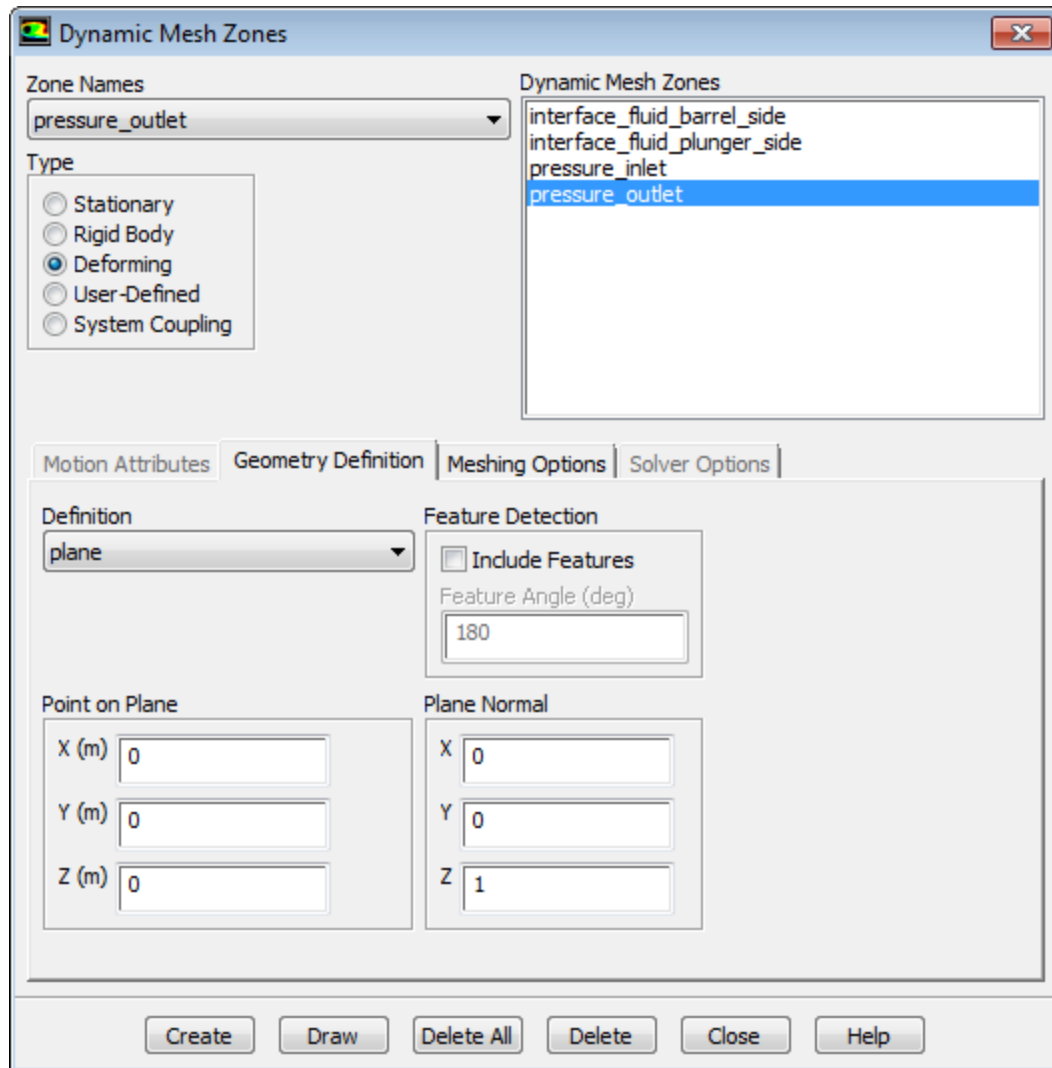
A.   Retain the selection of **Deforming** from the **Type** group box.

B.   In the **Geometry Definition** tab select **plane** from the **Definition** drop-down list.

C.   Retain **0**, **0**, and **0** for **X**, **Y**, and **Z** respectively in the **Point on Plane** group box.

D.   Enter 0, 0, and 1 for **X**, **Y**, and **Z** respectively in the **Point Normal** group box.

E.   Click **Create**.

iii.   Select **symmetry_fluid_side1** from the **Zone Names** drop-down list.

A. Retain the selection of **Deforming** from the **Type** group box.

B. In the **Geometry Definition** tab retain the selection of **plane** from the **Definition** drop-down list.

C. Retain **0**, **0**, and **0** for **X**, **Y**, and **Z** respectively in the **Point on Plane** group box.

D. Enter 0, 1, and 0 for **X**, **Y**, and **Z** respectively in the **Point Normal** group box.

E. Click **Create**.

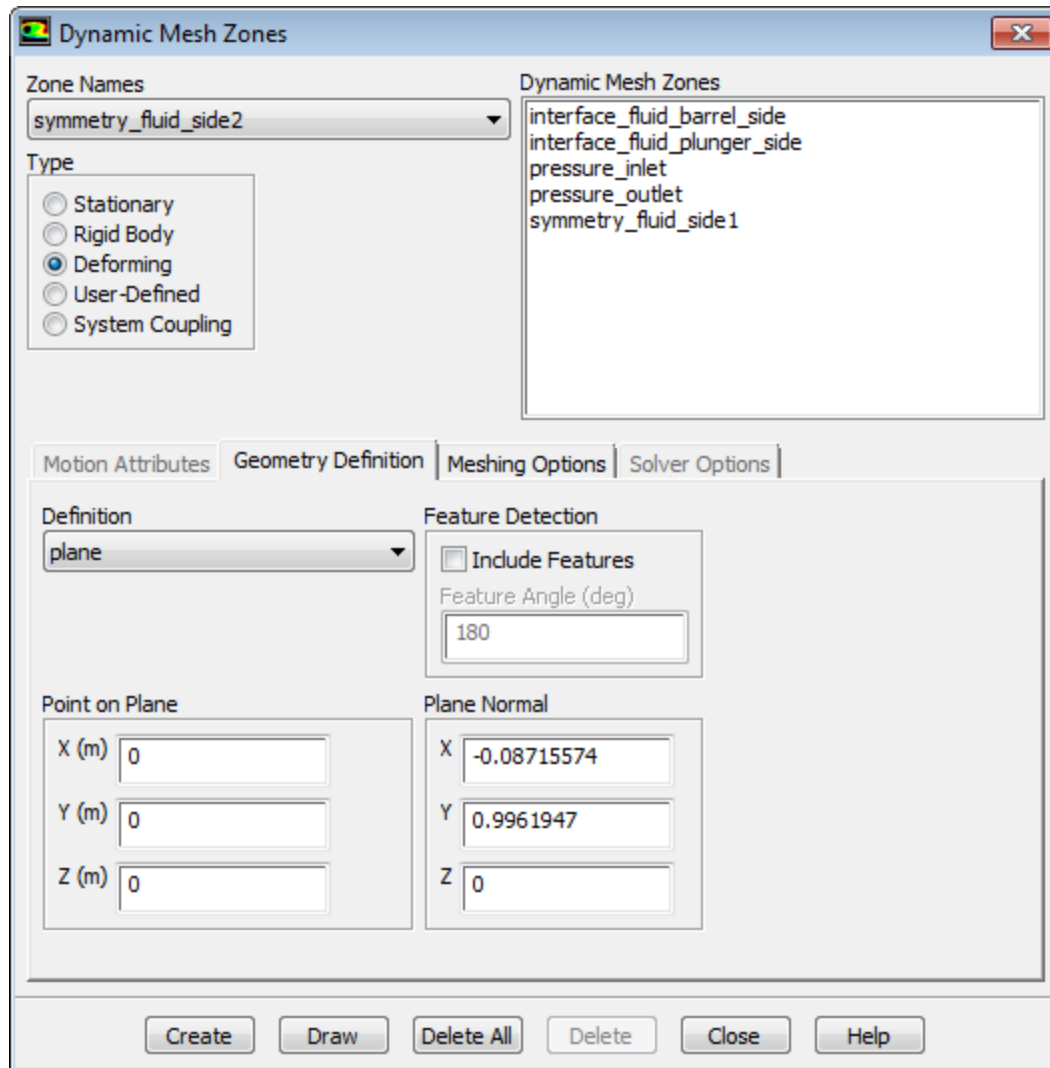iv. Select **symmetry_fluid_side2** from the **Zone Names** drop-down list.

A. Retain the selection of **Deforming** from the **Type** group box.

B. In the **Geometry Definition** tab retain the selection of **plane** from the **Definition** drop-down list.

C. Retain **0**, **0**, and **0** for **X**, **Y**, and **Z** respectively in the **Point on Plane** group box.

D. Enter `-0.08715574`, `0.9961947`, and `0` for **X**, **Y**, and **Z** respectively in the **Point Normal** group box.

E. Click **Create**.

Note

The plane normal is equal to (-sin 5°, cos 5°, 0). When using the plane option it is important to define the point and normal accurately. The mesh motion is constrained to the plane defined by these values. If the point and normal are inconsistent with the initial geometry/mesh position, then the mesh will snap to the defined plane at the start of the solution and probably cause a dynamic mesh failure.

> v. Close the **Dynamic Mesh Zones** dialog box.

4. Click on **Run Calculation** in the tree.

- In the **Run Calculation** task page enter 10 for **Number of Iterations**.

Note

This is the number of Fluent iterations per coupling iteration. For a steady-state FSI case convergence is typically reached over many coupling iterations and Fluent is not driven to full convergence at every coupling iteration.

---

5. In the Fluent window click on the button **Synchronize WB cell status** ⚠ on the toolbar to pass the latest Fluent settings to Workbench.

6. Return to the Workbench window and double click on the **Setup** cell (**C2**) of the **System Coupling** system.

    Click **Yes** in the dialog box that appears asking to read the upstream data.

2.6. System Coupling Setup

In **System Coupling** the **Regions** available for coupling are shown below **Static Structural** and **Fluid Flow (FLUENT)**. Selecting these regions shows the variables that can be received (**Input**) and sent (**Output**). Note that both thermal and structural variables are available.

1. In the **Outline of Schematic** multi select (by holding down the **Ctrl** key) **FSI Barrel** from under **Static Structural Regions** and **interface_fluid_barrel_side** from under **Fluid Flow (FLUENT) Regions**. Right-click and select **Create Data Transfer** from the context menu.

> **Note**
>
> Review the 5 **Data Transfers** that are created. Fluent will send **force**, **heat transfer coefficient**, and **near wall temperature** to Mechanical. Mechanical will send displacement and temperature to Fluent. If necessary you could change the data transfers so Fluent sends heat flow to Mechanical and receives temperature, or vice-versa. This is less stable but potentially faster to converge. The default heat transfer coefficient option will be used here.

2. Create the second set of data transfers by multi-selecting **FSI Plunger** from under **Static Structural Regions** and **interface_fluid_plunger_side** from under **Fluid Flow (FLUENT) Regions**, right-click and selecting **Create Data Transfer** from the context menu.
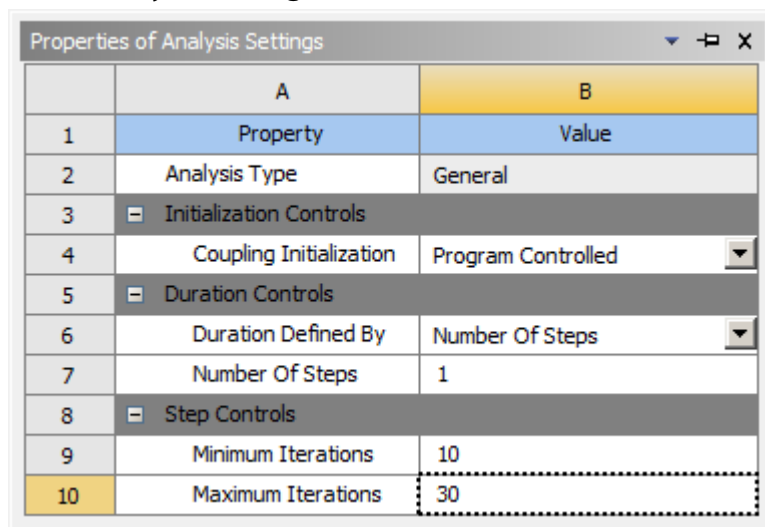
> **Note**
>
> By default all data transfers are created with a **Under Relaxation Factor** of **1** and no **Ramping**. Thermal coupling using temperature and HTC is very stable and should not be under-relaxed. Force and/or displacements will typically need under-relaxation or ramping in steady-state cases.
>
> For this case the displacements are largely due to thermal stresses. Without under-relaxation or ramping the initial deformations due to thermal stresses will cause large displacements in Fluent and a dynamic mesh failure. To introduce the displacements gradually to Fluent we could use a small under-relaxation factor (say 0.05), but this would take a long time to iterate to convergence. Ramping displacements is a better option here.

3. Select **Analysis Settings** in the **Outline of Schematic**.

	A	B
	Property	Value
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	10
10	Maximum Iterations	30

Properties of Analysis Settings

a. In the **Properties of Analysis Settings** panel enter 10 for **Minimum Iterations**.

b. Enter 30 for **Maximum Iterations**.

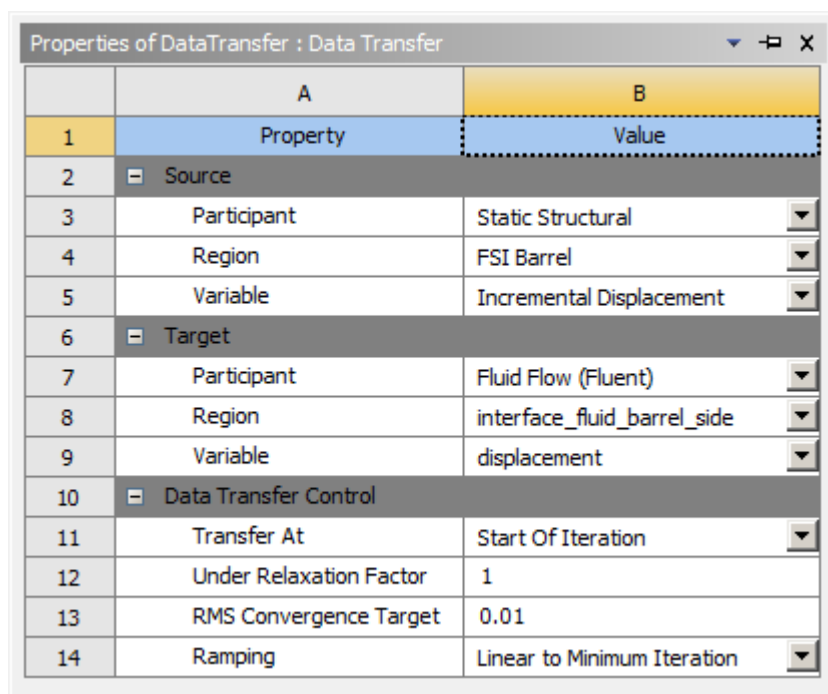4. Select **Data Transfer 6** in the **Outline of Schematic**.

> **Note**
>
> Ensure that it is a displacement transfer.

- In the **Properties of Data Transfer** select **Linear to Minimum Iteration** from the **Ramping** drop-down list.

5. Similarly, for **Data Transfer** (the first one) ensure that it is a displacement transfer and set the **Ramping** to **Linear to Minimum Iteration**.

	A	B
	Property	Value
2	⊟ Source	
3	Participant	Static Structural
4	Region	FSI Barrel
5	Variable	Incremental Displacement
6	⊟ Target	
7	Participant	Fluid Flow (Fluent)
8	Region	interface_fluid_barrel_side
9	Variable	displacement
10	⊟ Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	1
13	RMS Convergence Target	0.01
14	Ramping	Linear to Minimum Iteration

Properties of DataTransfer : Data Transfer

> **Note**
>
> The two displacement transfers will now be ramped over the first 10 coupling iterations. In the first coupling iteration 1/10th of the displacement calculated by Mechanical will be passed to Fluent. In the second coupling iteration 2/10th will be passed, etc. From coupling iteration 10 to 30 the full displacements are applied.
>
> Note that 1 coupling step with up to 30 coupling iterations is used here. Be careful if using the alternative approach of 30 coupling steps with 1 coupling iteration per step; three key points to note are:
>
> - Independently of any under-relaxation or ramping set in **System Coupling**, Mechanical will ramp all loads received from **System Coupling** across the number of coupling steps defined.
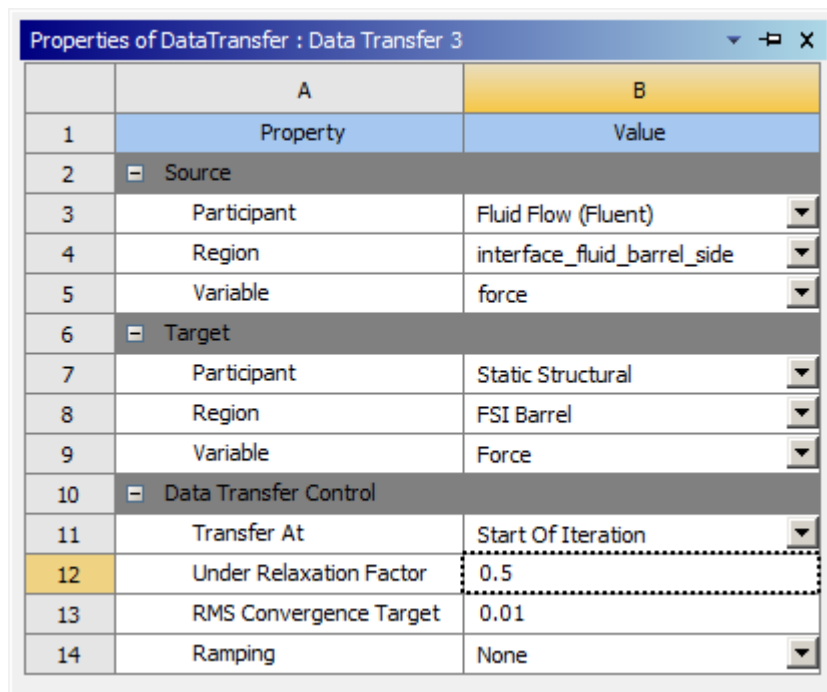

- Incremental displacement (relative to the last step) is always transferred. This must be fully converged at the end of each step to avoid error accumulation.

- By default Mechanical will save results at every step.

Now add some under-relaxation for the force transfer.

6.   Select **Data Transfer 3** in the **Outline of Schematic**.

Note

Ensure that the **Variable** is set to **force**.

	A	B
	Property	Value
1		
2	⊟ Source	
3	Participant	Fluid Flow (Fluent)
4	Region	interface_fluid_barrel_side
5	Variable	force
6	⊟ Target	
7	Participant	Static Structural
8	Region	FSI Barrel
9	Variable	Force
10	⊟ Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.5
13	RMS Convergence Target	0.01
14	Ramping	None

Properties of DataTransfer : Data Transfer 3

- In the **Properties of Data Transfer** enter 0 . 5 for **Under Relaxation Factor**.

7.   Repeat the last step for **Data Transfer 8**.

8.   Save the project.

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

10. Check the mapping diagnostics as the solution starts and confirm 100% mapping for all data transfers.

```
Solution Information : System Coupling                       ▼ ⊟ ✕
================================================================
     +------------------------------------------------------+
     |                   MAPPING SUMMARY                    |
     +------------------------------------------------------+
     | Data Transfer            |                           |
     |   Diagnostic             |   Source Side   Target Side |
     +--------------------------+---------------------------+
     | Data Transfer            |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 2          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 3          |                           |
     |   Percent Nodes Mapped   |      100           100     |
     |   Percent Area Mapped    |      100           100     |
     | Data Transfer 4          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 5          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 6          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 7          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 8          |                           |
     |   Percent Nodes Mapped   |      100           100     |
     |   Percent Area Mapped    |      100           100     |
     | Data Transfer 9          |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     | Data Transfer 10         |                           |
     |   Percent Nodes Mapped   |      N/A           100     |
     +------------------------------------------------------+
```

11. While the solution is running right-click on **Chart Monitors** and select **Create Chart** from the context menu.

12. Right-click on the newly created **Chart 2** and select **Add Variable** > **Fluid Flow (Fluent)** > **Data Transfer 2** > **Value** > **Average**.
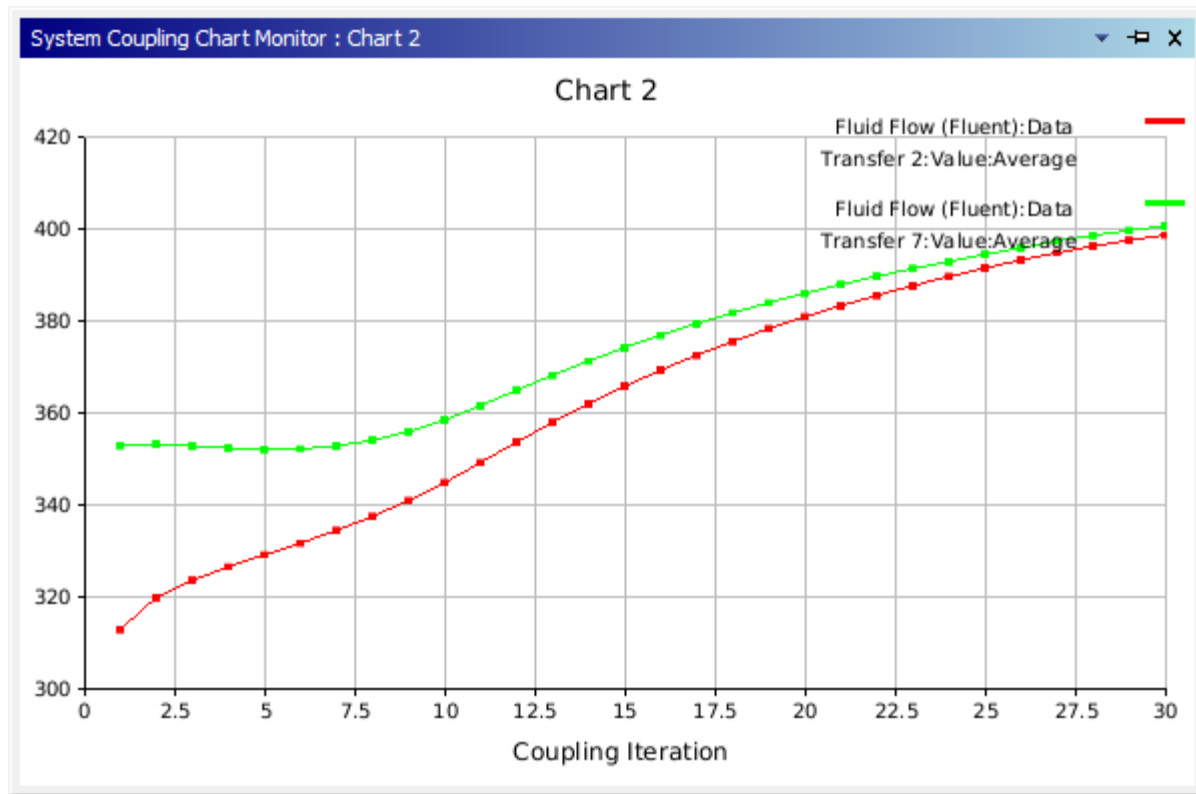

Note

This adds the (nodal) average temperature on **FSI Barrel** to the chart.

---

13. Similarly add the average temperature on **FSI Plunger** to the same chart using:

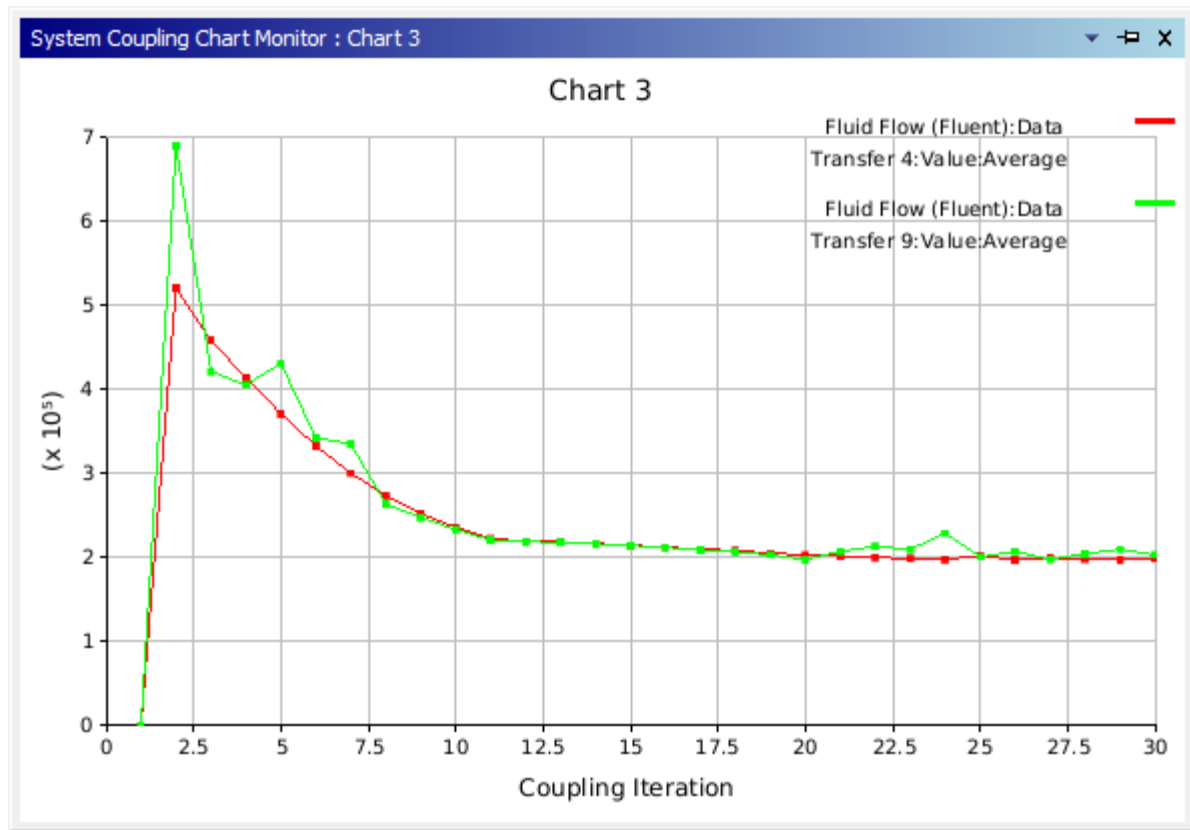**Add Variable** > **Fluid Flow (Fluent)** > **Data Transfer 7** > **Value** > **Average**

14. Create another chart and add the average heat transfer coefficient values using:

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer 4** > **Value** > **Average**

    and

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer 9** > **Value** > **Average**
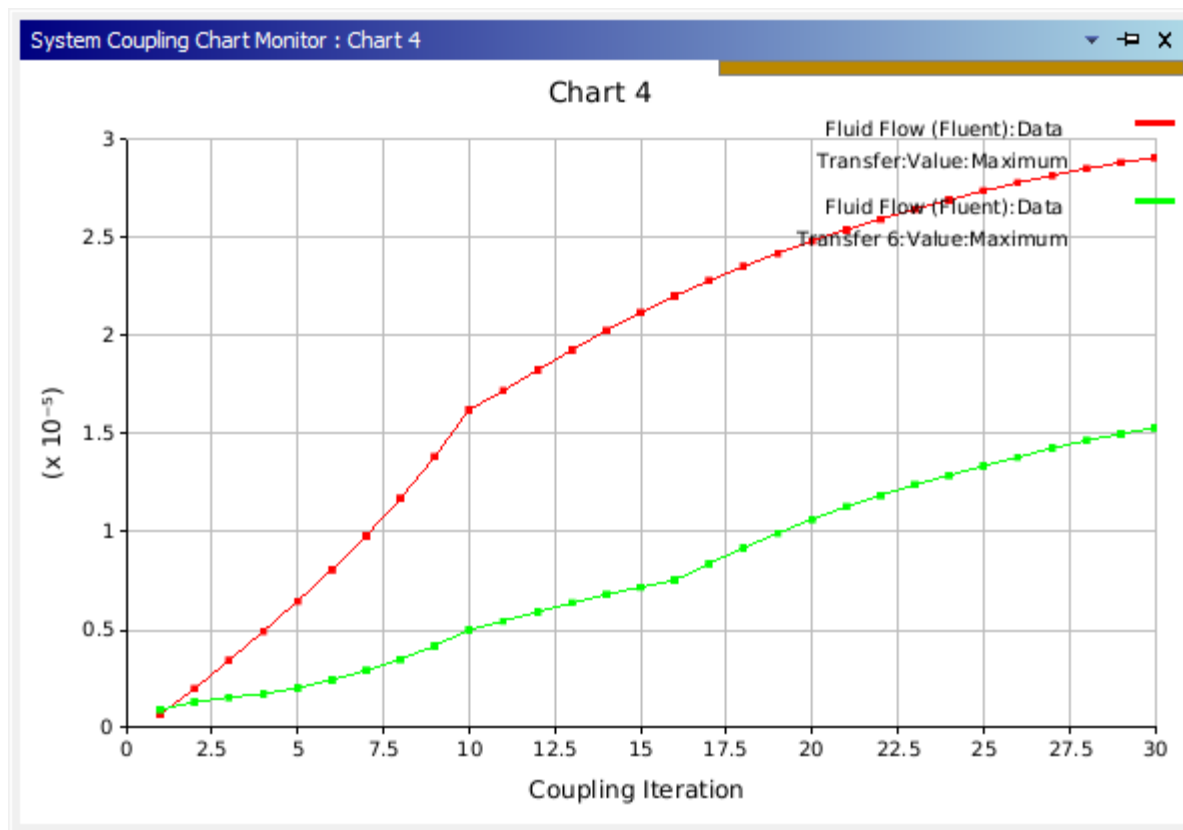
15. Create another chart and add the maximum displacement values using:

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer** > **Value** > **Maximum**

    and

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer 6** > **Value** > **Maximum**

Note

The variables added to all the previous charts were all on the Fluent side. This represents the quantities **System Coupling** receives from or sends to Fluent. You can plot the same variables on the **Structural** side. If under-relaxation or ramping is used then you will see a difference between the Fluent side and **Structural** side values.

- Add the following variables to **Chart 4**:

  Add Variable > **Static Structural** > **Data Transfer** > **Value** > **Maximum**

  and

  Add Variable > **Static Structural** > **Data Transfer 6** > **Value** > **Maximum**
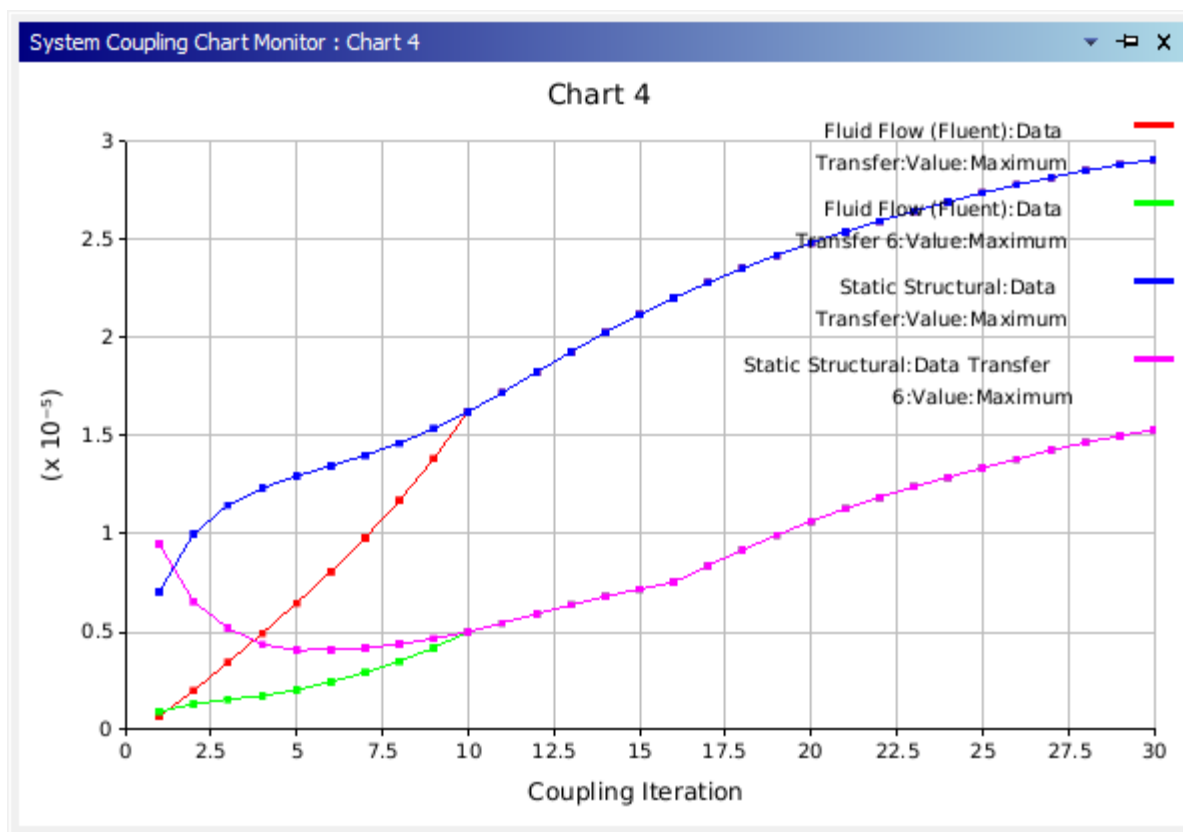
Note

Now you can see the effect that ramping has on the maximum displacement sent to Fluent, compared to the maximum displacement received from Mechanical.

Note that when plotting averages and sums the nodal average/sum of the interface data is used. So a nodal average of Temperature on the Fluent side generally won't equal a nodal average of Temperature on the **Structural** side. Nodal sums of forces and heat flow should be equal (with 100% mapping).

Note

The charts clearly show that the solution has not fully converged after 30 coupling iterations. You could add another coupling step now and extend the run, but it will in fact take many more coupling iterations to reach full convergence. The slow convergence is because the near wall reference temperature associated with the heat transfer coefficient is not a good representation of the free stream temperature. Instead of extending the run you will re-run using Temperature and Heat Flow coupling, with some under-relaxation to stabilize the thermal coupling.

16. Return to **Project Schematic** by clicking on the **Project** tab.

   • Right-click on the **Solution** cell (**C3**) of **System Coupling** system and select **Clear Generated Data** from the context menu.

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

17. Return to **System Coupling** by clicking on the **System Coupling** tab.

    a. In the **Outline of Schematic** under **Data Transfers** right-click **Data Transfer 5** and **Data Transfer 10** (with near wall temperature variables) and select **Delete** from the context menu.

        Click **Yes** in the confirmation dialog box that appears.

    b. Select **Data Transfer 2(Expert)** under **Data Transfers** in the **Outline of Schematic**.

	A	B
	Property	Value
1		
2	Source	
3	Participant	Static Structural
4	Region	FSI Barrel
5	Variable	Heat Flow
6	Target	
7	Participant	Fluid Flow (Fluent)
8	Region	interface_fluid_barrel_side
9	Variable	heatflow
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.5
13	RMS Convergence Target	0.01
14	Ramping	None

Properties of DataTransfer : Data Transfer 2 (Expert)

    i.   In the panel **Properties of DataTransfer** select **Heat Flow** from the **Variable** drop-down list under **Source**.

    ii. Select **heatflow** from the **Variable** drop-down list under **Target**.

    iii. Enter 0.5 for **Under Relaxation Factor**.

    c. Repeat the procedure for **Data Transfer 7(Expert)**.

Note

Heat flow will be sent from **Static Structural** to Fluent. Sending heat flow to a **Static Structural** system can be unstable because the structure immediately gives the final steady-state solution. This can produce large temperature changes during convergence. The temperature in Fluent responds more slowly when receiving a heat flow, particularly when a small number of Fluent iterations are used per coupling iteration.

---

    d. Select **Data Transfer 4(Expert)** under **Data Transfers** in the **Outline of Schematic**.

	A	B
	Property	Value
1	Property	Value
2	⊟ Source	
3	Participant	Fluid Flow (Fluent)
4	Region	interface_fluid_barrel_side
5	Variable	temperature
6	⊟ Target	
7	Participant	Static Structural
8	Region	FSI Barrel
9	Variable	Temperature
10	⊟ Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.5
13	RMS Convergence Target	0.01
14	Ramping	None

Properties of DataTransfer : Data Transfer 4 (Expert)

    i.   In the panel **Properties of DataTransfer** select **temperature** from the **Variable** drop-down list under **Source**.

    ii.   Select **Temperature** from the **Variable** drop-down list under **Target**.

    iii.   Enter 0.5 for **Under Relaxation Factor**.

    e.   Repeat the procedure for **Data Transfer 9(Expert)**.

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

19. After the solution starts select **Chart 2**.

> **Note**
>
> This chart shows the average value for **Data Transfer 2** and **Data Transfer 7**, which are now heat flow data transfers. It makes more sense to plot the heat flow sums.
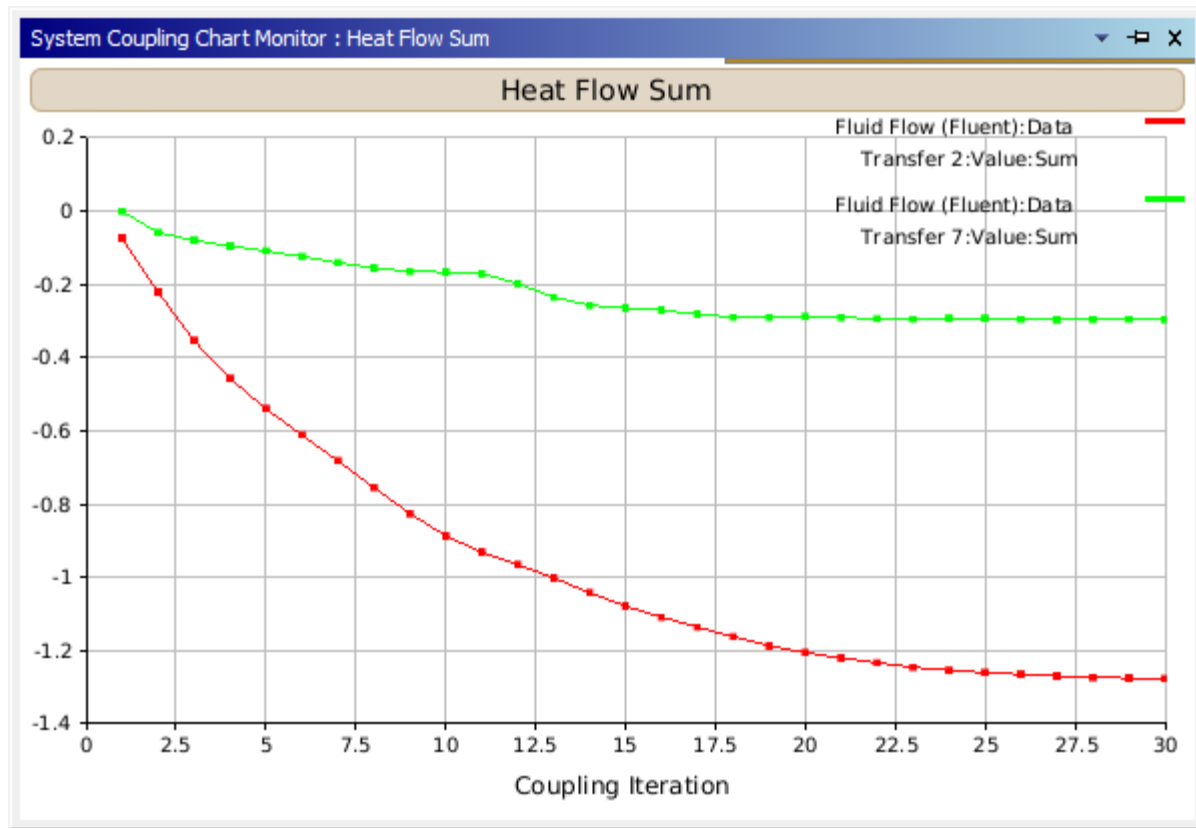
a. Right-click on **Chart 2** and select **Remove Variable** from the context menu and remove the two lines currently plotted.

b. Add the following variables to **Chart 2**:

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer 2** > **Value** > **Sum**

    and

    Add Variable > **Fluid Flow (Fluent)** > **Data Transfer 7** > **Value** > **Sum**

c. Rename **Chart 2** to `Heat Flow Sum`.

20. Rename **Chart 3** to `Average Temperature`.
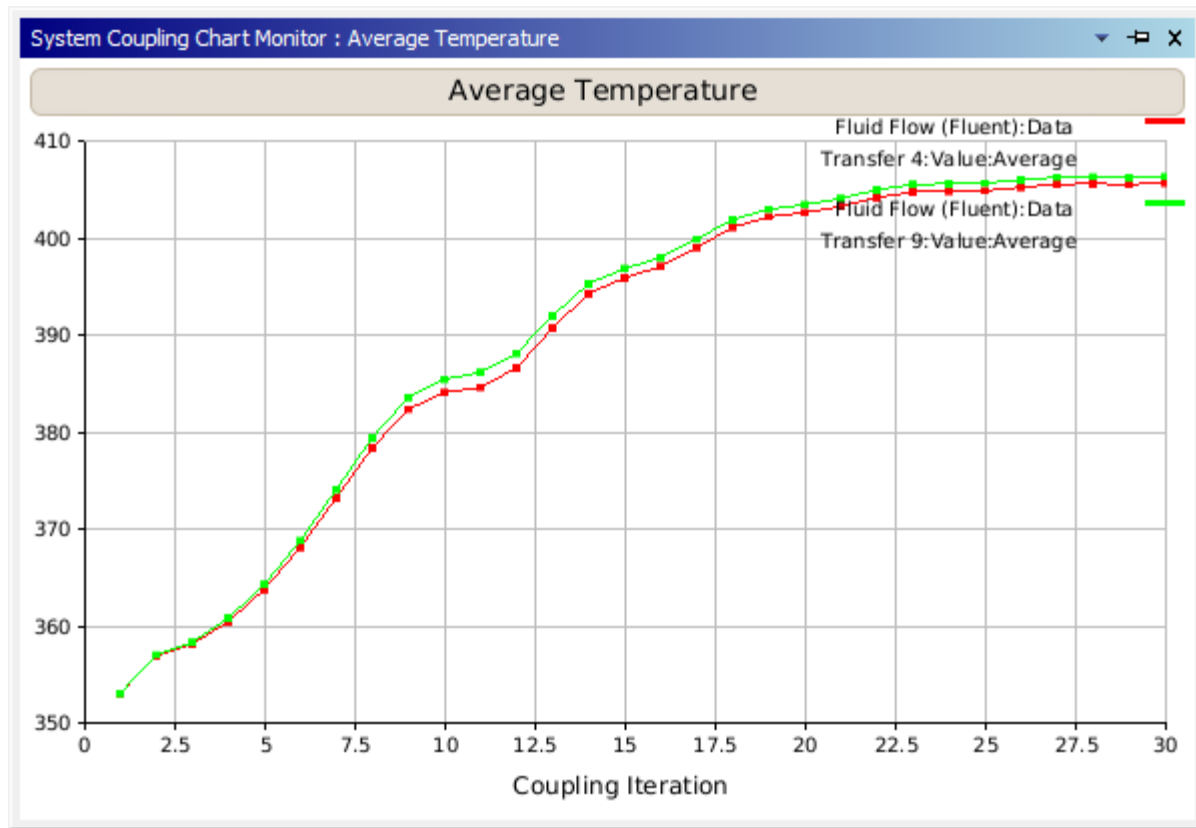
Note

Chart 3 shows the average for **Data Transfer 4** and **Data Transfer 9**, which is now temperature.

21. **Chart 4** should still show the maximum displacements on both the Fluent and **Structural** side. Rename it to `Maximum Displacements`.
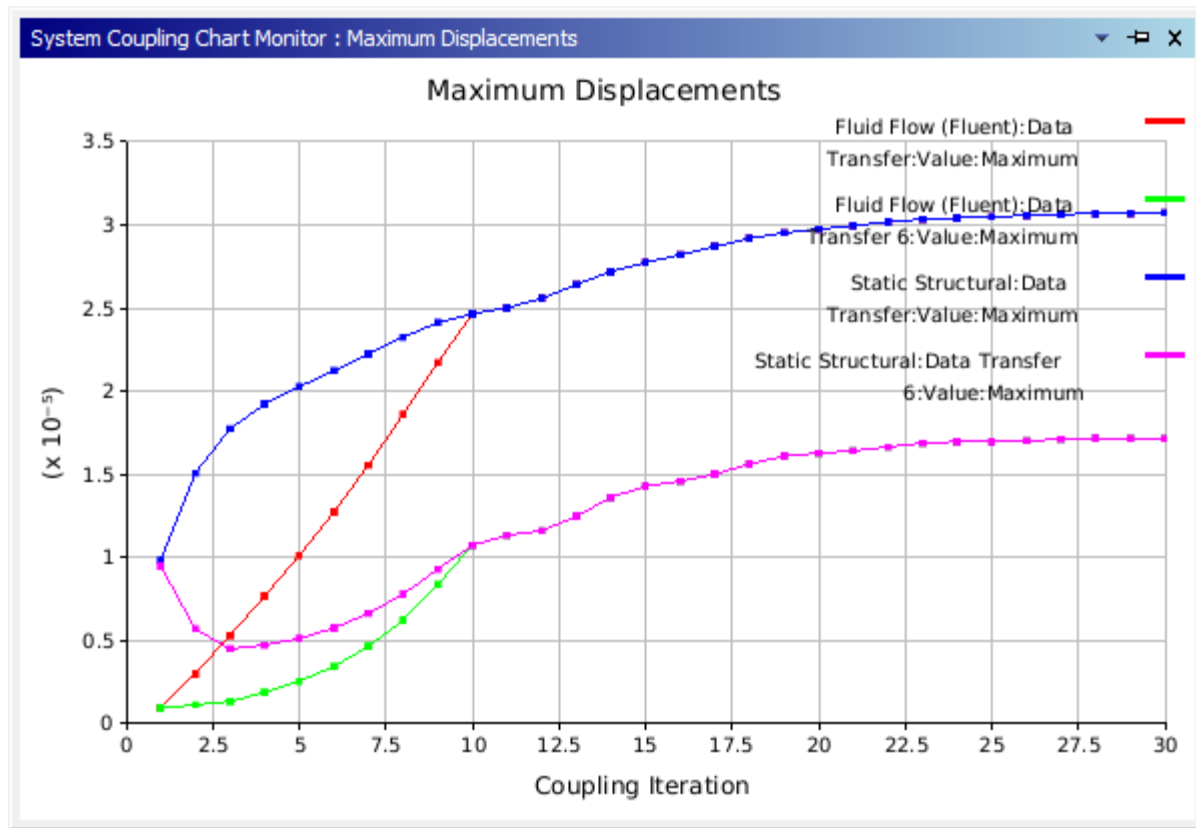
22. Right-click on **Chart Monitors** and select **Create Chart** from the context menu.

    a. Rename **Chart 5** to `Force Sum`.

    b. Add the following variables to **Force Sum**:

- **Add Variable** > **Fluid Flow (Fluent)** > **Data Transfer 3** > **Value** > **Sum**

- **Add Variable** > **Static Structural** > **Data Transfer 3** > **Value** > **Sum**

- **Add Variable** > **Fluid Flow (Fluent)** > **Data Transfer 8** > **Value** > **Sum**

- **Add Variable** > **Static Structural** > **Data Transfer 8** > **Value** > **Sum**
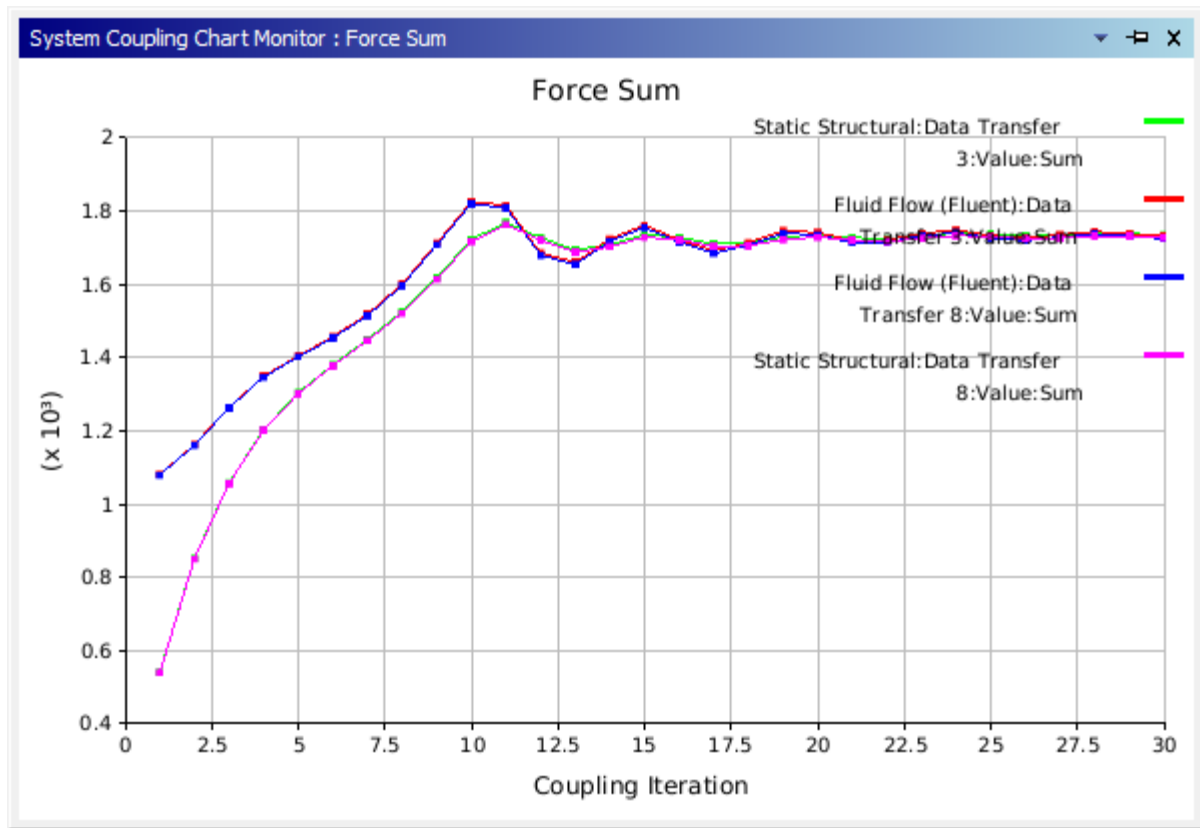
You can see the effect of under-relaxation on the force data.

The charts now show good convergence for all data transfers. The case ran for the full 30 coupling iterations because Fluent did not reach its convergence criteria. The Fluent residuals still show good convergence.

23. Save the project.

File > **Save**

2.7. Postprocessing

1. Return to the **Project Schematic** by clicking on the **Project** tab.

2. Double-click on the **Results** cell (**A7**) of the **Static Structural** system.

Note

You can post-process the structural solution to view deformations, stresses, etc. Next you'll create a temperature plot.

---

a. In the Mechanical window, right-click on **Solution** (**A6**) in the tree and select **Insert** > **User Defined Result** from the context menu.

b. In the **Details of User Defined Result** enter `TEMP` for **Expression**.

c. Right-click on **User Defined Result** in the tree under **Solution (A6)** and select **Evaluate All Results** from the context menu.

d. Select the **Auto Scale** Result | 46 (Auto Scale) ▼ option from the **Result** drop-down menu in the toolbar.
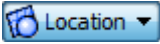

A: Static Structural
User Defined Result
Expression: TEMP
Time: 1

317.12 Max
290.77
264.42
238.08
211.73
185.38
159.04
132.69
106.35
80 Min
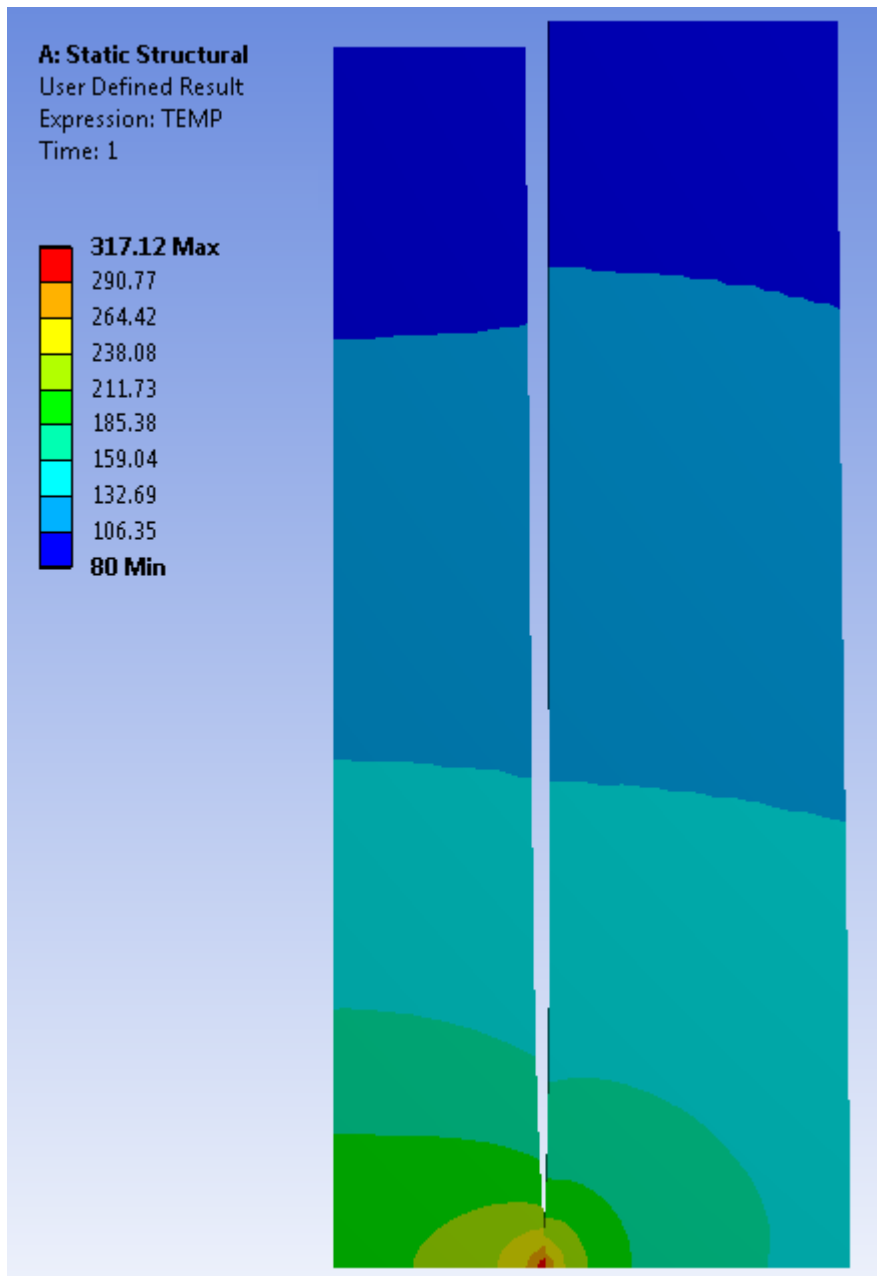
Note

The highest temperature occurs at the low-Z (outlet) side next to the gap. This is where the gap is at its narrowest and you can expect the most viscous heating in the fluid here.

---

3. Return to the **Project Schematic** and connect the **Solution** cell (**A6**) of the **Static Structural** system to the **Results** cell (**B6**) of **Fluid Flow (Fluent)** system.

4. Double-click on the **Results** cell (**B6**) of the **Fluid Flow (Fluent)** system to start CFD-Post with both the Fluent and Structural results.

a. Click on **Location** in the toolbar and select **Surface Group** from the drop-down list.

b. In the **Details of Surface Group** panel click on the **Location editor** button ![...] next to **Locations**.
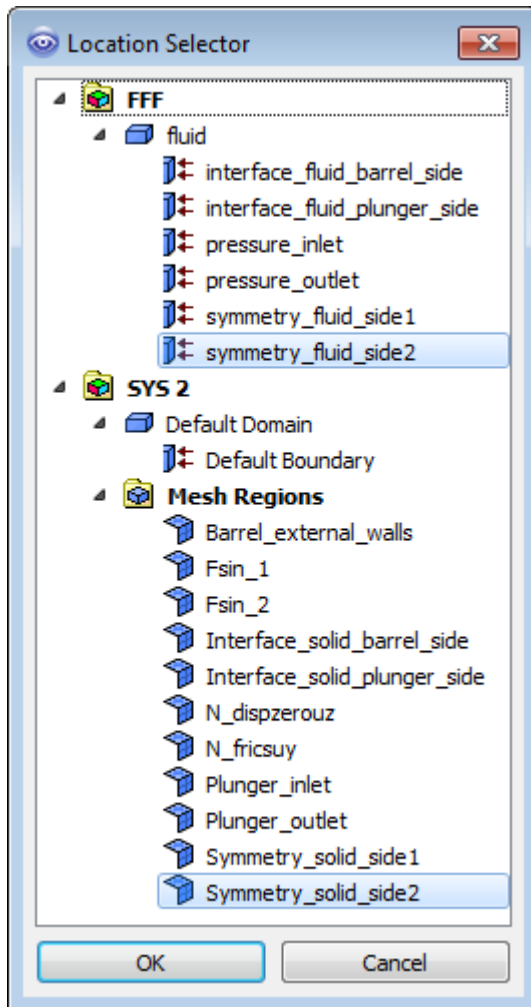
c. Select **symmetry_fluid_side2** under **fluid** and **Symmetry_solid_side2** under **Mesh Regions**.

d. In the **Color** tab of **Details of Surface Group** panel select **Variable** from the **Mode** drop-down list.

e. Select **Temperature** from the **Variable** drop-down list.

f. Click **Apply**.

g. Zoom in to see the fluid temperature in the gap near the outlet.

5. Create a chart which shows the change in the gap size.

a. Select **Polyline** from the **Location** drop-down list.

b. In the **Details of Polyline** panel select **Boundary Intersection** from the **Method** drop-down list in the **Geometry** tab.

c. Click on the **Location editor** button ![...] next to **Boundary List**.

- Select **Symmetry_solid_side1** under **Mesh Regions** in the **Location Selector** dialog box.

d. Click on the **Location editor** button [...] next to **Intersect With**.

- Select **Interface_solid_barrel_side** under **Mesh Regions** in the **Location Selector** dialog box.

e. Click **Apply**.

f. Click on **Insert** > **Chart** from the menu.

    i. In the **Details of Chart** panel in the **Data Series** tab select **Polyline 1** from the **Location** drop-down list.

    ii. In the **X Axis** tab select **Z** from the **Variable** drop-down list.

    iii. In the **Y Axis** tab select **Total Mesh Displacement X** from the **Variable** drop-down list.

    iv. Click **Apply**.

g. Create another **Polyline**

    i. In the **Details of Polyline** panel select **Boundary Intersection** from the **Method** drop-down list in the **Geometry** tab.

    ii. Click on the **Location editor** button [...] next to **Boundary List**.

        - Select **Symmetry_solid_side2** under **Mesh Regions** in the **Location Selector** dialog box.

    iii. Click on the **Location editor** button [...] next to **Intersect With**.

        - Select **Interface_solid_plunger_side** under **Mesh Regions** in the **Location Selector** dialog box.

h. Click **Apply**.

i. Click on **Chart** in the tree under **Report**.

    i. In the **Details of Chart** panel in the **Data Series** tab click on **New** [ ] to add another **Series**.

    ii. Select **Polyline 2** from the **Location** drop-down list.

    iii. Retain the selection of **Z** from the **Variable** drop-down list in the **X Axis** tab.

    iv. Retain the selection of **Total Mesh Displacement X** from the **Variable** drop-down list in the **Y Axis** tab.

j. Click **Apply**.

k.   Change the **Title** to `Change in Gap Size` in the **General** tab.

Chart 1

Change In Gap Size
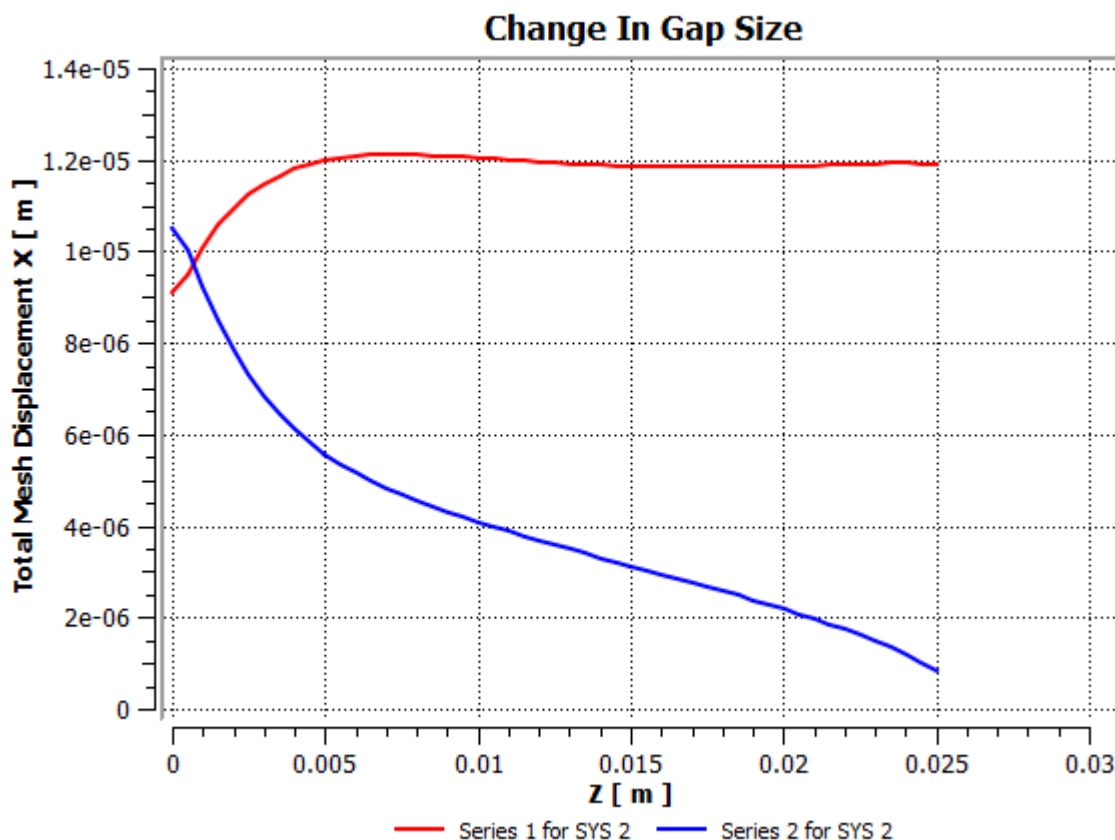
6.   You can look at other post-processing plots or quantities of interest. Some of the follow may be of interest:

   • Calculate the mass flow averaged temperature at the fluid outlet.

   • Plot the Total Mesh Displacement on one of the solid symmetry regions

   • Plot Pressure and Temperature on one of the Fluent FSI interface regions

7.   Save and close the project when you have finished post-processing.

2.8. Summary

This workshop has demonstrated how to solve an FSI model coupling Fluent with a thermal-structural solution in Mechanical. This coupling approach is needed when the deformations due to thermal stresses cannot be neglected in the CFD solution and will cause significant pressure and/or temperature changes in the fluid, which must be communicated back to the Mechanical solver.

The default thermal coupling sends a Heat Transfer Coefficient from Fluent to Mechanical. This is generally very stable, but can be slow to converge, particularly on low y+ CFD meshes. Heat Flow can be used instead of Heat Transfer Coefficients for thermal coupling. This is generally fast to converge but can be unstable.

If you find that neither of these thermal coupling options is suitable for your case then a third option is available. A CHT case can be solved in Fluent then the wall/wall-shadow boundaries at the fluid-solid interface in Fluent can be coupled with Mechanical. This means that both Fluent and Mechanical are solving the temperature field in the solid, with surface temperatures sent only one way from Fluent to Mechanical. Force is coupled between the fluid-side wall in Fluent and the Mechanical surface. Displacements are coupled between the Mechanical surface and both the wall/wall-shadow in Fluent. This approach provides a stable thermal solution and is fast to converge. Please contact your support provider for further information on this approach.