
Debugging System Coupling Simulations

This tutorial shows how to debug a 2-way coupled System Coupling simulation in Workbench. The aim is to understand a variety of common errors that may prevent you from solving a System Coupling simulation.

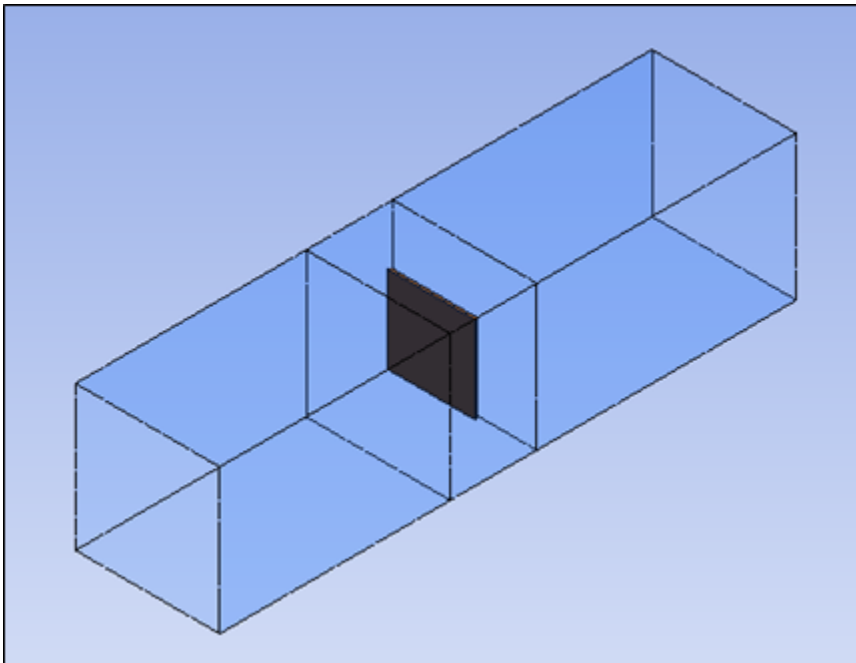
In this tutorial you will learn how to:

- Analyze the error-free case.
- Correct for interface mis-match (whether physical location or otherwise).
- Deal with large pressure forces and issues that arise from large time steps.
- Fix an FSI interface that is hitting another boundary in Fluent.

1. Problem Description

This tutorial considers the response of a buoyant plate in water undergoing constant acceleration. The flow is transient and the coupling involves 2-way FSI between Fluent and Mechanical.

Figure 50: Problem Schematic



The fluid region is a channel 200 mm high with a 50 mm x 50 mm base. A thin solid plate is suspended in the fluid channel. The density of both water and plate are identical, therefore the plate should have the same movements as the water. The water is initially at rest and is then accelerated at a rate of 100 m/s^2 by applying an inlet velocity that increases with time. The simulation is run for 10 time steps.

2. Setup And Solution

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

- 2.1. Preparation
- 2.2. Check Correct Solution
- 2.3. Mismatched Interfaces Project
- 2.4. Fluent Mesh Collapse

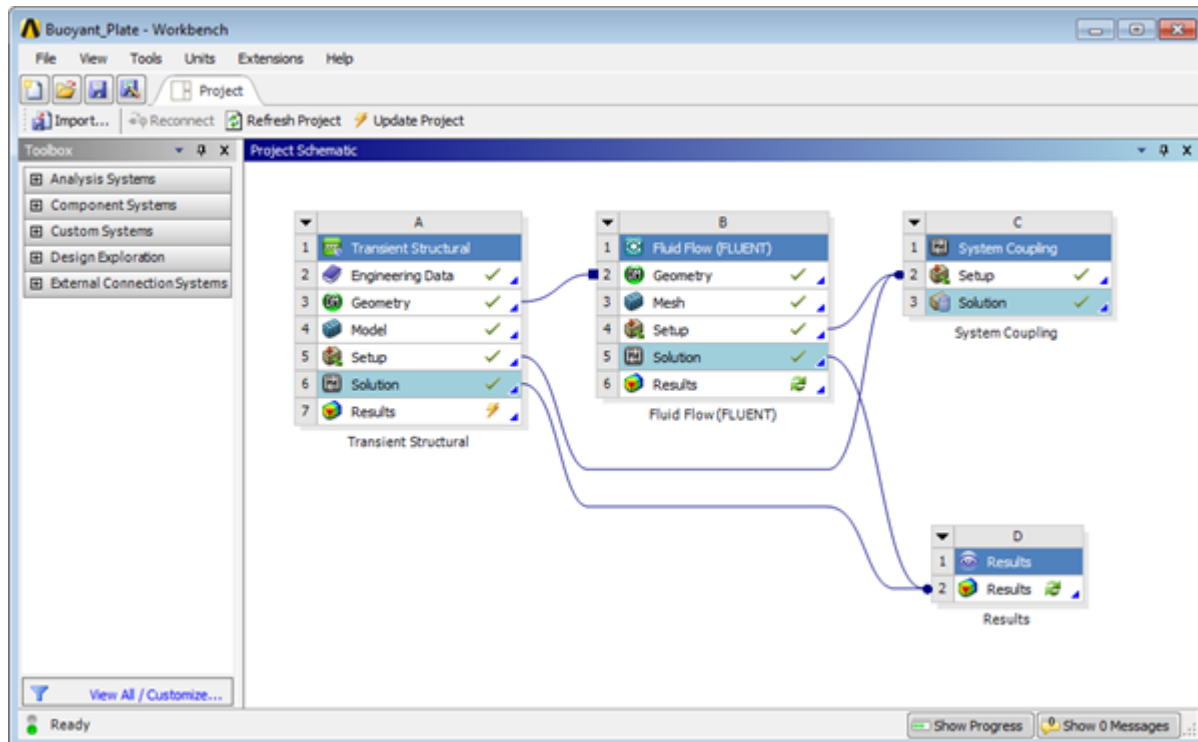
2.1. Preparation

1. Create a working folder on your computer.
2. Copy the files (Buoyant_Plate.wbpz, Buoyant_Plate_Mismatch.wbpz, and Buoyant_Plate_Mesh_Collapse.wbpz) to the working folder.

2.2. Check Correct Solution

First you will examine the correct solution to become familiar with the case.

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

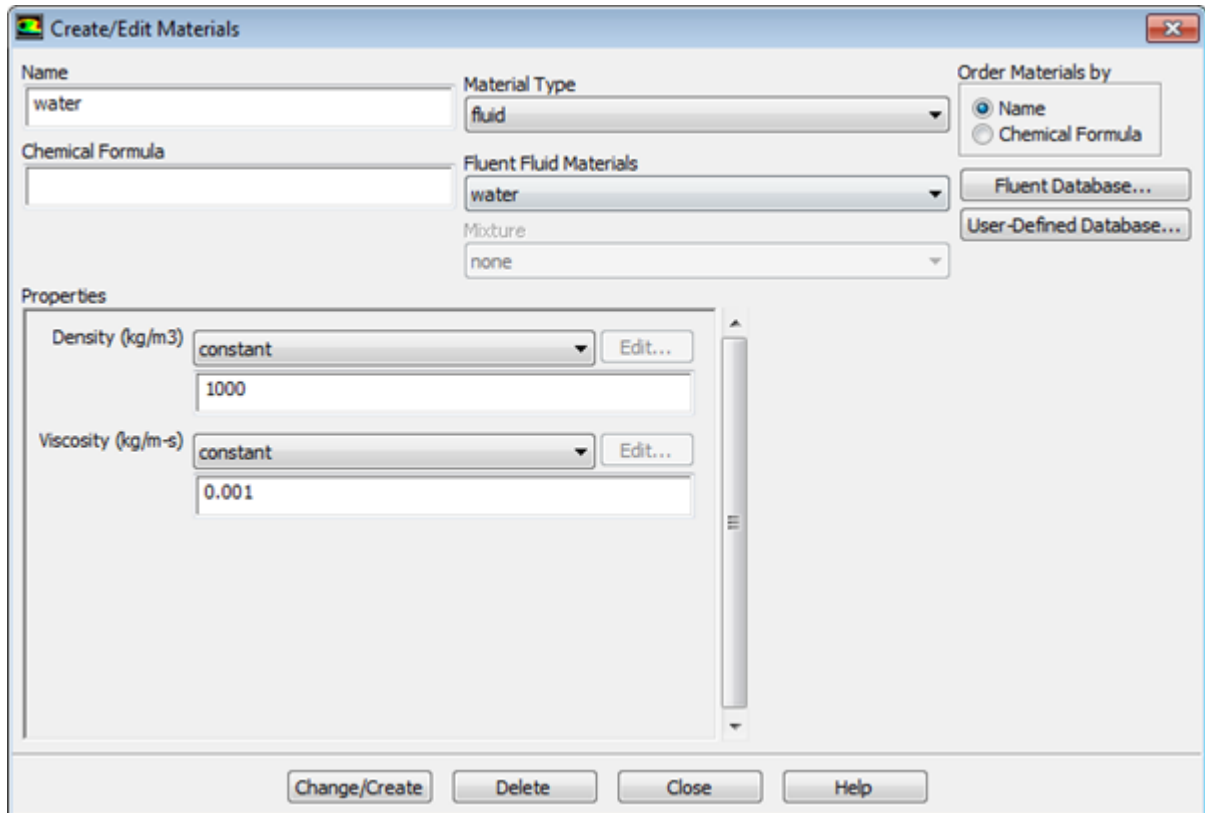


2. On the Workbench page, double-click on the **Solution** cell (B5) of the **Fluid Flow (FLUENT)** system.

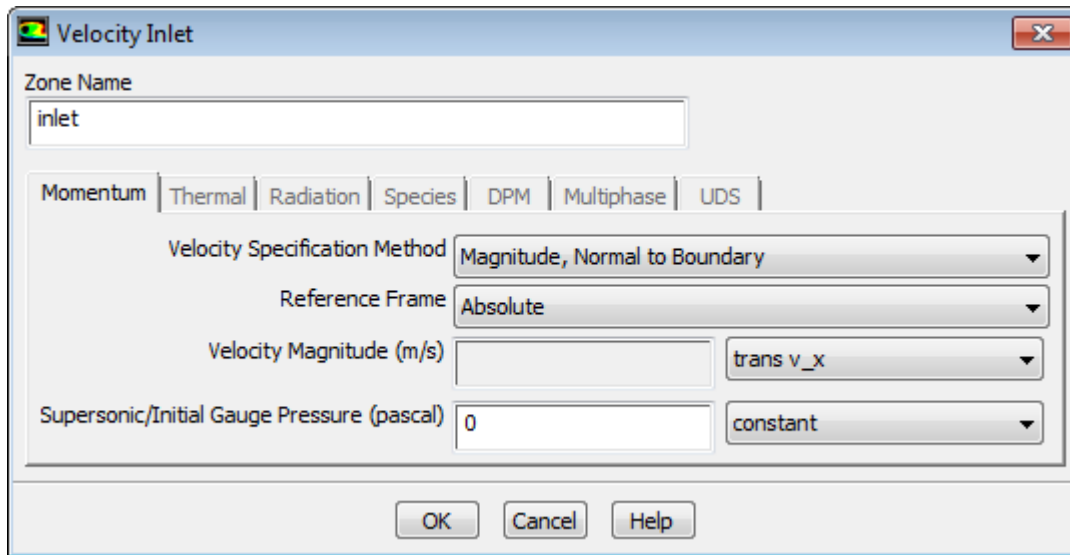
Note

This allows you to view the Fluent setup without causing downstream cells to go out-of-date and require a refresh, as would be the case if you opened the **Setup** cell.

- a. In the Fluent window, go to the **Materials** task page and verify that **water** has a **Density** of **1000 kg/m³** and **Viscosity** of **0.001 kg/(m-s)**.

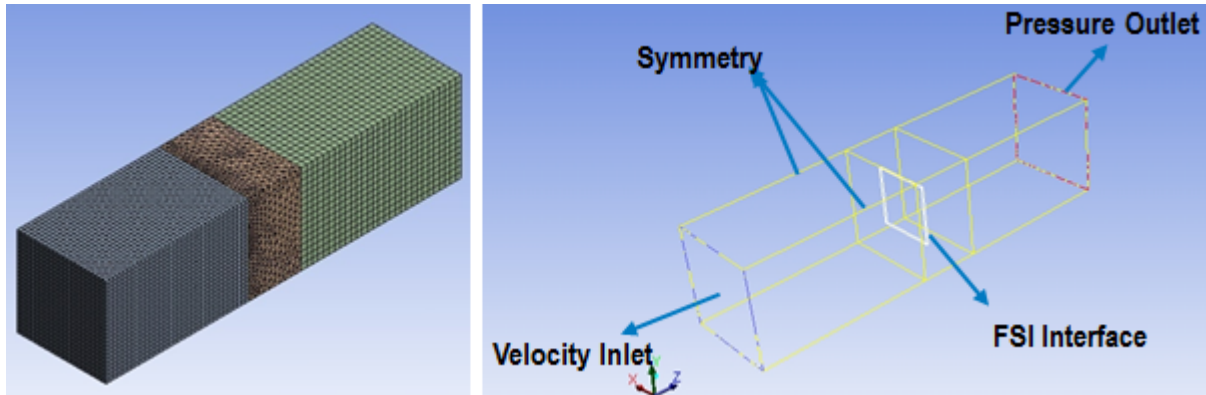


- b. In the **Boundary Conditions** task page double click **inlet** and verify that it is set to type **Velocity Inlet** and the **Velocity Magnitude** is defined by a UDF.



Note

The UDF applies an acceleration by defining the velocity as a function of time.



- c. Similarly verify that the **outlet** is set to type **Pressure Outlet** and the **Gauge Pressure** is set to **0 pascal**.
 - d. Close Fluent and return to the Workbench window.
3. On the Workbench page, double-click on the **Solution** cell (**A6**) of the **Transient Structural** system to open Mechanical.
 - a. In the tree expand **Geometry** and click on **solid** under it.

Details of "solid"	
+ Graphics Properties	
- Definition	
<input type="checkbox"/> Suppressed	No
Stiffness Behavior	Flexible
Coordinate System	Default Coordinate System
Reference Temperature	By Environment
- Material	
Assignment	Plate
Nonlinear Effects	Yes
Thermal Strain Effects	Yes
+ Bounding Box	
+ Properties	
+ Statistics	

You can see in the **Details of solid** panel that the **solid** body is assigned **Plate** material. This material was created in the **Engineering Data** cell (A2) of **Transient Structural** system in the **Project Schematic**.

- b. In the tree expand **Transient** and click on **Remote Displacement** under it.

Details of "Remote Displacement"	
+ Scope	
- Definition	
Type	Remote Displacement
<input type="checkbox"/> X Component	0. m (step applied)
<input type="checkbox"/> Y Component	0. m (step applied)
<input checked="" type="checkbox"/> Z Component	Free
<input type="checkbox"/> Rotation X	0. ° (step applied)
<input type="checkbox"/> Rotation Y	0. ° (step applied)
<input type="checkbox"/> Rotation Z	0. ° (step applied)
Suppressed	No
Behavior	Deformable
+ Advanced	

You can see in the **Details of Remote Displacement** panel that it allows motion only in the Z direction.

- c. Close the Mechanical window and return to the Workbench window.
4. On the Workbench page, double-click on the **Solution** cell (C3) of the **System Coupling** system.
- a. In the **Outline of Schematic** select **Analysis Settings**.

Properties of Analysis Settings		
	A	B
1	Property	Value
2	Analysis Type	Transient
3	Initialization Controls	
4	Coupling Initialization	Program Controlled
5	Duration Controls	
6	Duration Defined By	End Time
7	End Time [s]	0.01
8	Step Controls	
9	Step Size [s]	0.001
10	Minimum Iterations	1
11	Maximum Iterations	20

You can see that the step **End Time** is of **0.01 s**, and the **Step Size** is of **0.001 s**. This will result in a total of 10 time steps. There is a minimum of **1** coupling iteration per time step, and a maximum of **20**.

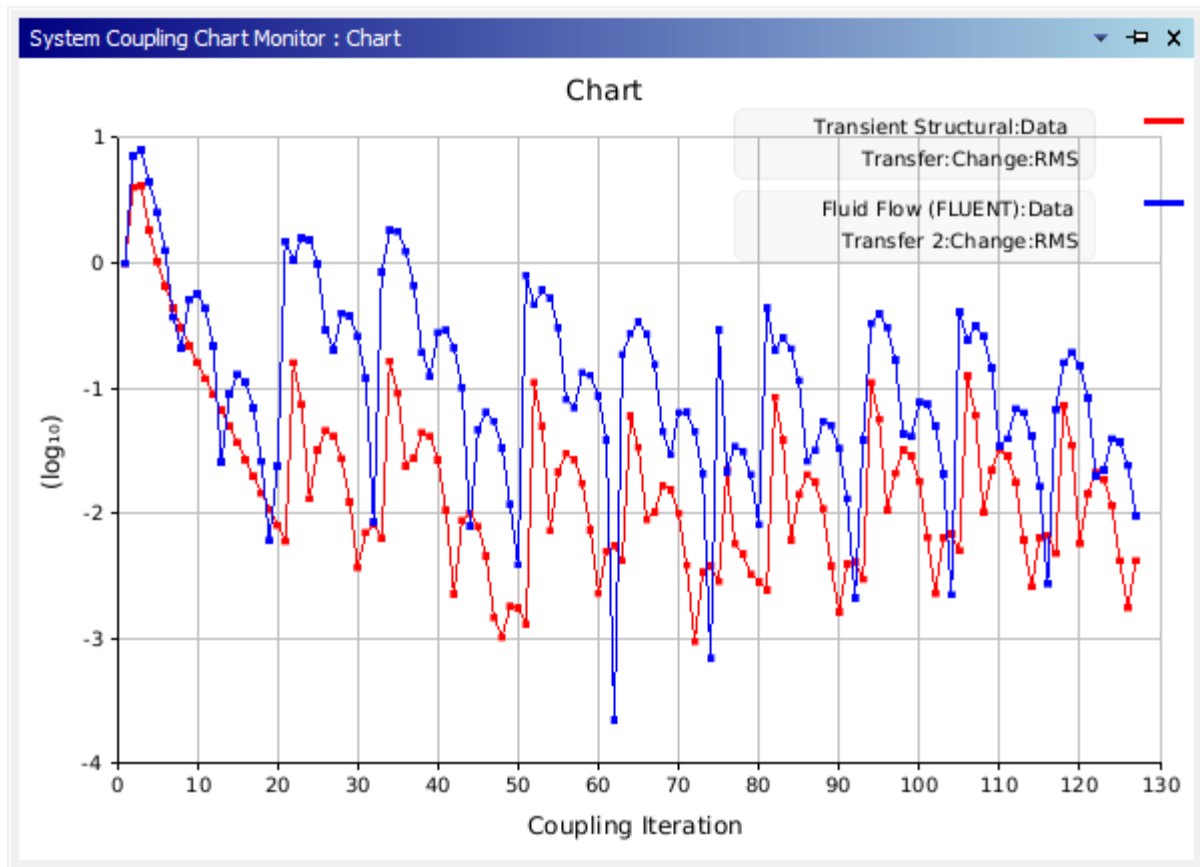
- b. Expand **Data Transfers**, and examine the details for **Data Transfer** and **Data Transfer 2**.

Properties of DataTransfer : Data Transfer		
	A	B
1	Property	Value
2	Source	
3	Participant	Fluid Flow (FLUENT)
4	Region	wall_plate
5	Variable	force
6	Target	
7	Participant	Transient Structural
8	Region	Fluid Solid Interface
9	Variable	Force
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.25
13	RMS Convergence Target	0.01
14	Ramping	None

Properties of DataTransfer : Data Transfer 2		
	A	B
1	Property	Value
2	Source	
3	Participant	Transient Structural
4	Region	Fluid Solid Interface
5	Variable	Incremental Displacement
6	Target	
7	Participant	Fluid Flow (FLUENT)
8	Region	wall_plate
9	Variable	displacement
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.25
13	RMS Convergence Target	0.01
14	Ramping	None

Force is transferred from Fluent to Mechanical. Displacement is transferred from Mechanical to Fluent. An **Under Relaxation Factor** of **0.25** has been used in both cases. Note that this is excessive under relaxation, and indicates that solution stabilization should be considered.

- c. Click on **Solution** in the tree.



- i. Click on **System Coupling** under **Solution Information** in the tree and review the results, including the **MAPPING SUMMARY** near the start of the **System Coupling** log file.
- ii. Click on **Transient Structural** under **Solution Information** in the tree and check the log file. You can see that in most cases both force and displacement converge at Equilibrium Iteration 1.
- iii. Click on **Fluid Flow (FLUENT)** under **Solution Information** in the tree and check the log file. You can see that at the end, by the end of each time step Fluent is well converged.

5. Close **System Coupling** and return to the Workbench window.

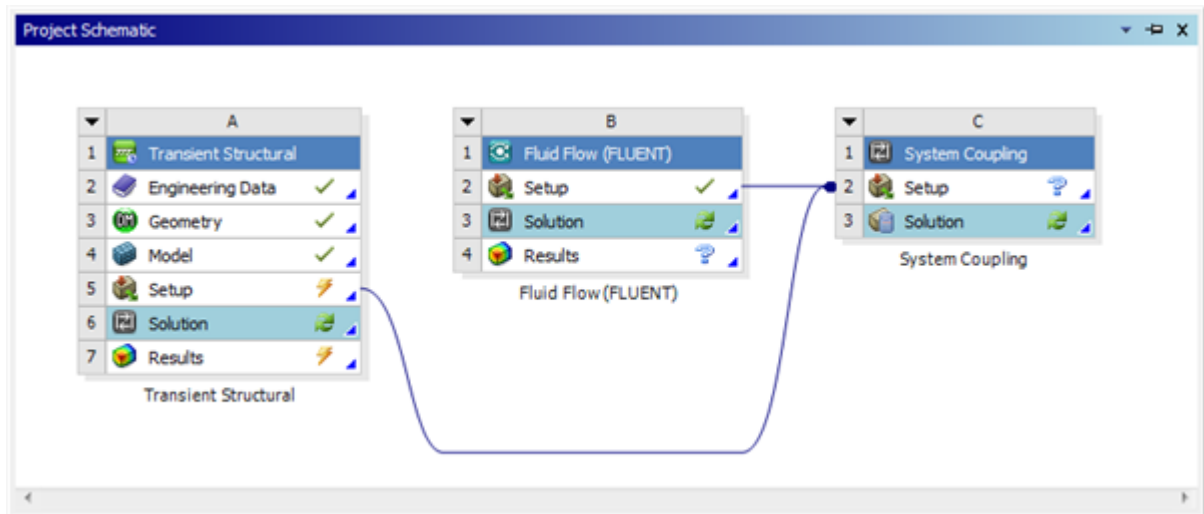
2.3. Mismatched Interfaces Project

Mismatching fluid-solid interfaces are a relatively common phenomenon that occur when fluid and structural geometry originate from separate files. Differences in the physical location of the interfaces can cause problems when it comes to mapping. Next you will learn how to identify problems associated with a mismatched interface.

Mismatched interfaces can be located by either manual inspection or by using the **System Coupling** mapping summary.

1. In the ANSYS Workbench window select **File >Restore Archive...** from the menu.
 - a. Select `Buoyant_Plate_Mismatch.wbpz` from your working folder.

- b. Save to your working folder.



Note

In this case a mesh has been imported to Fluent (there is no **Geometry** or **Mesh** cell), suggesting the fluid and solid geometries were created independently.

2. Update the **Setup** cell (A5) of **Transient Structural** system to create its input file.
3. Right-click on the **Setup** cell (C2) of **System Coupling** system and select **Refresh** from the context menu.
4. Double-click the **Solution** cell of **System Coupling** system to open it and then click **Update**.

Note

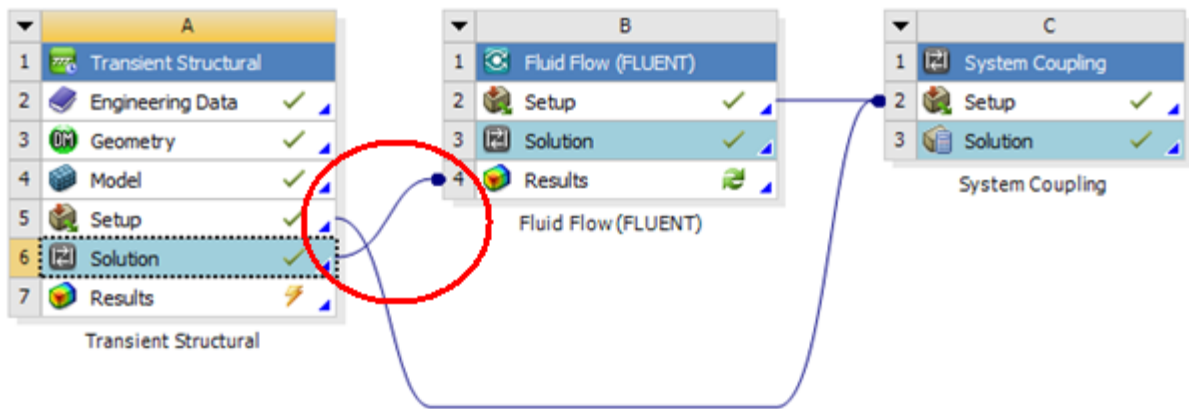
System Coupling will run, however the chart showing the convergence of the data transfers shows full convergence is achieved immediately.

- Expand **Solution Information** in the tree and click on **System Coupling**. You can see in the log file that is displayed, that at each **COUPLING STEP** only 1 **COUPLING ITERATION** is performed and full convergence is achieved. This suggests the data is not being transferred correctly.

MAPPING SUMMARY		
Data Transfer Diagnostic	Source Side	Target Side
Data Transfer 2 Percent Nodes Mapped	N/A	100
Data Transfer Percent Nodes Mapped	0	0
Percent Area Mapped	0	0

Also if you scroll to the beginning of the log file, under **MAPPING SUMMARY** section you can see that the **Data Transfer** has a **Mapped Area** of **0**, confirming mismatched interfaces.

5. Click on the **Project** tab and return to the Workbench window.
 - Connect the **Solution** cell (**A6**) of the **Transient Structural** system to the **Results** cell (**B4**) of the **Fluid Flow (FLUENT)** system.

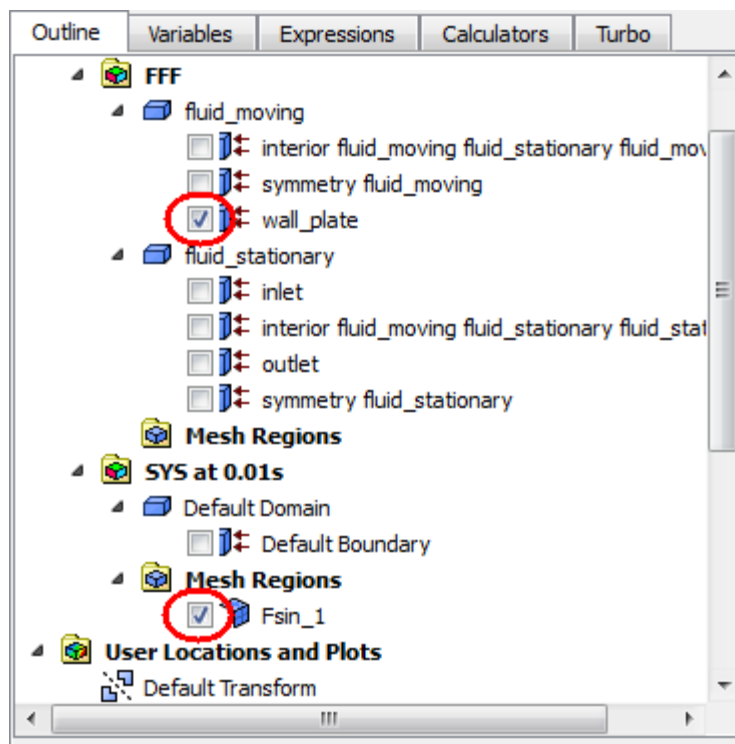


6. Double-click on the **Results** cell (**B4**) of the **Fluid Flow (FLUENT)** system to open **CFD-Post**.
 - a. In **CFD-Post** enable **wall_plate** under **fluid_moving** beneath **FFF** in the tree.

Note

This displays the plate in the Fluent FSI region.

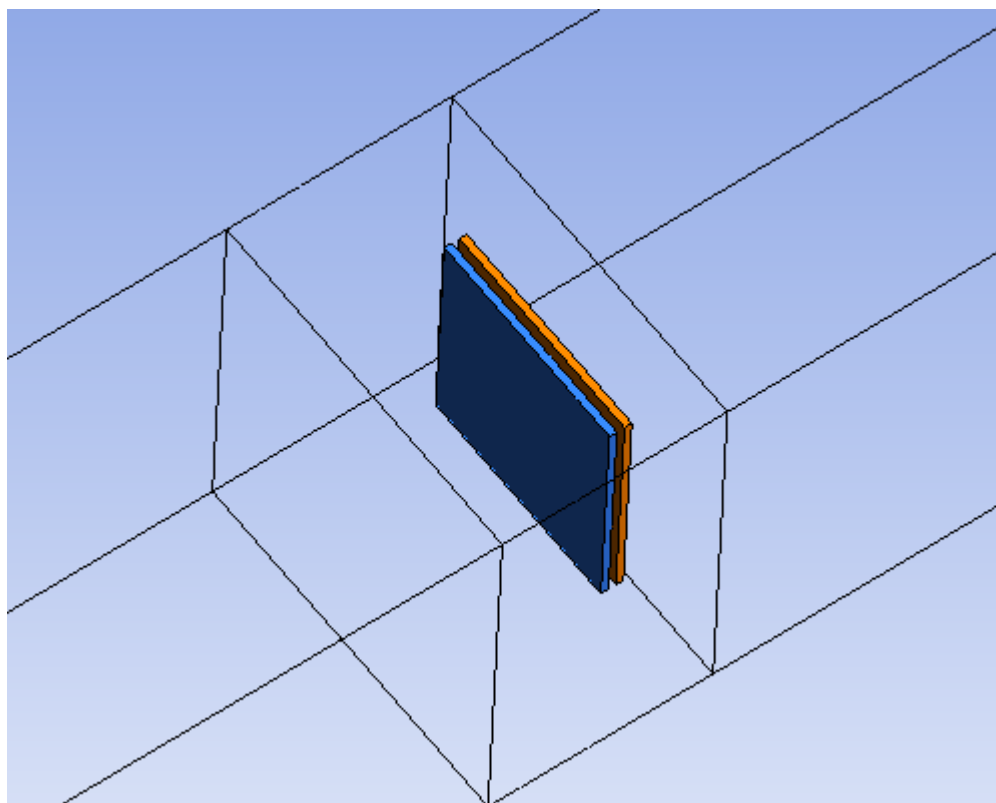
- b. Enable **Fsin_1** under **Mesh Regions** beneath **SYS** in the tree.



Note

This displays the plate in the Mechanical FSI region.

Figure 51: Mismatched Plates



As you can see from the [Figure 51: Mismatched Plates \(p. 10\)](#) it is obvious that the plate does not line up correctly between the Fluent and Mechanical meshes.

- c. Close **CFD-Post** and return to the **Project Schematic**.

Note

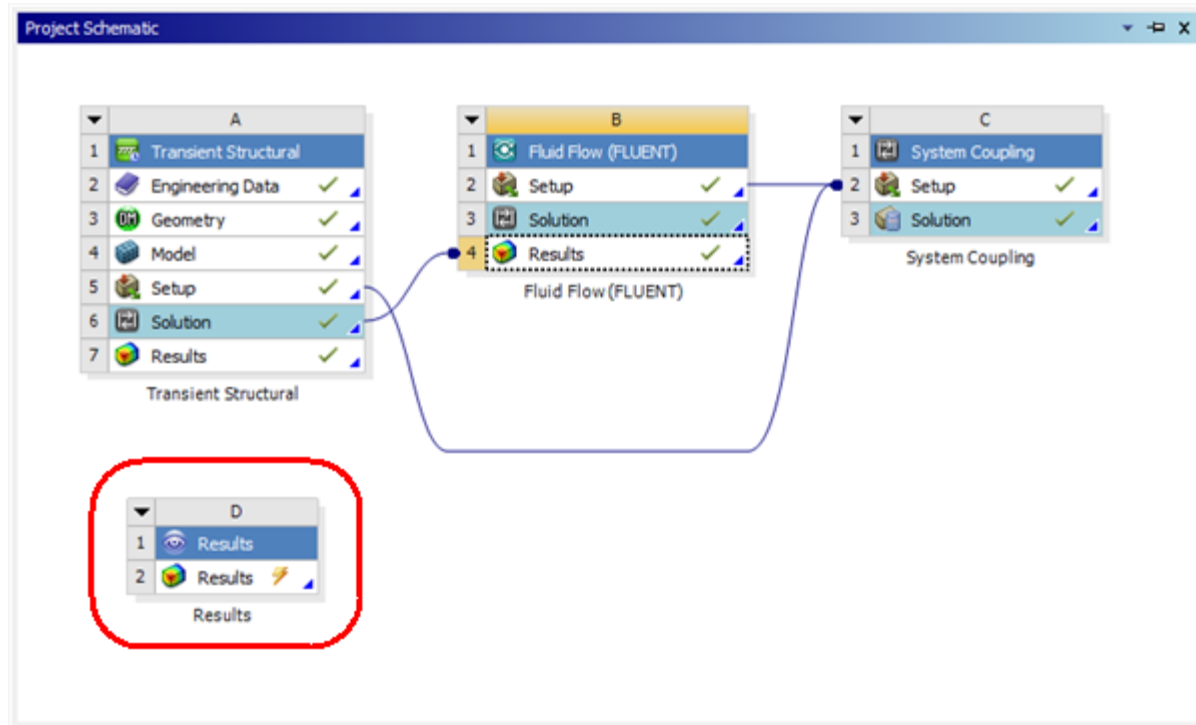
Some cases may not get as far as writing out results files, so you cannot load the results into CFD-Post to examine the interface regions. **System Coupling** can also output the interface meshes at the mapping stage. Next you will work through viewing the interface meshes using this approach.

7. Right-click on the **Solution** cell (**C3**) of **System Coupling** and select **Reset** from the context menu. Click **OK** in the warning dialog box that appears.
8. Double-click the **Setup** cell (**C2**) of **System Coupling**.
 - a. In the **Outline of Schematic** in **System Coupling** right-click on **Expert Settings** under **Execution Control** and select **Add Property > DumpInterfaceMeshes** from the context menu.
 - b. In the **Properties of Expert Settings** panel ensure that **DumpInterfaceMeshes** is set to **CFD-Post**.
 - c. Click **Update** to run the solution.

Note

The **MAPPING SUMMARY** again shows that no nodes were mapped.

- d. After the solution is complete click on the **Project** tab to return to the **Project Schematic** window.
9. In the Workbench window enable **View > Files** if the **Files** window is not visible.
 - Scroll down in the **Files** view till you see the four **.csv** files written out by **System Coupling**.
The four files represent the source and target meshes for the two **Data Transfer** objects. Next you will load these files into **CFD-Post**.
10. From the **Component Systems** toolbox drag-and-drop a standalone **Results** system onto the **Project Schematic**.



11. Double-click the **Results** cell (D2) to open **CFD-Post**.

a. In **CFD-Post** select **File > Import > Import Surface or Line Data....**

Leave the **Import** window open for now. Instead of browsing for the file, you are going to copy the directory location and paste it into the **Import** window.

b. Return to the **Project Schematic**, right-click in the **Location** column for one of the **.csv** files in the **Files** view and select **Copy** from the context menu.

c. Return to the **Import** dialog box in **CFD-Post** and paste the directory path into the **File** text box and click the **Open** button next to it.

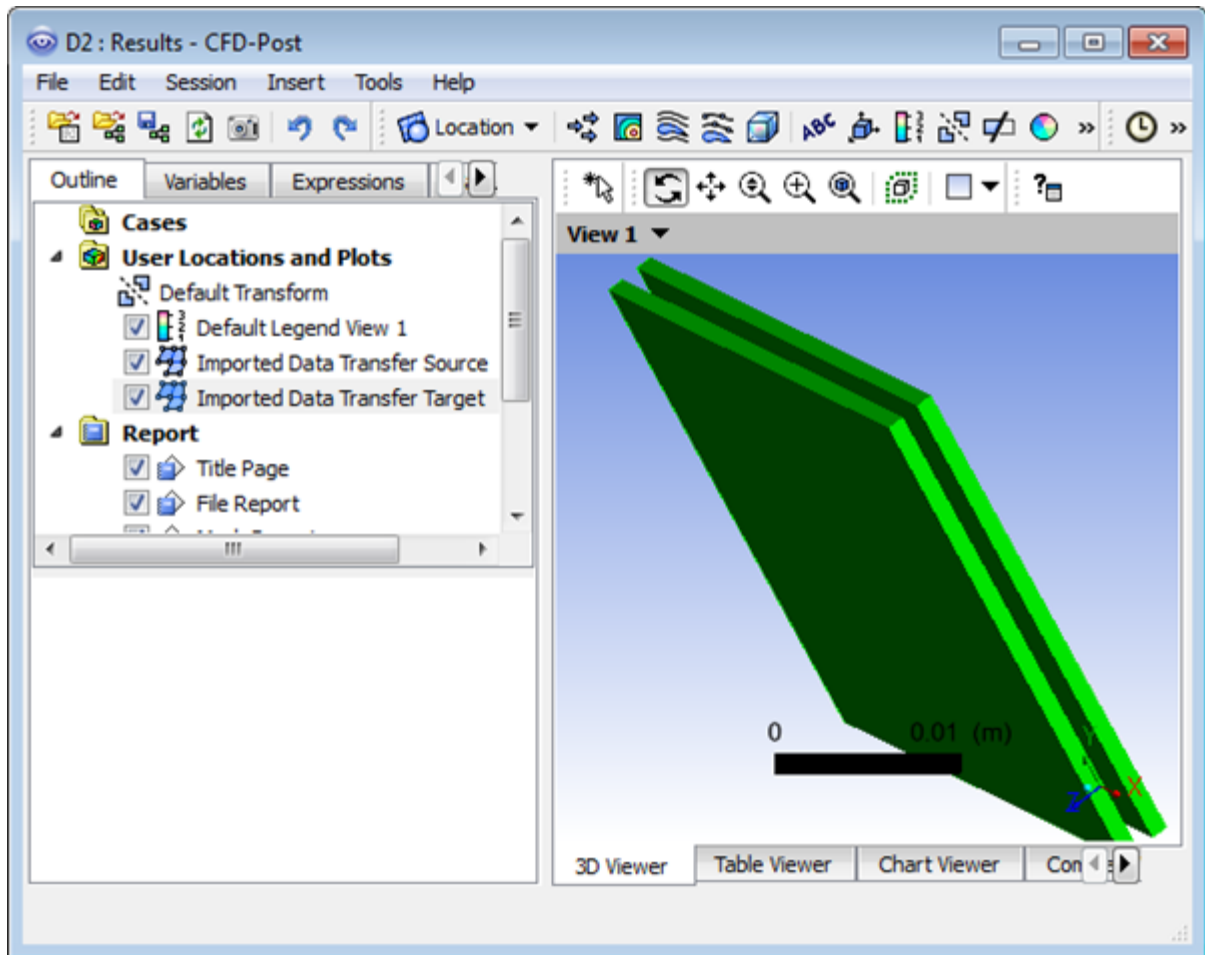
- In the **Import CFX Data File** dialog box select the file `Data Transfer source.csv` from the list and click **Open**.

d. Click **OK** in the **Import Surface or Line Data** dialog box.

In **CFD-Post** you will now see a **Imported Data Transfer Source** object under **User Locations and Plots** in the tree.

e. Enable **Imported Data Transfer Source** and check the display in the viewer.

f. Repeat the import process for the file `Data Transfer Target.csv` (the path will have been saved in the import window) and enable the visibility check box again.



You will now have objects for the source and target from **Data Transfer**. You can see that the meshes do not align. In some cases the meshes will partially overlap and it is useful to view the mapped and unmapped areas.

- g. Double-click on **Imported Data Transfer Source** to edit it.
 - i. In the **Details of Imported Data Transfer Source** panel click on the **Color** tab and select **Variable** from the **Mode** drop-down list.
 - ii. Ensure that **Mapped on Imported Data Transfer Source** is selected from the **Variable** drop-down list.
 - iii. In the **Render** tab enable **Show Mesh Lines**.
 - iv. Click **Apply**.

You now see the mesh for the source side and it will be colored to show the mapped nodes. A value of 1 means mapped while 0 means unmapped. In this case all nodes are unmapped. To correct this problem the geometries should be made consistent, which will not be done here.

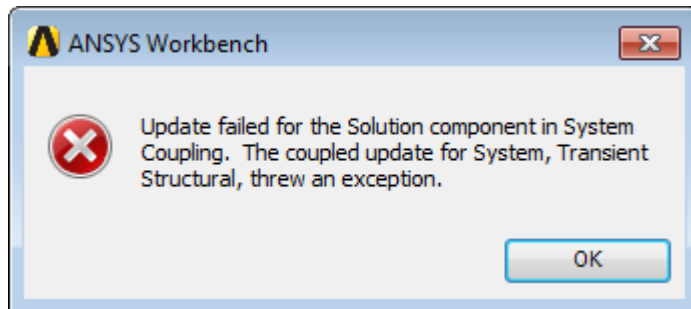
12. Close **CFD-Post** and save the project.

2.4. Fluent Mesh Collapse

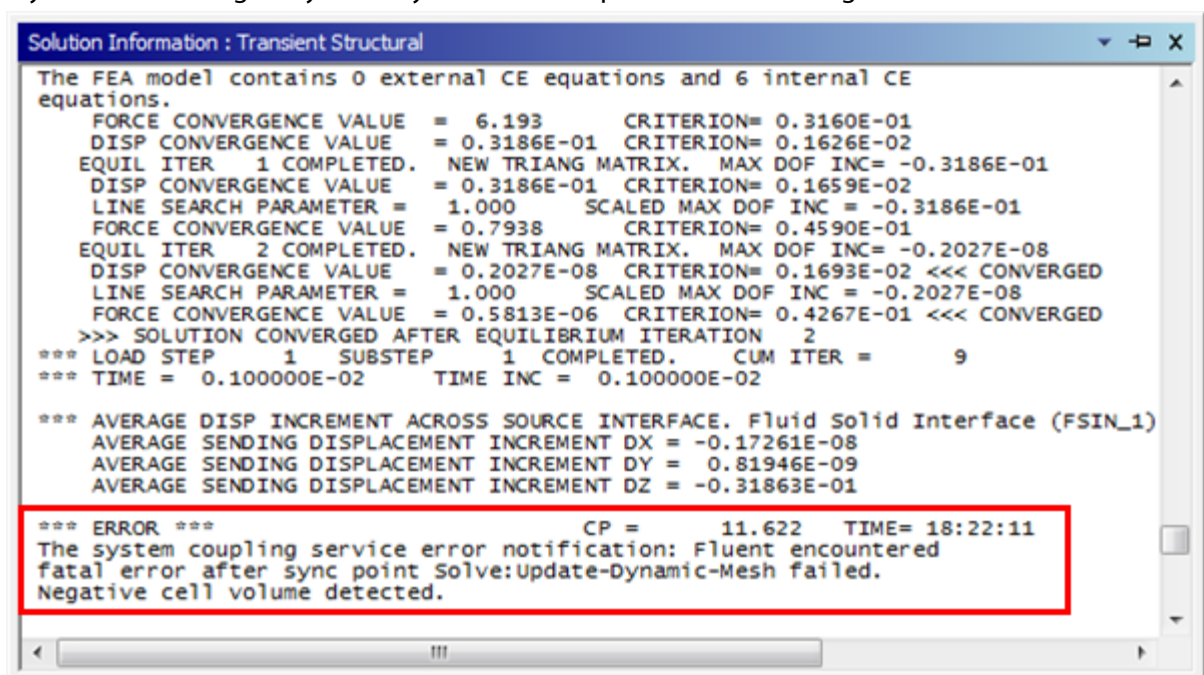
A common error in FSI simulations is a collapsed mesh in Fluent or an element formulation error due to excessive element distortion in the Mechanical APDL solver. When this happens it is often because unreasonable displacements were received by Fluent or unreasonable forces were sent to Mechanical APDL. In this case you will learn debugging a collapsed mesh in Fluent.

1. In the Workbench window select **File > Restore Archive...** from the menu.
 - a. Select `Buoyant_Plate_Mesh_Collapse.wbpz` from your working folder.
 - b. Save to your working folder.
2. Update the **Setup** cell (C2) of **System Coupling**.
3. The case is now ready to run. Double-click the **Setup** cell (C2) of **System Coupling** and then click **Update** to start the solution.

You can see that the solution will fail after about six **COUPLING ITERATIONS** in the first **COUPLING STEP**. The error dialog box that pops up says that the **Transient Structural** solver failed. Check the **Transient Structural** log file. In some cases the Fluent solver may also fail first. In this case examine the **Fluid Flow (FLUENT)** log file.

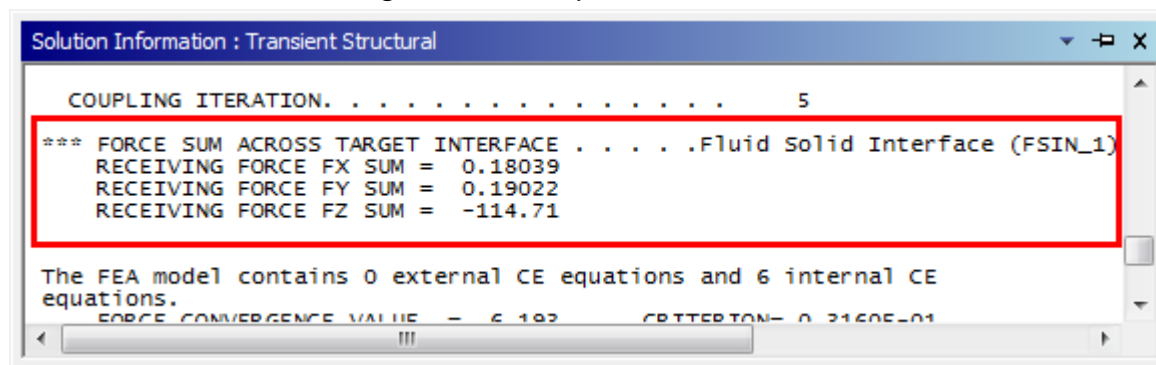


- a. If you check the log it says that dynamic mesh update failed and negative cell volume detected.



Since the Fluent mesh collapsed you should check if reasonable forces were sent to Mechanical APDL before the failure. In this case no monitor data was created; you will correct that in a moment, but the forces are also reported in the structural solver output.

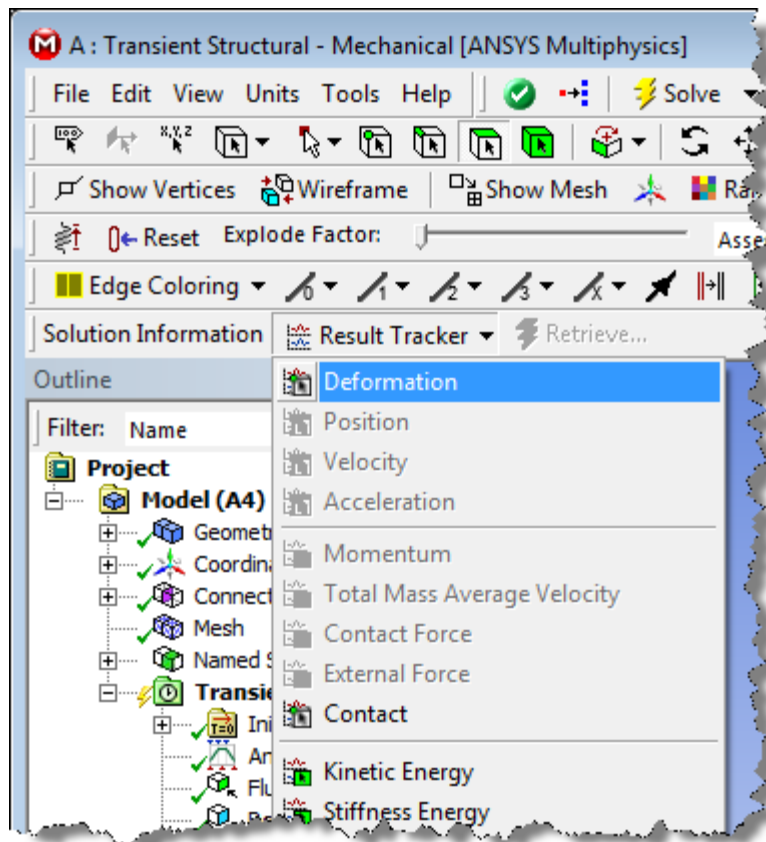
- b. In the **Transient Structural** log file check the part on the last set of forces received from Fluent.




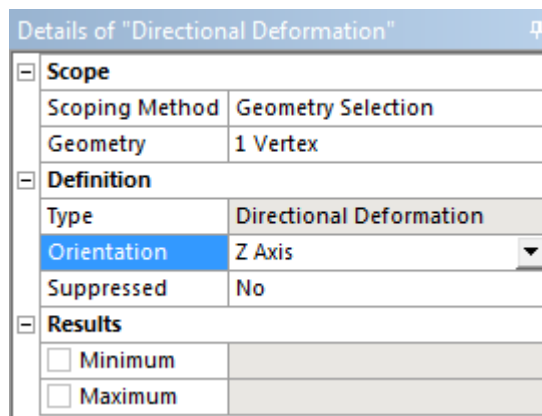
- c. If you scroll up further in the log file you will see that the forces received were oscillating and increasing in magnitude.

You can see that the Mechanical APDL solver received a force of about **-114 N** from the Fluent solution at the start of the sixth **COUPLING ITERATION**. It then successfully calculated the new displacements and sent them back to Fluent, at which point Fluent failed. You need to check whether -114 N is a reasonable force. The area of one side of the plate is $9\text{e-}4 \text{ m}^2$, so to generate a force of -114 N you need a pressure difference of about 130 kPa (1.3 atm). This is high. Next you will verify that the displacements Mechanical APDL is calculating are too large by using a Results Tracker.

4. Return to the Workbench window by clicking on the **Project** tab.
 - a. In the **Project Schematic** right-click on the **Solution** cell (**C3**) of **System Coupling** and select **Reset** from the context menu. Click **OK** in the warning dialog box that appears.
 - b. Double-click the **Setup** cell (**A5**) of **Transient Structural** to open Mechanical.
5. In the Mechanical window click on **Solution Information** in the tree.
 - a. In the **Solution Information** toolbar select **Deformation** from the **Result Tracker** drop-down list.



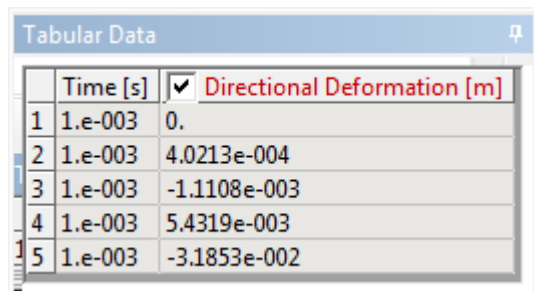
- b. Click on the **Vertex/Node** button in the toolbar  and select any corner vertex of the geometry in the graphics window.
- c. In the **Details of Directional Deformation** panel click **Apply** next to **Geometry**.



- d. In the **Details of Directional Deformation** panel select **Z Axis** from the **Orientation** drop-down list.
6. Return to the Workbench window and in the **Project Schematic** right-click on the **Setup** cell (C2) of **System Coupling** system and select **Update** from the context menu.
7. Update the **Solution** cell (C3) of the **System Coupling** system.

The case will fail in the same way it failed earlier. You can open **System Coupling** to check.

8. In the Mechanical window click on **Directional Deformation** under **Solution Information** beneath **Solution** in the tree. In the **Worksheet** tab check the **Tabular Data** panel.



	Time [s]	<input checked="" type="checkbox"/> Directional Deformation [m]
1	1.e-003	0.
2	1.e-003	4.0213e-004
3	1.e-003	-1.1108e-003
4	1.e-003	5.4319e-003
5	1.e-003	-3.1853e-002

Your value may differ depending on which vertex you selected. In this case, it shows that the maximum deformation calculated by the structural solver is about **-3.2e-2** m. This is roughly the same as the width/height of the plate, so the plate has moved much further in the first time step than expected.

Note

To debug this case further you could change the **Analysis Settings** in **System Coupling** so that only a single time step with five **COUPLING ITERATION** are requested. After the fifth **COUPLING ITERATION** the solution would finish, writing out results files. Examining the results files would show the plate moved the distance reported by the Mechanical **Results Tracker**.

The important question to ask here is why are the forces/displacements oscillating with increasing magnitude? The problem here is due to unstable coupling between the force and displacement on the interface. The displacements calculated are over-shooting the correct position, resulting in an increasing force in Fluent that pulls the plate back too far in the opposite direction.

Using under-relaxation on the data transfers can help here. This will make the solution more stable at the expense of more **COUPLING ITERATIONS** to reach convergence. Note that some cases remain unstable even when under-relaxation is used; for these cases see solution **2022119** in the **Knowledge Resources > Solutions** section on the ANSYS Customer Portal.

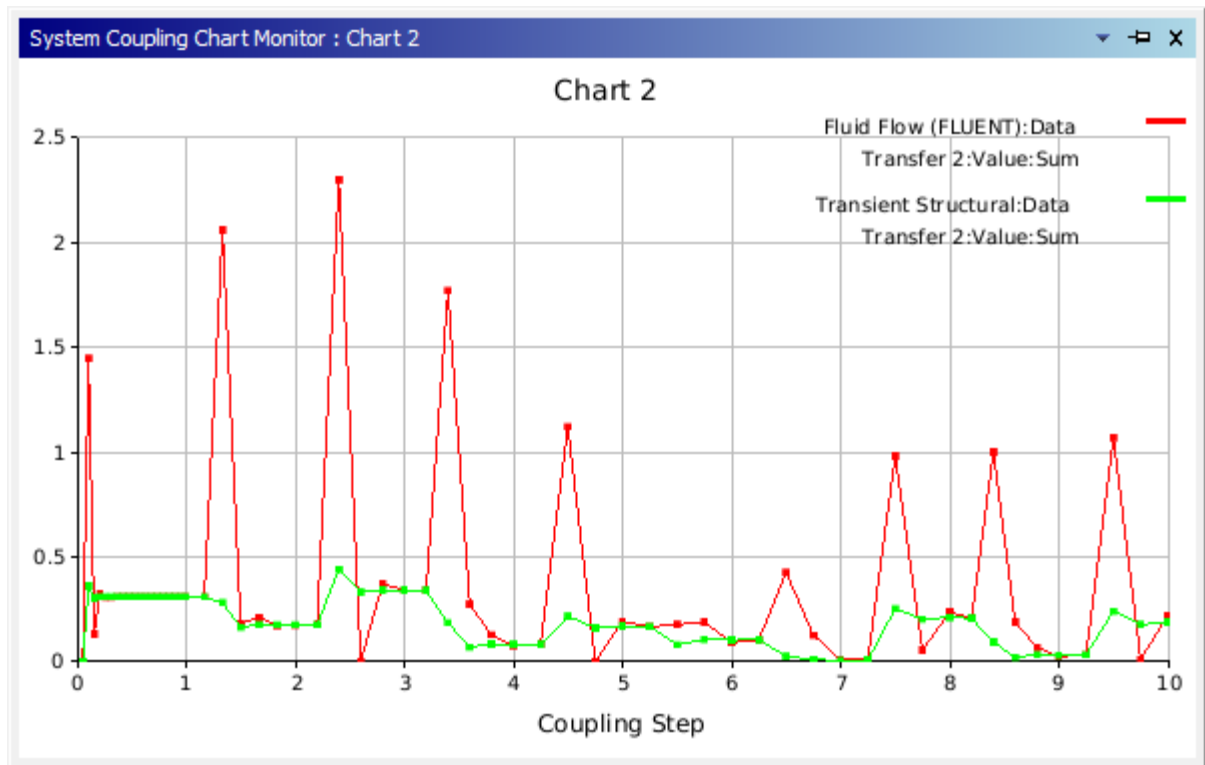
9. Return to the **Project Schematic** and then right-click on the **Solution** cell (C3) of **System Coupling** and select **Reset**.
10. Clear the **Messages** window.
11. Double-click on the **Setup** cell (C2) of **System Coupling**.
- In the **System Coupling** tab select **Data Transfer 2** from the tree.
 - In the **Properties of DataTransfer** panel enter 0.25 for the **Under Relaxation Factor**.

Properties of DataTransfer : Data Transfer 2		
	A	B
1	Property	Value
2	Source	
3	Participant	Fluid Flow (FLUENT)
4	Region	wall_plate
5	Variable	force
6	Target	
7	Participant	Transient Structural
8	Region	Fluid Solid Interface
9	Variable	Force
10	Data Transfer Control	
11	Transfer At	Start Of Iteration
12	Under Relaxation Factor	0.25
13	RMS Convergence Target	0.01
14	Ramping	None

This is the force data transfer. It is best not to under-relax the displacement data transfer because the Fluent and Mechanical mesh positions will be slightly different at the end of the first time step. This small difference will carry forward and accumulate in subsequent time steps.

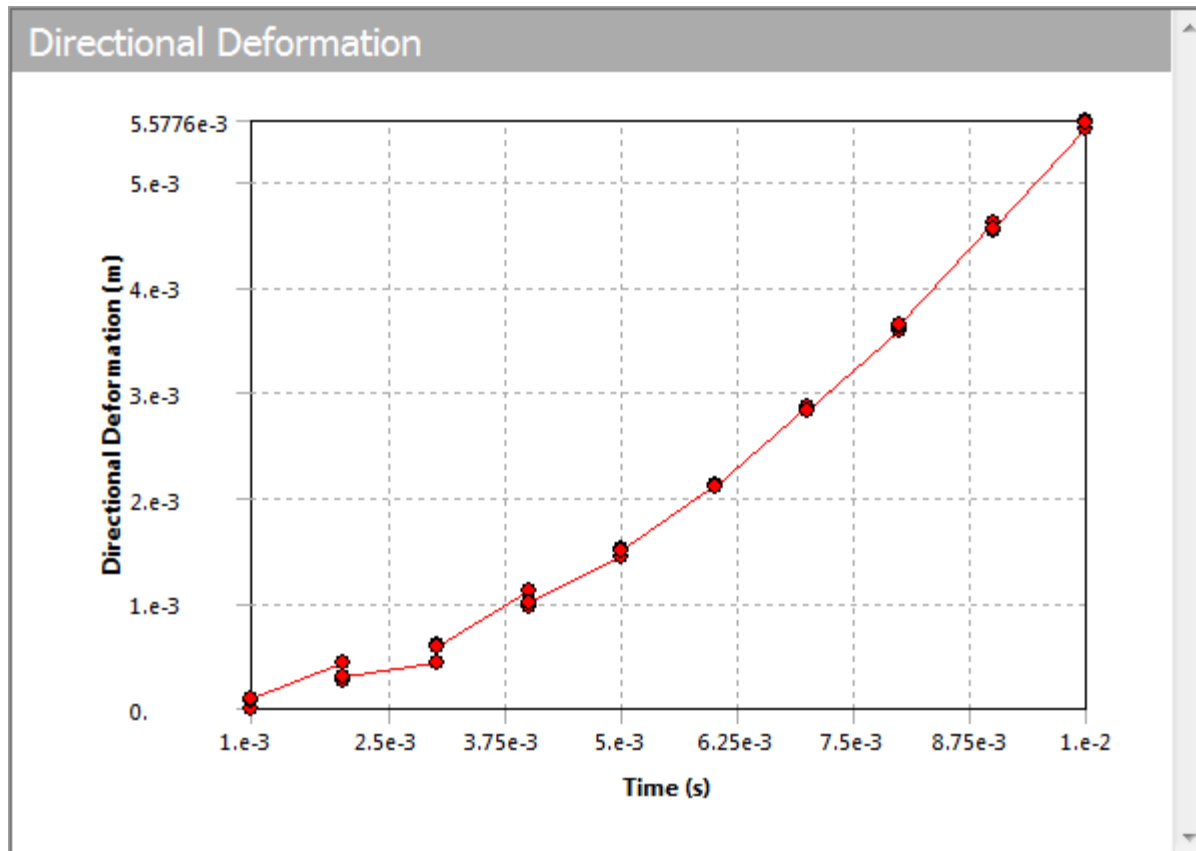
- c. Now you will create a chart to track the force on the interface as the solution progresses.
 - i. In the tree right-click on **Chart Monitors** and select **Create Chart**.
Chart 2 will be added to the tree.
 - ii. Right-click on **Chart 2** and select **Add Variable > Fluid Flow (FLUENT) > Data Transfer 2 > Value > Sum** from the context menu.
 - iii. Add a second variable to the chart using **Add Variable > Transient Structural > Data Transfer 2 > Value > Sum** from the context menu.
 - iv. Select **Chart 2** and in the **Properties of Convergence Chart : Chart 2** panel select **Coupling Step** from the **Quantity** drop-down list in the **Axis X** group box.
- d. Click **Update**.

The nodal sum of the values for **Data Transfer 2** (Force) will be shown. The Fluent side represents the forces received from Fluent, while the structural side shows the relaxed forces sent to Mechanical APDL.



As you can see the forces are converging within each coupling step. The solution will finish after 10 steps. Notice in the last coupling step the force from Fluent was still changing significantly but the force sent to Mechanical APDL changed very little. A tighter force convergence criteria might be appropriate here.

12. Return to the Mechanical window and select **Directional Deformation** from the tree.



As you can see in the graph the vertex displacement history is shown. Values from every Coupling Iteration are shown, not just the converged values. This is useful for debugging.

13. Close Mechanical, then save and exit the project.

3. Summary

- This tutorial has demonstrated how to deal with some commonly occurring problems when solving FSI cases.
- When problems occur, examining the Mechanical APDL, Fluent and **System Coupling** output files can give clues to the problem.
- The mapping summary should always be checked. Unmapped nodes can be identified by loading the results files or `CSV` files dumped from **System Coupling** into CFD-Post.
- When a case fails due to mesh collapse in Fluent or an element formulation error in the Mechanical APDL solver the forces and displacements should be examined. **Results Trackers** in Mechanical and force charts in **System Coupling** (or surface monitors in Fluent) should be used for this.