
2-way FSI for a Hyperelastic Flap Including Dynamic Remeshing

This tutorial shows how to prepare a 2-way coupled Systems Coupling solution in ANSYS Workbench. The purpose of the tutorial is to understand all the key steps necessary for solving a full 2-way FSI simulation within Workbench that result in a large deformation which requires dynamic re-meshing.

In this tutorial you will learn how to do:

- Setup of the Transient Structural case for the hyperelastic material.
- Setup of the Fluent dynamic-mesh case, including smoothing and re-meshing.
- Setup and solution of the coupled flow case.

1. Problem Description

This example considers the large deformation of a hyperelastic flap as a result of the hydrodynamic forces from a surrounding fluid flow. The flow is transient, and the coupling involves 2-way FSI between Fluent and Mechanical.

The fluid region is a channel 0.15 m high and 0.25 m long. Air enters at the left hand side at 20 m/s. This causes the flap, made of a hyperelastic rubber, to deform. The simulation is run over 75 time steps. The solution is a full 2-way FSI:

- ANSYS Fluent transfers the pressure force on the flap to ANSYS Mechanical.
- ANSYS Mechanical computes the deformation, and transmits this to Fluent.
- Fluent modifies the mesh (using smoothing and re-meshing) to resolve the motion.

System Coupling must be a 3D analysis. In this case you will generate a mesh just 1 element/cell thick. The Fluent cells will be triangular prisms, but you can re-mesh these by using the 2.5D remeshing scheme.

2. Setup And Solution

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

- 2.1. Preparation
- 2.2. Starting Workbench
- 2.3. Fluent Setup
- 2.4. Mechanical Setup
- 2.5. System Coupling
- 2.6. Running the Simulation
- 2.7. Postprocessing
- 2.8. CFD-Post

2.1. Preparation

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

2.2. Starting Workbench

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

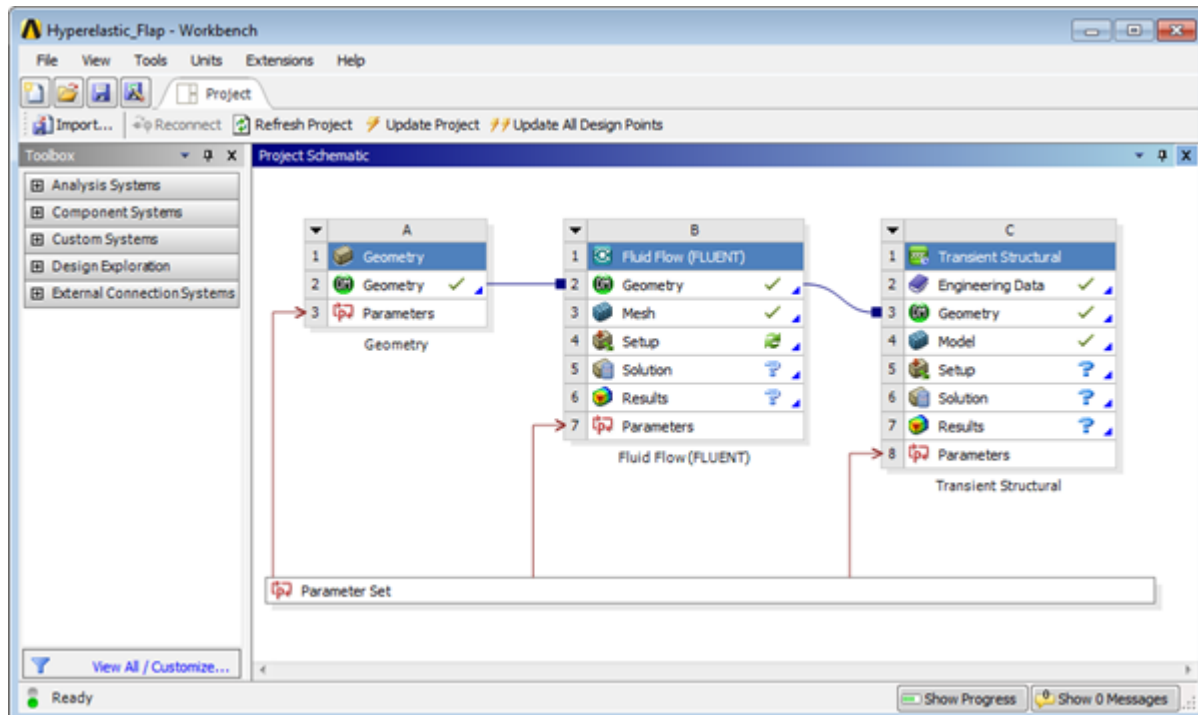
Note

The geometry and mesh have already been created (both fluid and solid regions).

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

Note

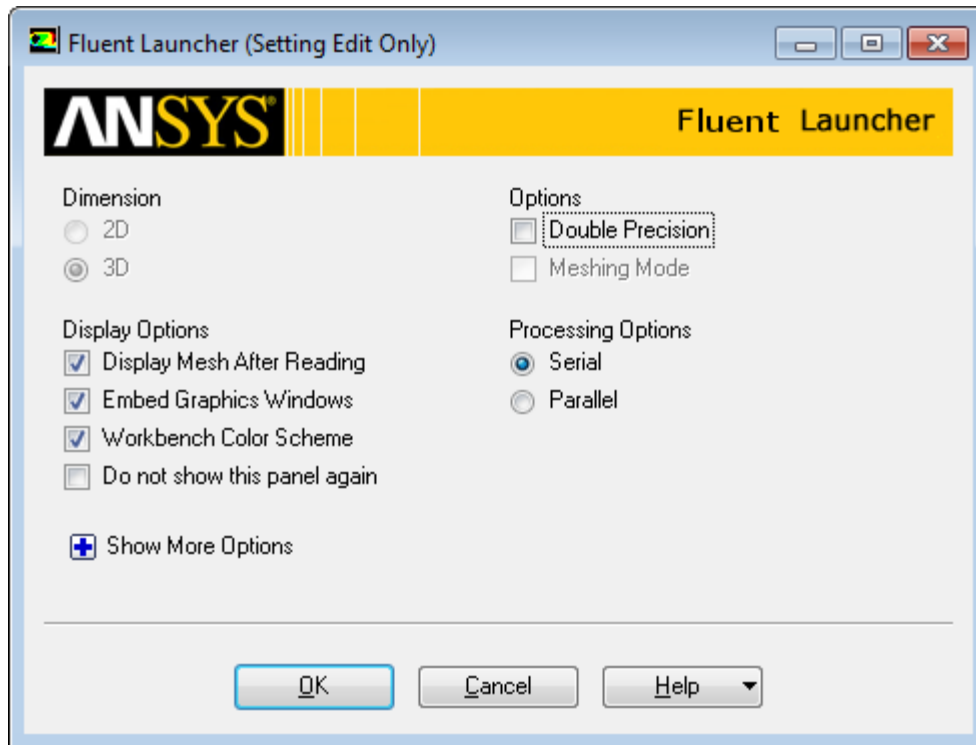
Parameters were used during the initial geometry creation (for controlling the top curve on the flap). This tutorial will not be modifying these parameters, though you may wish to try this yourself later.



2.3. Fluent Setup

In the **Project Schematic** window double-click on the **Setup** cell, **B4** in the **Fluid Flow (FLUENT)** analysis system.

1. Retain the default settings and click **OK** in the **Fluent Launcher** window.

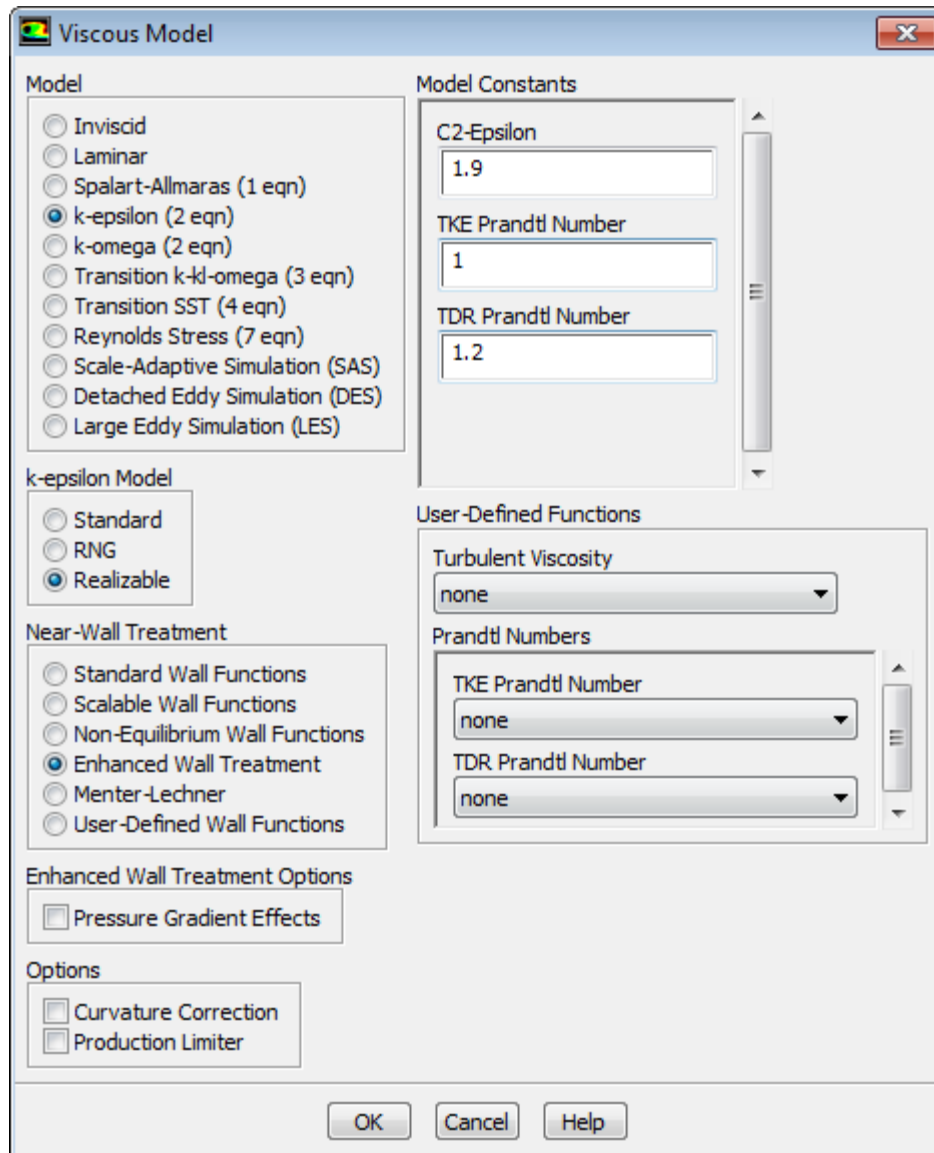


2. In the Fluent window, in the **General** task page, click **Check**.

This will perform various checks on the mesh and will report the result to the console. Ensure that the reported minimum volume is a positive number.

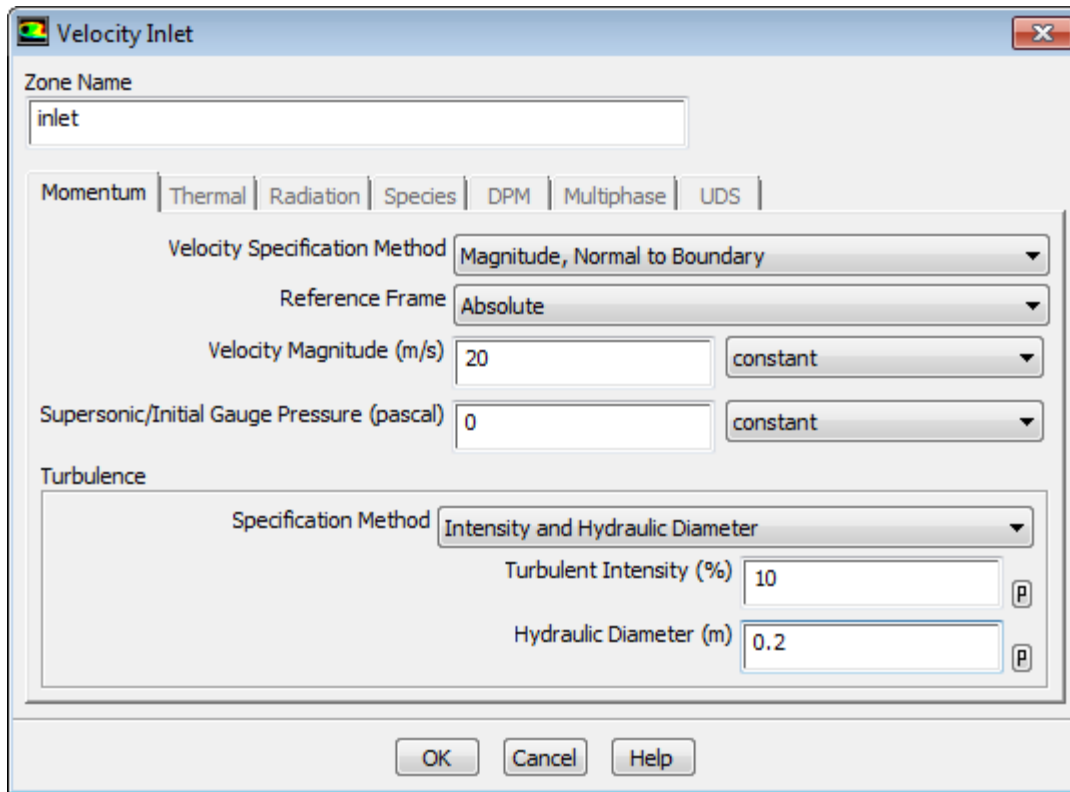
- Enable **Transient** under **Time**.

3. Click on **Models** in the tree.
 - a. In the **Models** task page select **Viscous — Laminar** and click **Edit...**
 - b. Select **k-epsilon (2 eqn)** from the **Model** list in the **Viscous Model** dialog box.
 - c. In the **Viscous Model** dialog box select **Realizable** in the **k-epsilon Model** group box.
 - d. Select **Enhanced Wall Treatment** from the **Near-Wall Treatment** group box.

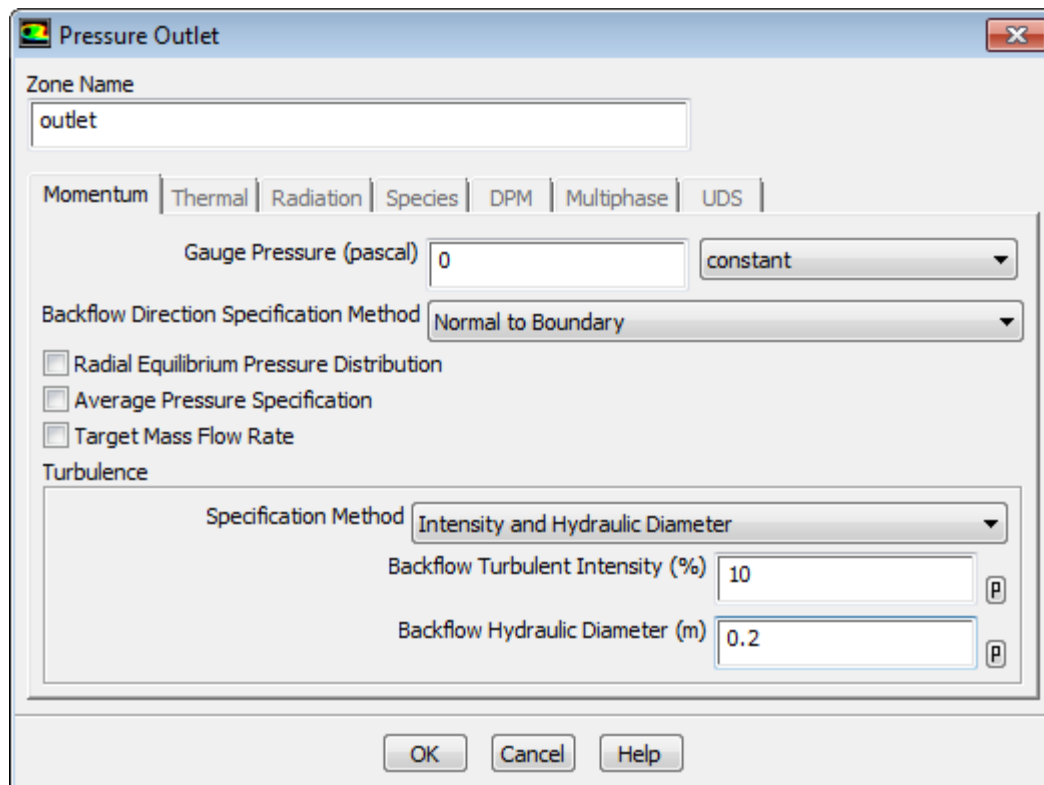


e. Click **OK** to close the **Viscous Model** dialog box.

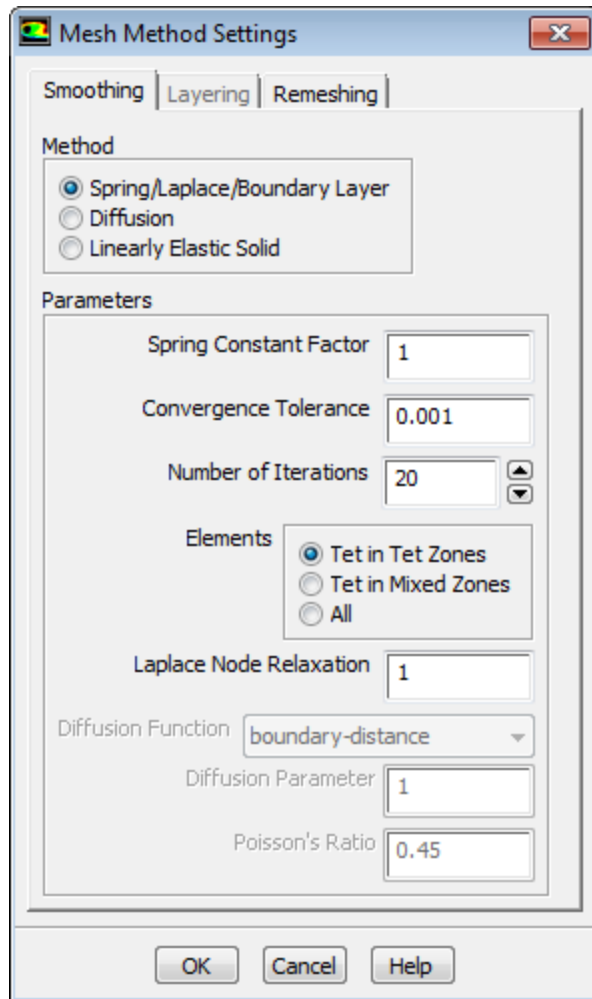
4. Click on **Materials** in the tree. In the **Materials** task page you can see that **air** is selected as the default material.
5. Click on **Boundary Conditions** in the tree.
 - a. Select **inlet** from the list of **Zone** and click **Edit....**



- i. Enter 20 for **Velocity Magnitude (m/s)**.
 - ii. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
 - iii. Enter 10 for **Turbulent Intensity (%)**.
 - iv. Enter 0.2 for **Hydraulic Diameter (m)**.
 - v. Click **OK** to close the **Velocity Inlet** dialog box.
- b. Select **outlet** from the list of **Zone** and click **Edit....**



- i. Retain **0** for **Gauge Pressure (pascal)**.
 - ii. Select **Intensity and Hydraulic Diameter** from the **Specification Method** drop-down list.
 - iii. Enter 10 for **Backflow Turbulent Intensity (%)**.
 - iv. Enter 0.2 for **Backflow Hydraulic Diameter (m)**.
 - v. Click **OK** to close the **Pressure Outlet** dialog box.
- c. Ensure that zones **symmetry_1** and **symmetry_2** are set to **symmetry** under **Type**.
 - d. Similarly ensure that zones **bottom_wall**, **top_wall**, and **wall_cfd_coupled** are set to **wall** under **Type**.
6. Click on **Dynamic Mesh** in the tree.
 - a. Enable **Dynamic Mesh** in the task page of **Dynamic Mesh**.
 - b. Retain the selection of **Smoothing** in the **Mesh Methods** group box.
 - c. Enable **Remeshing**.
 - d. Click **Settings...** in the **Mesh Methods** group box.
 - i. In the **Mesh Method Settings** dialog box, click on the **Smoothing** tab.

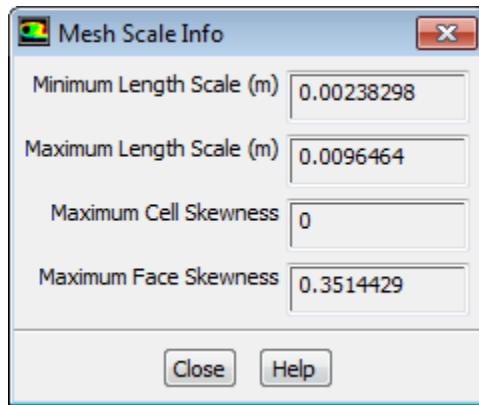


- ii. Retain the selection of **Spring/Laplace/Boundary Layer** from the **Method** group box.

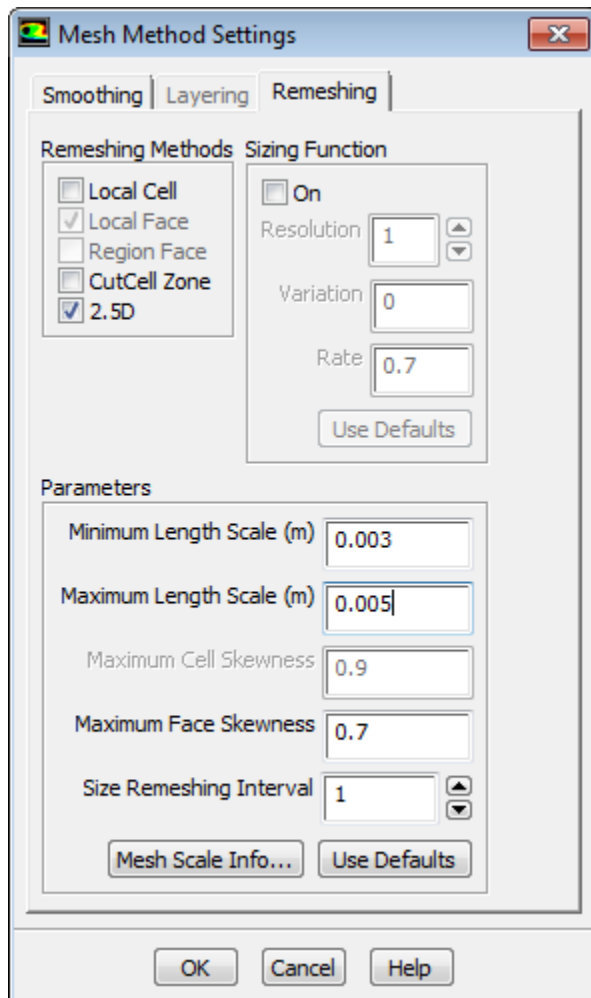
Note

Diffusion smoothing is usually the first choice for FSI cases, but when 2.5D remeshing is used, Fluent will always use Laplace smoothing, regardless of the choice made on the **Smoothing** tab.

- iii. Click the **Remeshing** tab.
- iv. Enable **2.5D** in the **Remeshing Methods** group box.
- v. Click on **Mesh Scale Info** and observe the current maximum and minimum cell sizes.



- vi. Enter 0.003 for **Minimum Length Scale (m)**.



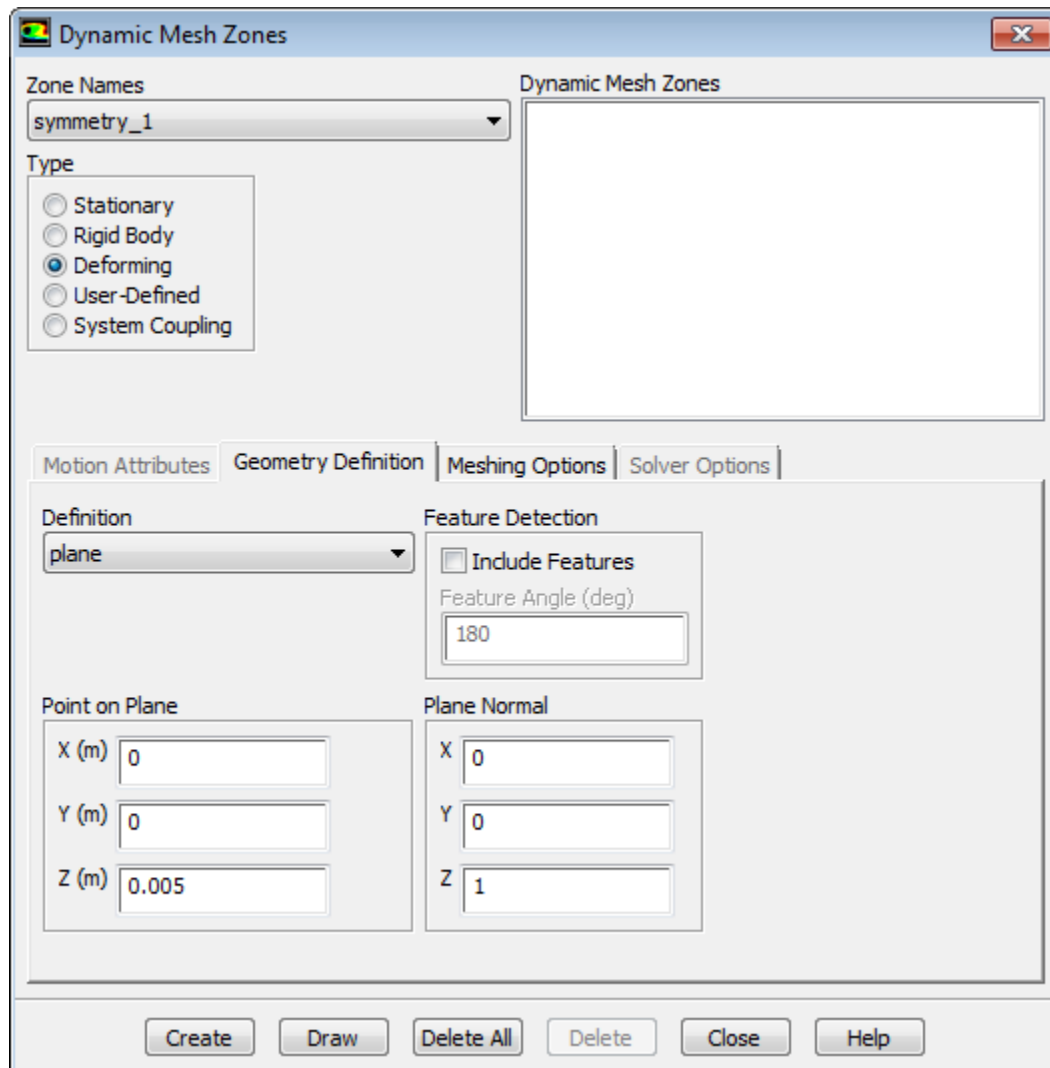
- vii. Enter 0.005 for **Maximum Length Scale (m)**.
- viii. Retain **0.7** for **Maximum Face Skewness**.
- ix. Enter 1 for **Size Remeshing Interval**.
- x. Close the **Mesh Scale Info** dialog box.

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

Note

Fluent will normally only remesh either (in 2D) triangular cells, or (in 3D) tetrahedral cells. However in cases with a swept mesh like this, you can apply 2.5D remeshing so that re-meshing is applied to the tri-prism grid cells.

- e. Click **Create/Edit...** under **Dynamic Mesh Zones** | the **Dynamic Mesh** task page.
- i. In the **Dynamic Mesh Zones** dialog box, select **symmetry_1** from the **Zone Names** drop-down list.



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

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

Note

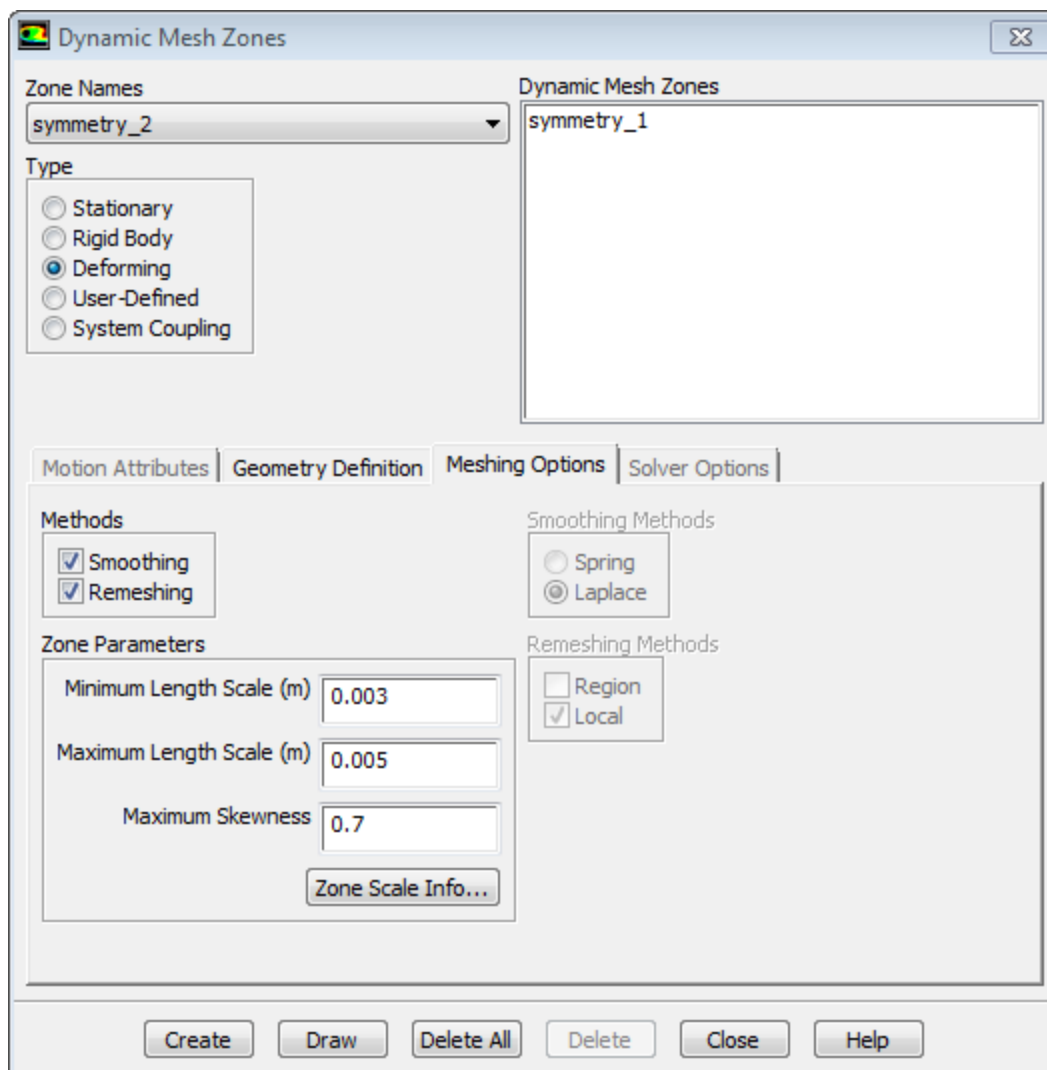
Fluent does not have the underlying geometry for the model, it just sees the starting mesh. If you are to slide/move grid cells on this symmetry plane then Fluent needs to understand the geometric shape it is working with, in this case a plane. The symmetry plane mesh can now move, but it is constrained to the defined plane.

- C. Set **Point on Plane** to 0, 0, 0.005.
- D. Set **Plane Normal** to 0, 0, 1.
- E. In the **Meshing Options** tab disable **Smoothing** and **Remeshing** under **Methods**.

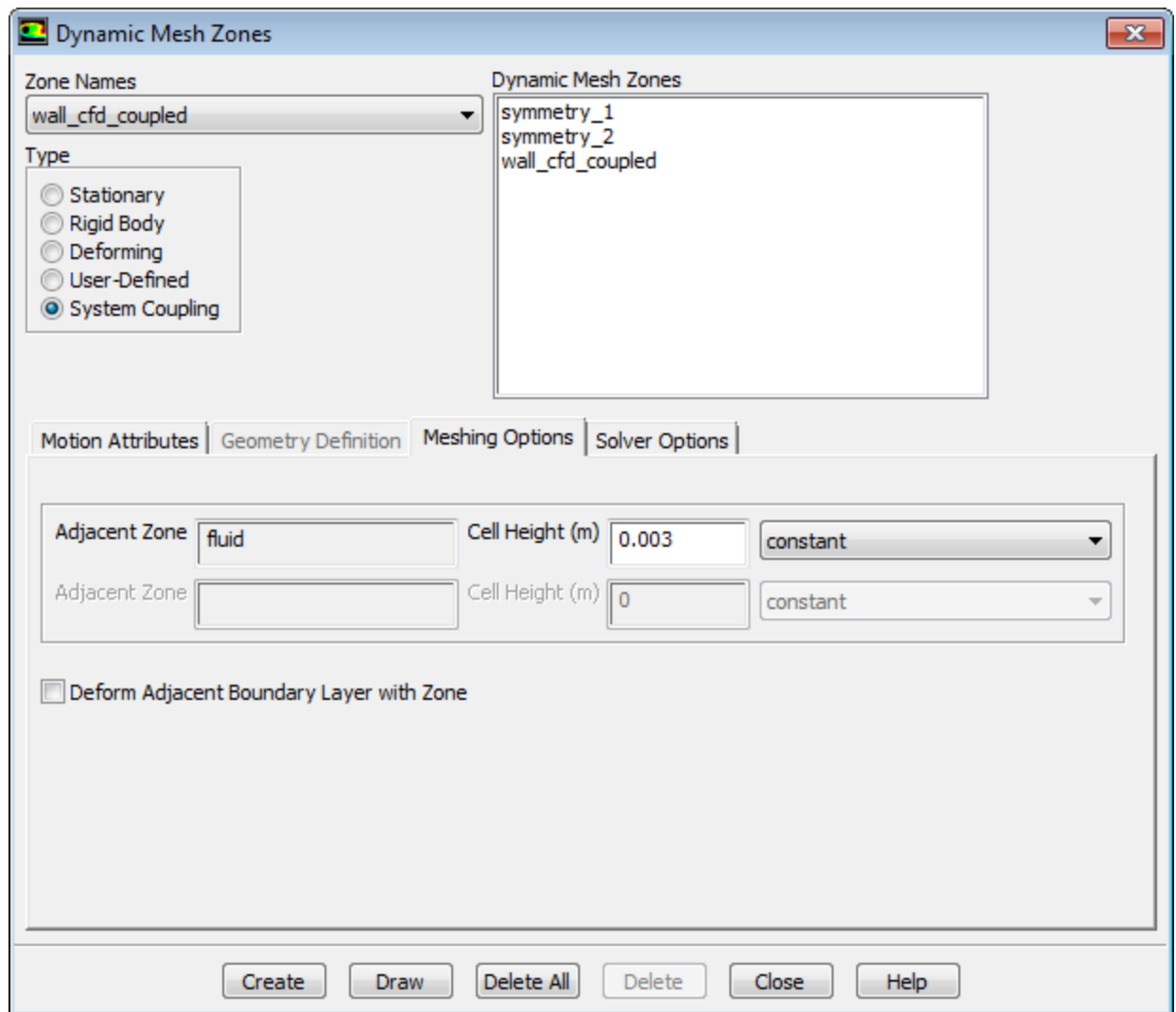
Note

For 2.5D re-meshing, one face will lead (in this case **symmetry_2**) and will have smoothing and re-meshing active. The face, **symmetry_1** replicates the mesh of the lead face, and therefore you do not want smoothing and re-meshing active here.

- F. Click **Create**.
- ii. Select **symmetry_2** 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. Set **Point on Plane** to 0 , 0 , 0.
 - D. Set **Plane Normal** to 0 , 0 , 1.
 - E. In the **Meshing Options** tab enable **Smoothing** and **Remeshing** under **Methods**.
 - F. Enter 0 . 003 for **Minimum Length Scale (m)**.
 - G. Enter 0 . 005 for **Maximum Length Scale (m)**.
 - H. Enter 0 . 7 for **Maximum Face Skewness**.
 - I. Click **Create**.
- iii. Select **wall_cfd_coupled** from the **Zone Names** drop-down list.



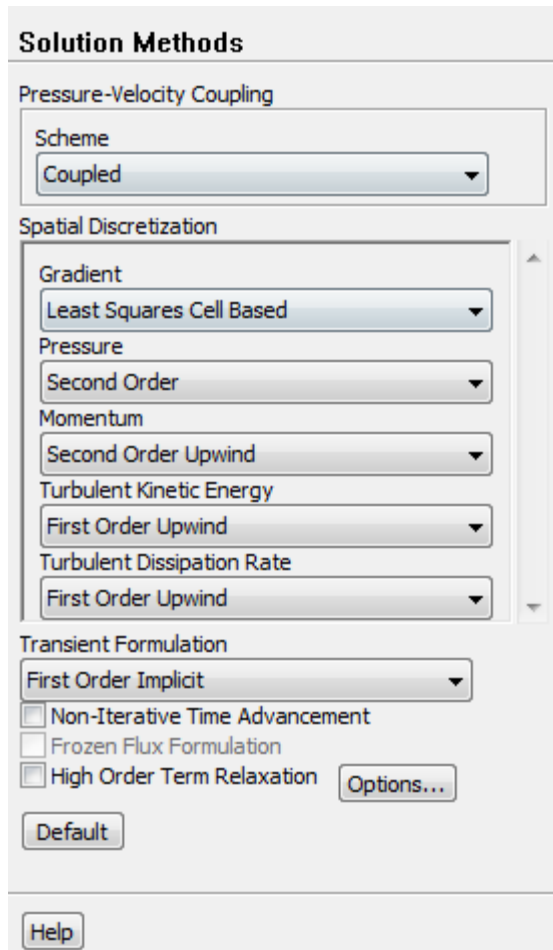
- A. Select **System Coupling** from the **Type** group box.
- B. In the **Meshing Options** tab enter 0 . 003 for **Cell Height**.
- C. Click **Create**.

Note

This is the key setting in Fluent to allow 2-way FSI. This boundary comprises the 3 surfaces (2 sides and top) of the flexible flap. The deformed shape of this part is being computed by ANSYS Mechanical, and being transferred to Fluent. The deformation vector is defined for each individual node that makes up the coupled surface.

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

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



- In the **Solution Methods** task page select **Coupled** from the **Scheme** drop-down list.
8. Click **Solution Initialization** in the tree and select **Hybrid Initialization** in the **Initialization Methods** group box. Click **Initialize**.
 9. Click **Calculation Activities** in the tree and enter 1 for **Autosave Every (Time Steps)**.

Note

Use this setting (or **Automatic Export**) to create post-processing data for Fluent. To create backup/restart points use the setting in **System Coupling**.

10. Click **Run Calculation**.
 - a. Enter 1 for **Number of Time Steps**.

Note

This value is not used, but must be greater than zero. You do not need to set the **Time Step Size**, this will be controlled externally from the **System Coupling** process.

- b. Retain **20** for **Max Iterations/Time Step**.

Note

For **System Coupling** cases this is actually the number of Fluent iterations per Coupling Iteration.

- 11. Save the project.

File > Save Project

- 12. Close Fluent.

2.4. Mechanical Setup

In the Workbench project window, double-click on the **Model** cell, **C4** in the **Transient Structural** system.

Note

This will open the ANSYS Mechanical window.

- 1. Select **Metric (m, kg, N, s, V, A)** from the **Units** menu.
- 2. In the tree expand **Geometry** and select **fea**.

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

In the **Details of fea** panel note the selection of **Hyperelastic** for **Assignment** for **Material**.

Note

This material was created in the **Engineering Data** cell (**C2**) in the **Project Schematic**. Next you need to apply the symmetry condition to the front and back (high and low z faces).

3. Right-click on **Model (C4)** at the top of the tree and select **Insert > Symmetry** from the context menu. A new object **Symmetry** is added in the tree.
4. Right-click on **Symmetry** in the tree and select **Insert > Symmetry Region** from the context menu.

Details of "Symmetry Region"	
Scope	
Scoping Method	Named Selection
Named Selection	symmetry_a
Definition	
Scope Mode	Manual
Type	Symmetric
Coordinate System	Global Coordinate System
Symmetry Normal	Z Axis
Suppressed	No

- a. In the **Details of Symmetry Region** panel select **Named Selection** from the **Scoping Method** drop-down list.
 - b. From the **Named Selection** drop-down list select **symmetry_a**.
 - c. Select **Z Axis** from the **Symmetry Normal** drop-down list.
 - d. Right-click on the **Symmetry Region** in the tree and select **Duplicate** from the context menu.
 - e. For **Symmetry Region 2** select **symmetry_b** from the **Named Selection** drop-down list in the details view.
5. Right-click on **Transient (C5)** in the tree and select **Insert > Fixed Support** from the context menu.

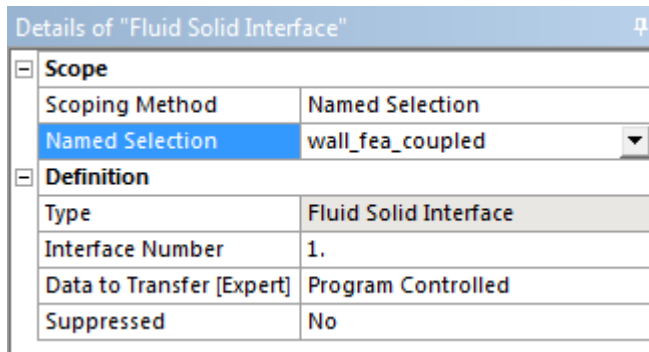
Details of "Fixed Support"	
Scope	
Scoping Method	Named Selection
Named Selection	clamped
Definition	
Type	Fixed Support
Suppressed	No

- a. In the **Details of Fixed Support** panel select **Named Selection** from the **Scoping Method** drop-down list.
- b. From the **Named Selection** drop-down list select **clamped**.

Note

This is the small square face at the bottom of the flap that is rigidly fastened to the bottom of the channel.

- c. Right-click on **Transient (C5)** in the tree and select **Insert > Fluid Solid Interface** from the context menu.

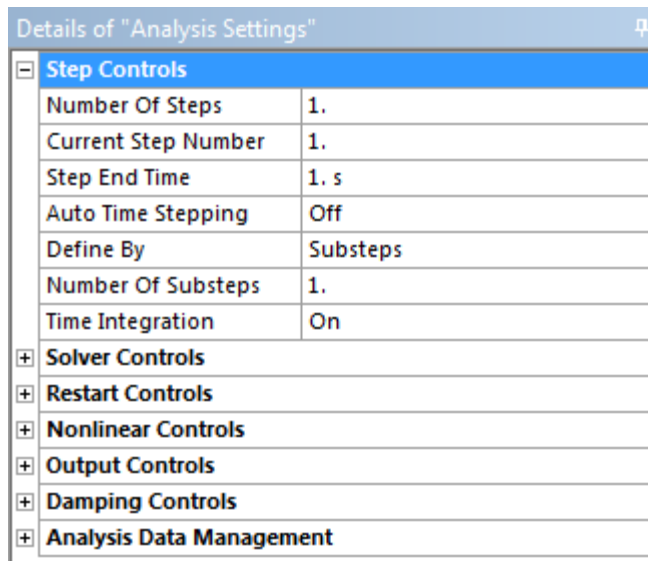


- i. In the **Details of Fluid Solid Interface** panel select **Named Selection** from the **Scoping Method** drop-down list.
- ii. From the **Named Selection** drop-down list select **wall_fea_coupled**.

Note

This is the key step for the Mechanical model in order to perform a 2-way FSI simulation. This surface is the wetted outer surface in contact with the fluid. **System Coupling** will map the forces from the CFD computation on to this surface, and transfer back the resulting deformation to Fluent.

6. Click **Analysis Settings** in the tree.



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

- c. Enter 1 for **Number Of Substeps**.

Note

A single substep is usually recommended when using Mechanical with **System Coupling**.

7. Save the project.

File > Save Project...

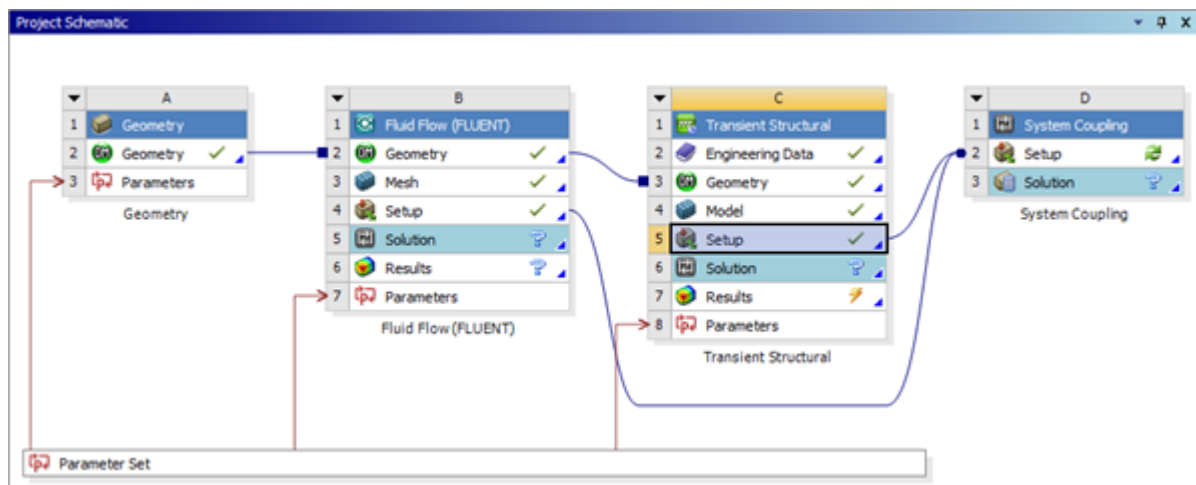
Note

The Mechanical setup is now complete. Since you have not set any output controls, the results will be saved at every time step.

8. Close the Mechanical window.

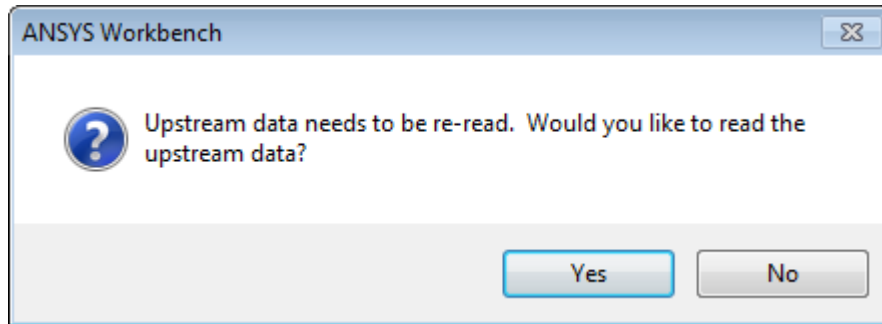
2.5. System Coupling

1. In the Workbench window drag a **System Coupling** from under **Component System** onto the **Project Schematic**.



- Draw a connector from the **Setup** cell (**B4**) of **Fluid Flow (FLUENT)** system to the **Setup** cell (**D2**) of the **System Coupling** system.
- Similarly, draw a connector from the **Setup** cell (**C5**) of Transient Structural system to the **Setup** cell (**D2**) of the **System Coupling** system.
- Note the **Setup** cell (**B4**) of **Fluid Flow (FLUENT)** requests an update. Right-click on the cell and select **Update** from the context menu.
- Note the **Setup** cell (**C5**) of **Transient Structural** requests an update. Right-click on the cell and select **Update** from the context menu.

2. Double-click on the **Setup** cell **D2** of **System Coupling**, and click **Yes** to re-read the upstream data.



3. Under **Outline of Schematic** in the **System Coupling** tab click on **Analysis Settings**.

Note

If the **Outline of Schematic** is not visible select **View** > **Outline** from the menu bar.

 A screenshot of the "Properties of Analysis Settings" panel in ANSYS Workbench. It is a table with two columns: "A" (Property) and "B" (Value).

	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]	0.0075
8	[-] Step Controls	
9	Step Size [s]	5E-05
10	Minimum Iterations	1
11	Maximum Iterations	5

- a. In the **Properties of Analysis Settings** panel enter 0.0075 for **End Time [s]**.

Note

If the **Properties of Analysis Settings** is not visible select **View** > **Properties** from the menu bar.

- b. Enter 5e-05 for **Step Size [s]**.

4. Under **Participants** > **Fluid Flow (FLUENT)** > **Regions** select **wall_cfd_coupled**.

Also select **Fluid Solid Interface** under **Participants** > **Transient Structural** > **Regions** by holding down the **Ctrl** key.

- Right-click and select **Create Data Transfer**.

Note

This will create a data transfer between these two regions. Two new **Data Transfer** objects will be added.

	A	B
1	Property	Value
2	Source	
3	Participant	Fluid Flow (FLUENT)
4	Region	wall_cfd_coupled
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	1
13	RMS Convergence Target	0.01
14	Ramping	None

Data transfer takes the force computed on **wall_cfd_coupled** in Fluent and transmits it to **Fluid Solid Interface** in Transient Structural.

	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_cfd_coupled
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	None

Data Transfer 2 takes the displacement from **Fluid Solid Interface** and transmits it to **wall_cfd_coupled**.

5. Under **Execution Control** click on **Intermediate Restart Data Output**. Check that the **Output Frequency** is set to **None**.

Note

This setting should only be used for backup files.

6. Save the project.

File > Save

2.6. Running the Simulation

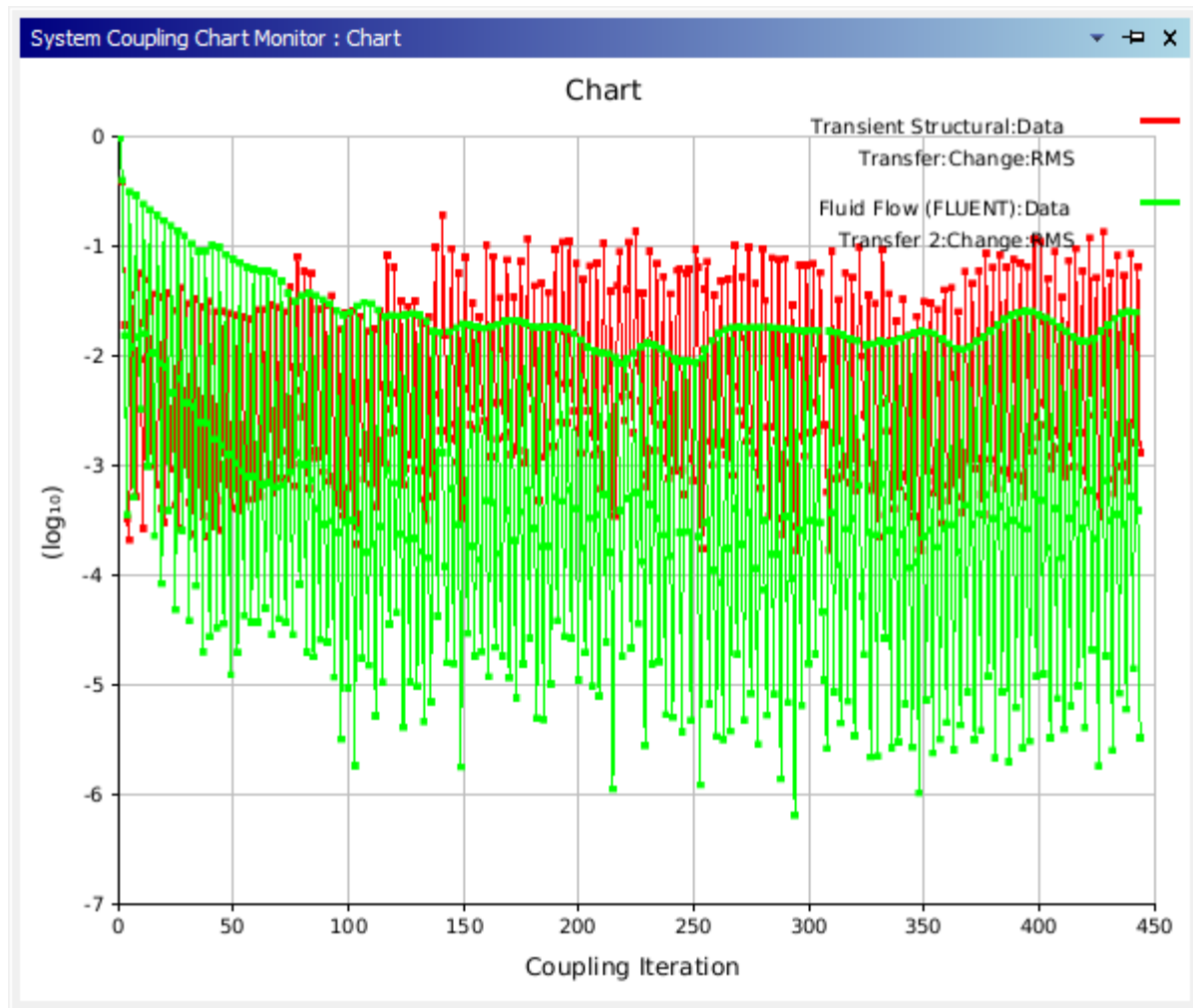
1. Click on **Update** in the toolbar to start the solution.

Note

When setting the time step size, the value (5e-05) will be used by both CFD and FEA codes. You have retained the default **Maximum Iterations** setting of **5** in **System Coupling**. This is the maximum number of iterations **System Coupling** will perform between the participant solvers per time step. Hence within a given time step, you will find that:

- Fluent performs up to 20 flow iterations.
- Fluent passes the loads to Mechanical to compute the displacement.
- Mechanical performs iterations to converge its solution.
- Mechanical passes the new position back to Fluent.

This whole process could then be repeated up to 5 times for each time step (in other words potentially 100 Fluent iterations per time step if convergence is poor).



- Under **Solution** in the **Outline of Schematic** of **System Coupling**, click on each object to view the output transcript from **System Coupling**, **Fluid Flow (FLUENT)**, and **Transient Structural**.
- Just before the main computation starts, check the **MAPPING SUMMARY** in the **System Coupling** log and confirm that 100% of the nodes have been mapped.

Solution Information : System Coupling

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

COUPLING STEP = 1 STIMULATION TIME = 5.00000e-005

4. Click on **System Coupling** under **Solution Information** and check the log file.

Solution Information : System Coupling		
COUPLING ITERATION = 1		
Transient Structural	Converged	
Data Transfer Change:RMS	1.70739e-003	1.64852e-003
Fluid Flow (FLUENT)	Converged	
Data Transfer 2 Change:RMS	2.56968e-002	2.56450e-002
COUPLING ITERATION = 2		
Transient Structural	Converged	
Data Transfer Change:RMS	6.94462e-002	6.66236e-002
Fluid Flow (FLUENT)	Converged	
Data Transfer 2 Change:RMS	3.94817e-004	3.95358e-004
COUPLING ITERATION = 3		
Transient Structural	Converged	
Data Transfer Change:RMS	1.37983e-003	1.33859e-003
Fluid Flow (FLUENT)	Converged	
Data Transfer 2 Change:RMS	3.36784e-006	3.37087e-006
Shut Down		
System Coupling Service shut down...		
System coupling run completed successfully.		

Review the output. Note that typically the coupling system reaches convergence within 3 coupling iterations (so a maximum of 5 coupling iterations was appropriate).

5. Click on **Fluid Flow (FLUENT)** under **Solution Information** and check the log file.

```

Solution Information: Fluid Flow (FLUENT)
COUPLING STEP = 150  COUPLING ITERATION = 3
1844 solution is converged
1844 y-velocity 4.7365e-06 2.3925e-09 1.1472e-05 1.5902e-05 0:00:00 20
1844 z-velocity 4.7365e-06 2.3925e-09 1.1472e-05 1.5902e-05 0:00:00 20
Deleting mesh as requested...done.
1845 solution is converged
1845 y-velocity 2.9931e-06 1.8325e-09 5.2246e-06 6.9201e-06 0:00:00 19
1845 z-velocity 2.9931e-06 1.8325e-09 5.2246e-06 6.9201e-06 0:00:00 19
Flow time = 0.007499991s, time step = 150
Writing "j gzfp -2cf > E:\tutorials\FluentFSI02_Hyperelastic_FlapHyperelastic_Flap_filesdp0FFFFluentFFF.1-2-00150.cas.gz"...
Writing temporary file C:\Users\mkolhatk\AppData\Local\Temp\flntgz-9960302 ...
5150 wedge cells, zone 4, binary.
7634 quadrilateral interior faces, zone 1, binary.
37 quadrilateral wall faces, zone 5, binary.
21 quadrilateral velocity-inlet faces, zone 6, binary.
21 quadrilateral pressure-outlet faces, zone 7, binary.
5150 triangular symmetry faces, zone 8, binary.
5150 triangular symmetry faces, zone 9, binary.
35 quadrilateral wall faces, zone 10, binary.
68 quadrilateral wall faces, zone 11, binary.
5334 nodes, binary.
5334 node flags, binary.
Done.
Writing "j gzfp -2cf > E:\tutorials\FluentFSI02_Hyperelastic_FlapHyperelastic_Flap_filesdp0FFFFluentFFF.1-2-00150.dat.gz"...
Writing temporary file C:\Users\mkolhatk\AppData\Local\Temp\flntgz-9960303 ...
Done.
CALCULATION COMPLETE
Deleting Client...done
Shutting down Server...done

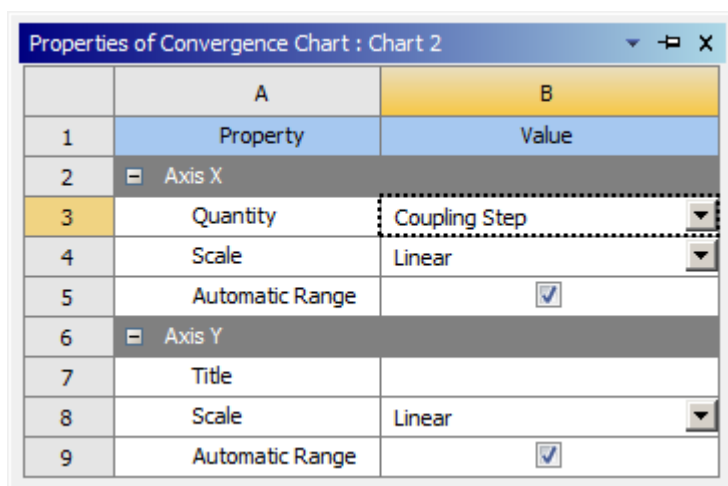
```

You can notice that the flow field is reaching convergence by the end of each time step.

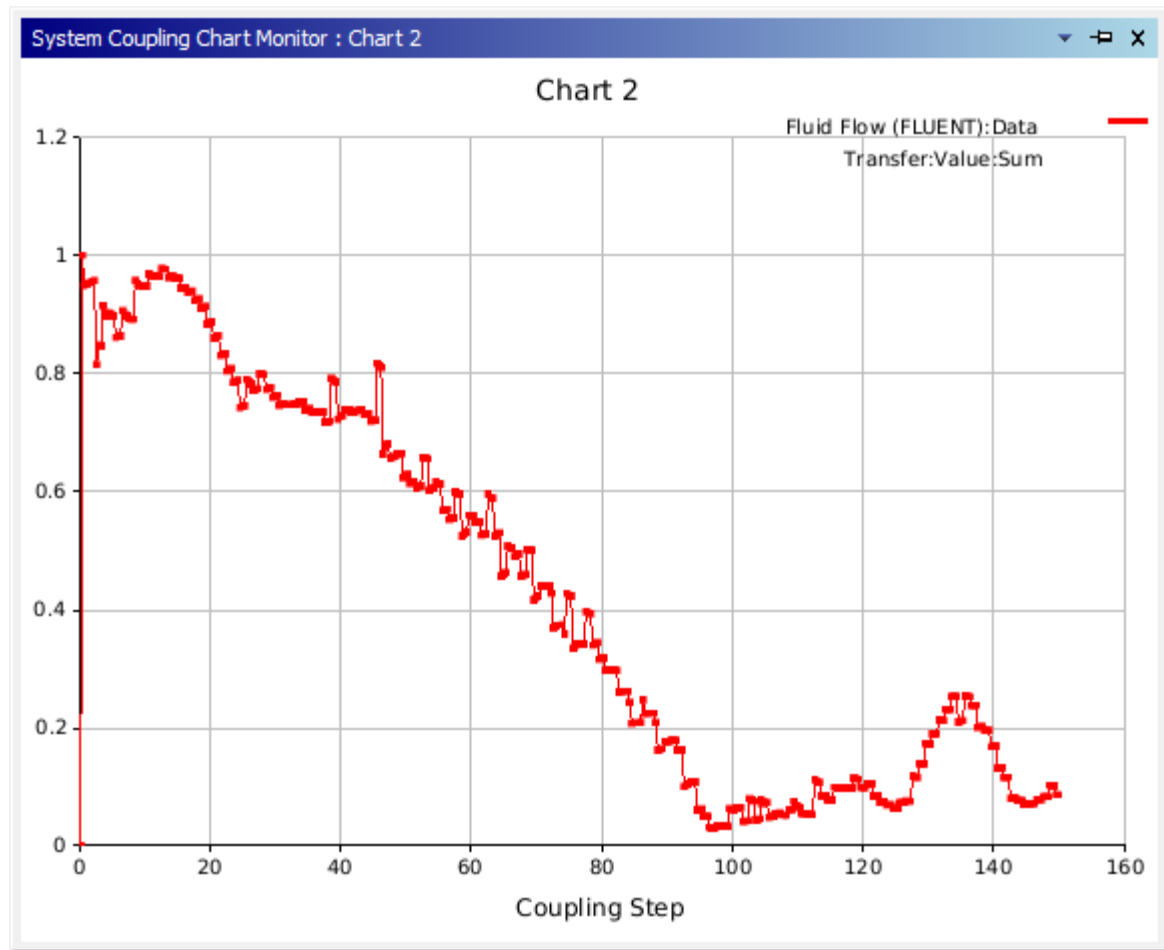
6. Click on **Transient Structural** under **Solution Information** and check the log file.

You can see that in most cases force and displacement convergence was achieved after 2 or 3 equilibrium iterations, which is good.

7. Add a chart to monitor the force on the FSI interface.
 - a. In the **Outline of Schematic** of **System Coupling** right-click on **Chart Monitors** and select **Create Chart** from the context menu.
 - b. Right-click on the newly added **Chart 2** and select **Add Variable > Fluid Flow (FLUENT) > Data Transfer > Value > Sum**.
 - c. In the **Properties of Convergence Chart** panel, select **Coupling Step** from the **Quantity** drop-down list under **Axis X**.



The chart is displayed which shows the sum of all nodal value for data transfer on the Fluent side. Data transfer corresponds to force in this case.



8. Save the project.

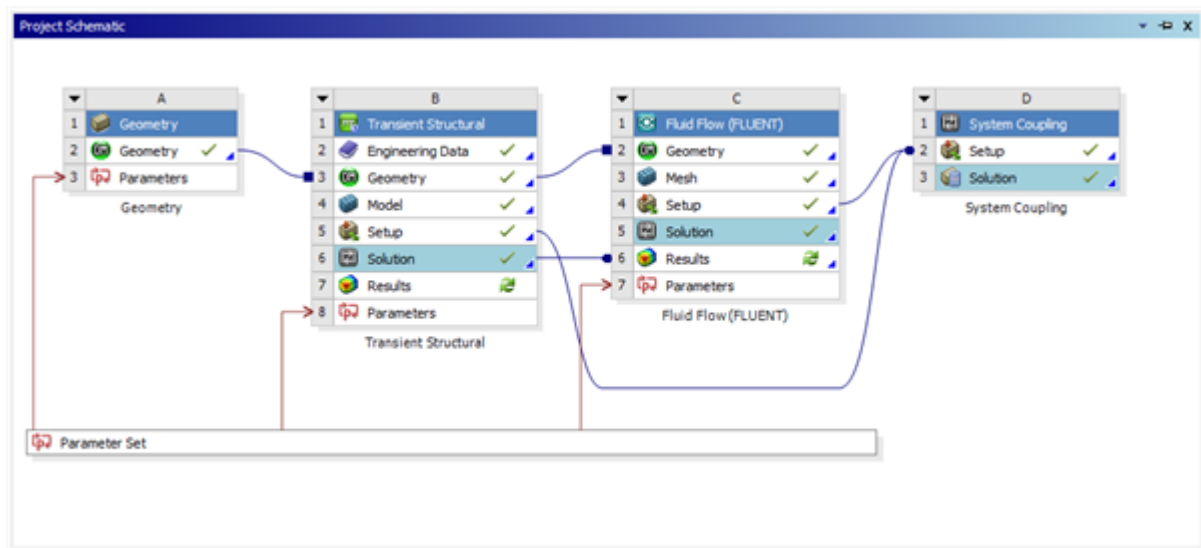
File > Save

2.7. Postprocessing

1. Return to the Workbench window by clicking on the **Project** tab.
2. Draw a connector from the **Solution** cell (C6) of **Transient Structural** system to the **Results** cell (B6) of the **Fluid Flow (FLUENT)** system.

Note

If this connection cannot be made drag cell **C1** to the left and drop it between systems **A** and **B** first.



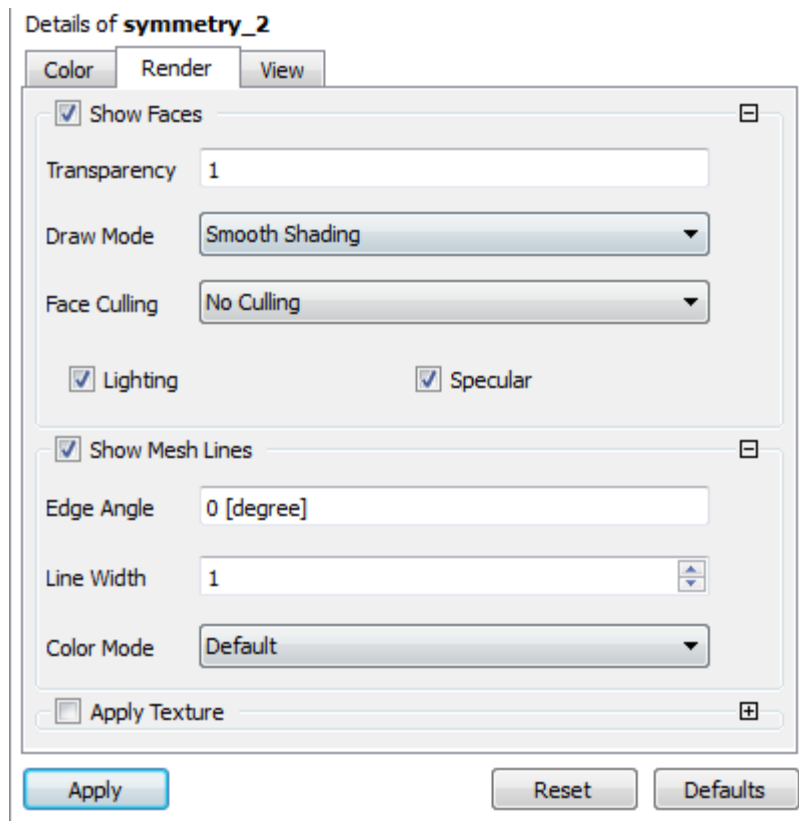
3. Double-click on the **Results** cell (**C6**) of **Fluid Flow (FLUENT)** system to launch CFD-Post.

Note

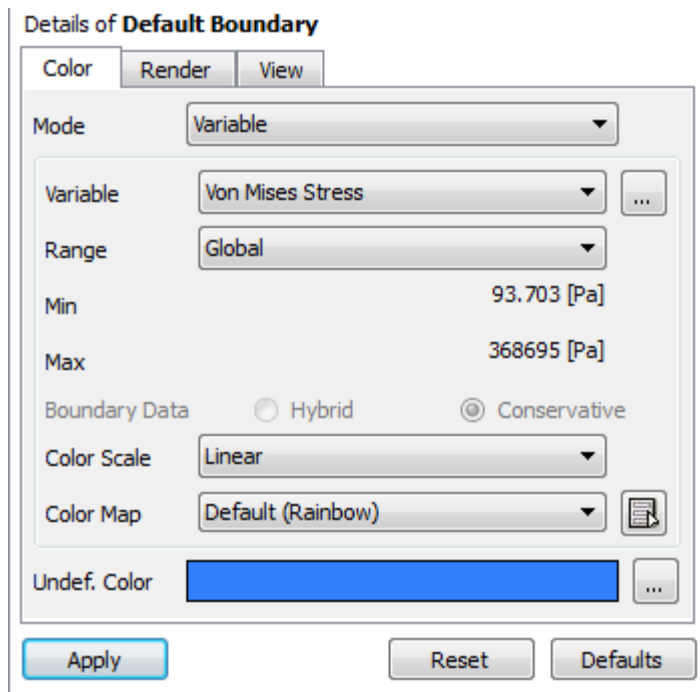
CFD-Post is able to read in the results data from both solvers to allow post-processing of both CFD and FEA data simultaneously.

2.8. CFD-Post

1. In the **CFD-Post** window, in the **Outline** tab double-click on **symmetry_2** under **fluid**.
 - a. In **Details of symmetry_2**, in the **Color** tab retain the selection of **Constant** from the **Mode** drop-down list.



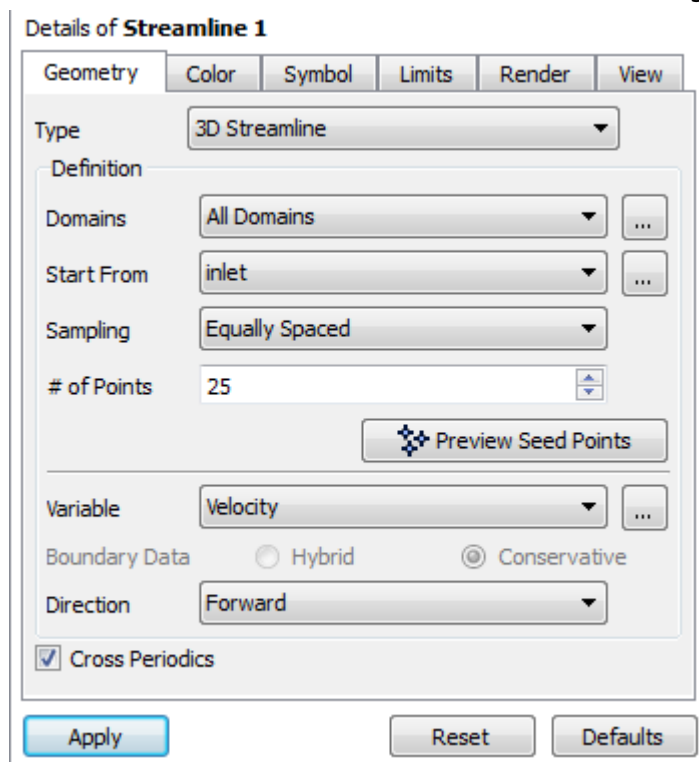
- b. In the **Render** tab enter 1 for **Transparency**.
 - c. To display the mesh enable **Show Mesh Lines**.
 - d. Click **Apply**.
2. Right-click in the graphics window and select **Predefined Camera** > **View from +Z** from the context menu.
3. Double-click on **Default Boundary** under **Default Domain** in the tree under **SYS**.



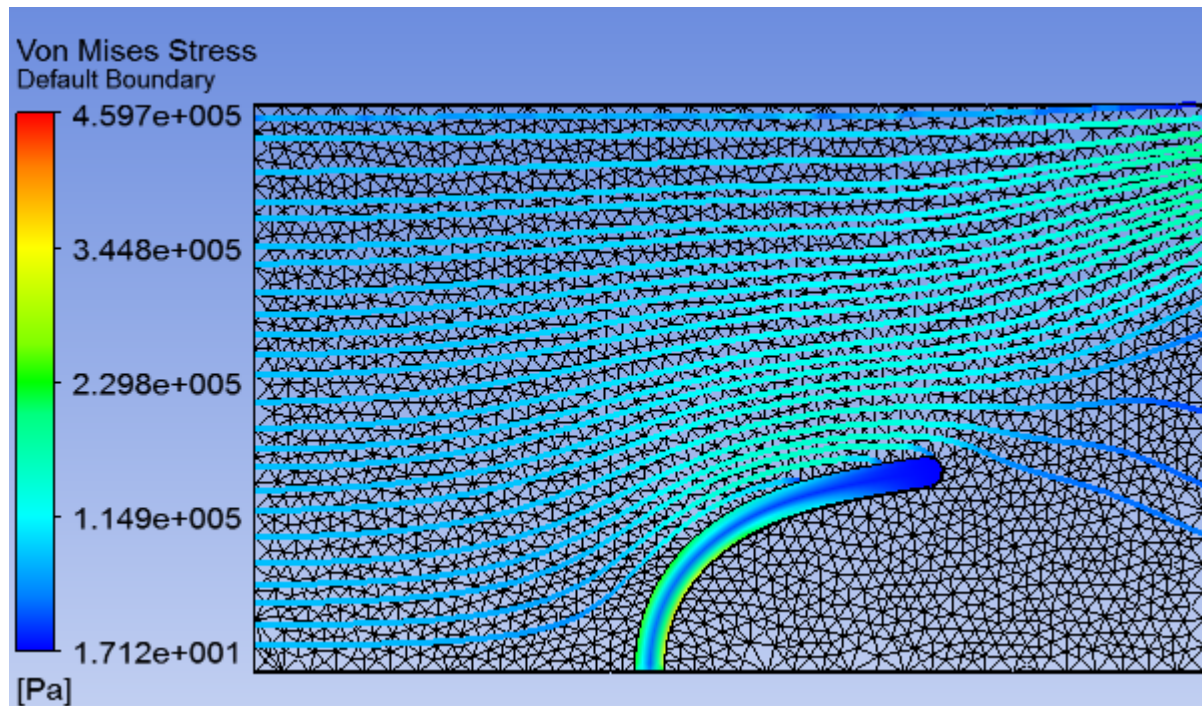
- a. Under **Details of Default Boundary** in the **Color** tab select **Variable** from the **Mode** drop-down list.
- b. Select **Von Mises Stress** from the **Variable** drop-down list.
- c. Click **Apply**.


4. From the menu select **Insert > Streamline** or click on  icon.

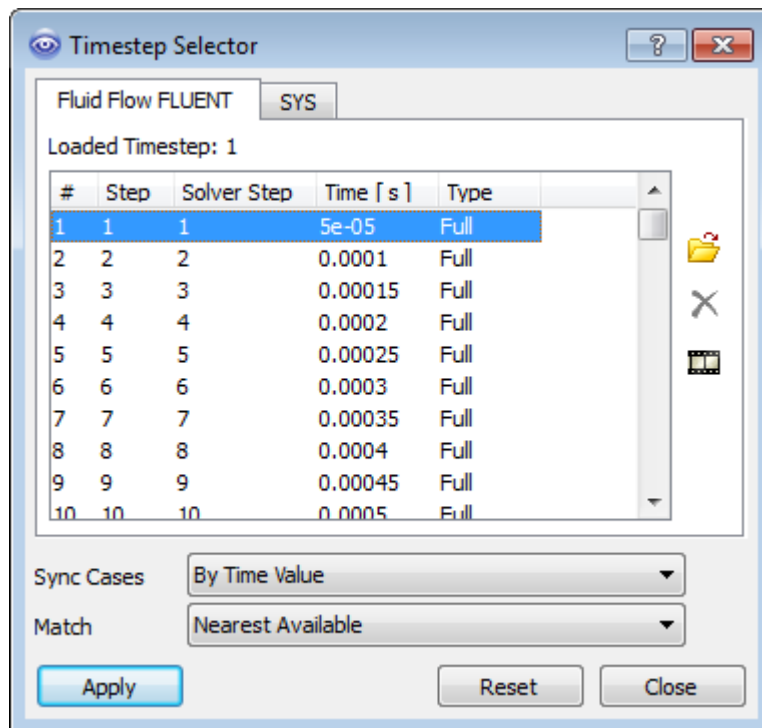
- a. Retain the default name in the **Insert Streamline** dialog box and click **OK**.




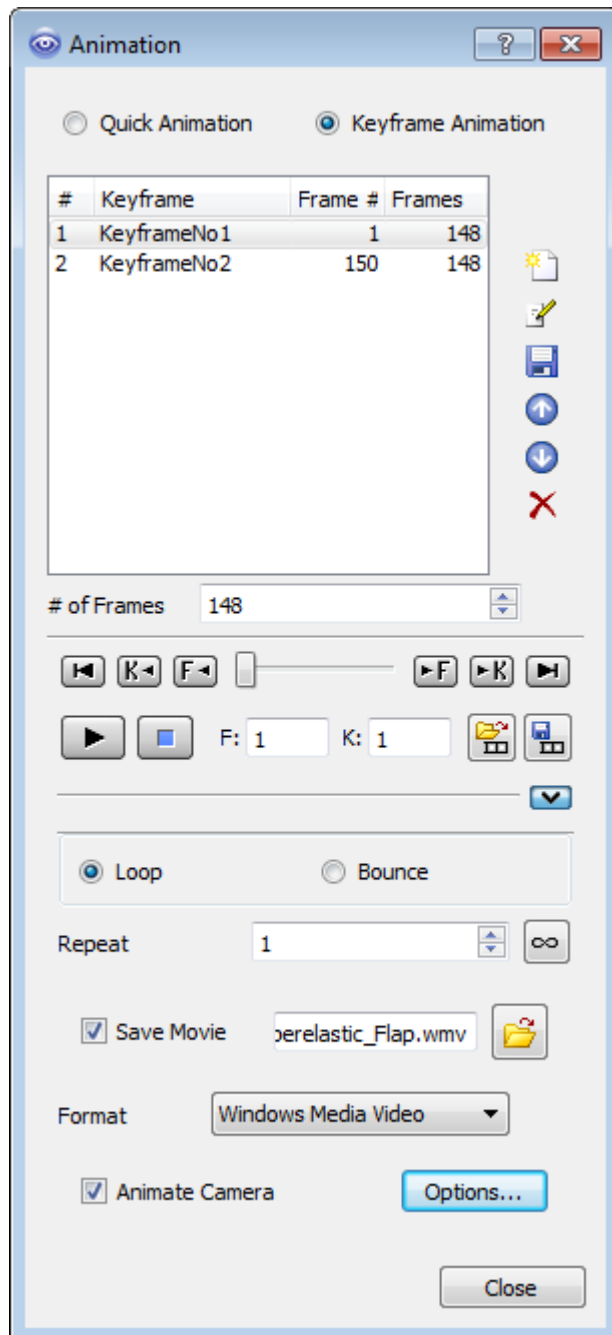
- b. In the **Details of Streamline** select **inlet** from the **Start From** drop-down list in the **Geometry** tab.
- c. In the **Symbol** tab enter 3 for **Line Width**.
- d. Click **Apply**.





5. You can view different stages in the simulation using the **Timestep Selector** icon  at the top of the window.



- a. In the **Timestep Selector** window, select **Nearest Available** from the **Match** drop-down list then double-click on some different time values to see how the flap has moved and how the mesh has responded.
 - Select **Step 1** corresponding to a **Time** of **5e-05 s** and **Apply**.
- b. Click the **Animation** icon  from the toolbar.




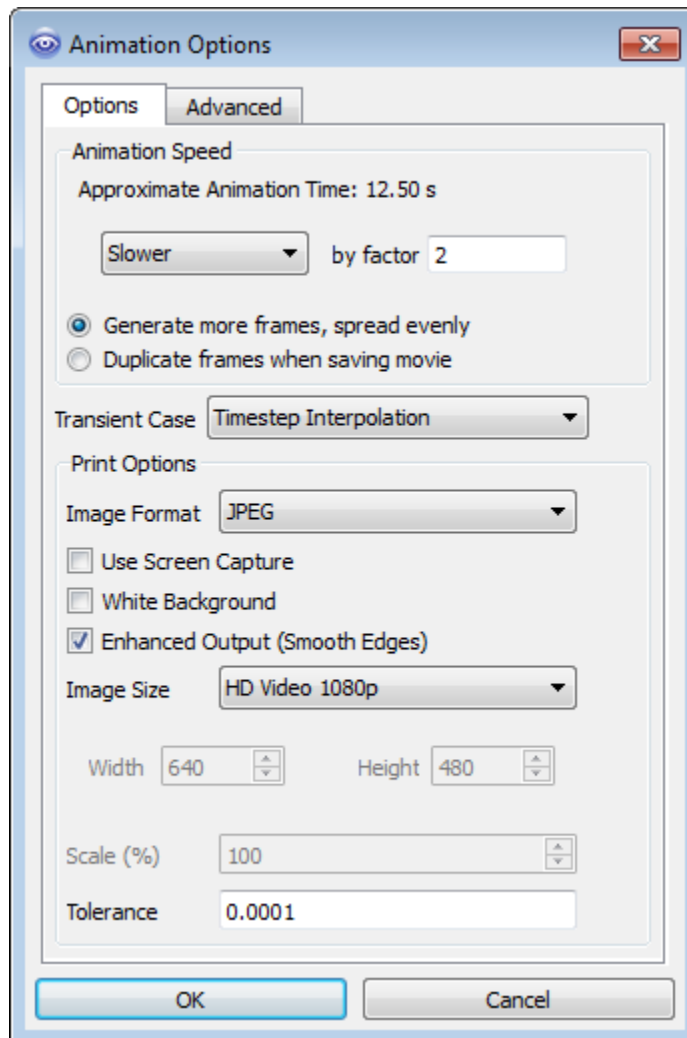
- i. Select **Keyframe Animation** in the **Animation** dialog box.
- ii. Click on the **New** button  on the right side to create a new keyframe, **KeyframeNo1**.
- iii. Enter 148 for **# of Frames**.

- iv. In the **Timestep Selector** dialog box select the final time step (**#150**) and click **Apply**.
- v. In the **Animation** dialog box create a new keyframe by clicking on the **New** button .


Note

Keyframe animations will produce a smooth variation between two or more keyframes. The changes could be in the camera-angle, the quantities plotted, or in this case the time value. The value of 148 is for the number of frames desired between **KeyframeNo1** and **KeyframeNo2**. Including the keyframes there will be 150 frames in total.

- vi. Expand the **Animation** window by clicking the down arrow  on the bottom right, if not already done.
- vii. Enable **Save Movie** and set an appropriate filename and directory.
- viii. Click on **Options...** to open the **Animation Options** dialog box.



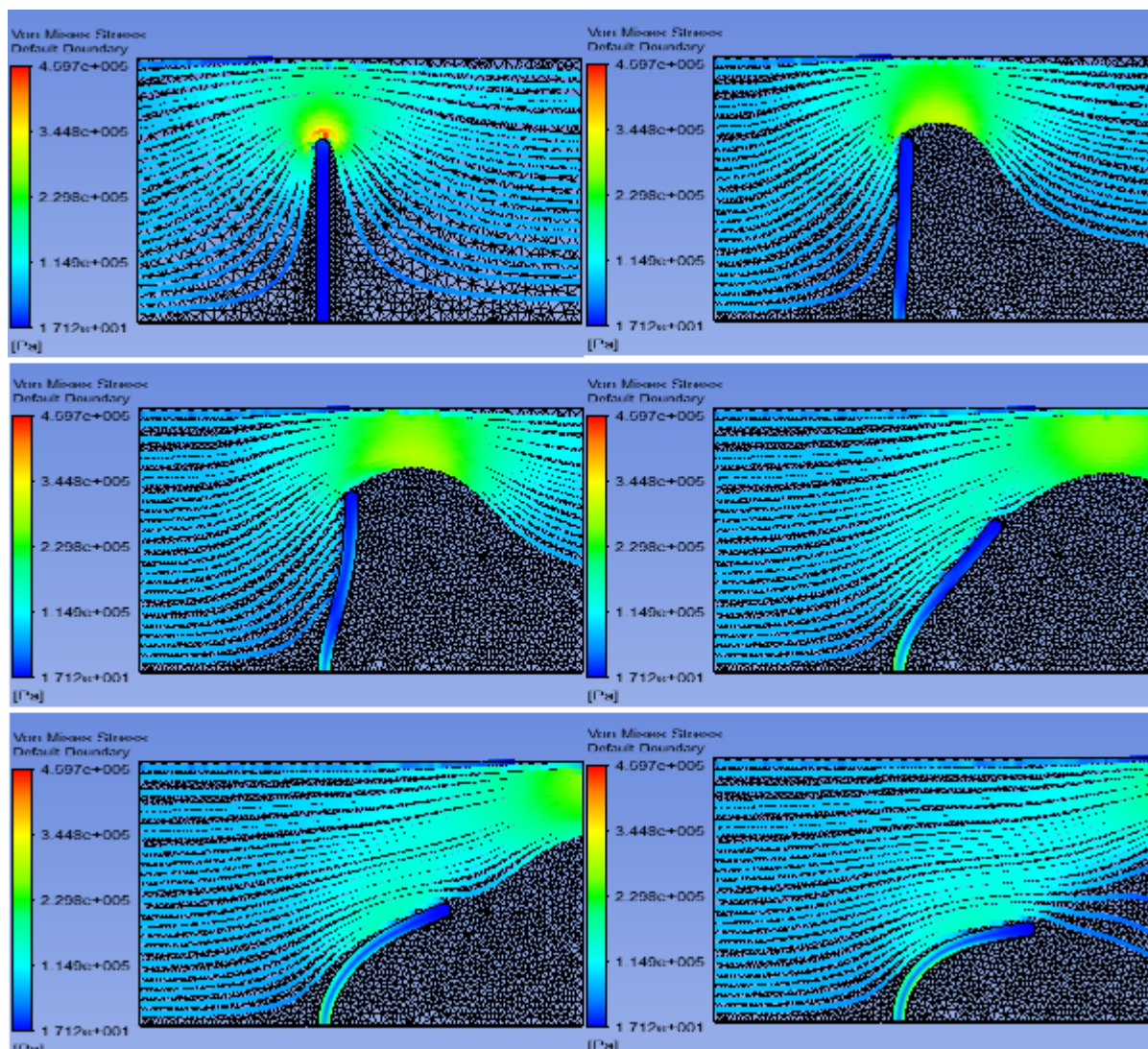
- A. In the **Animation Options** dialog box Select **Slower** in the **Animation Speed** group box.

- B. Enter 2 for **by factor**.
 - C. Select **HD Video 1080p** from the **Image Size** drop-down list.
 - D. In the **Advanced** tab select **Highest** from the **Quality** drop-down list.
 - E. Click **OK**.
- ix. Click the **Play** button , then wait for the movie to build.

Note

This will take several minutes. If you look in the **Animation** dialog box the **F:** value will show the progress from **1** to **150**.

- x. Play the movie file created.



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

3. Further Improvements

From the movie generated on the last step it can be seen that the motion is still ongoing, the flap has not reached its quasi-steady position. You may want to continue the run for additional time steps to see the rest of the motion. To do this just alter the maximum time set on the System Coupling panel (suggest from 0.0075s to 0.02s) and click update project.

If you want to try experimenting with different model setups (flow rates, material properties, mesh settings etc:

- Save the project, and then modify the settings in the relevant solver.
- On the Workbench **Project** page, right-click on the **Solution** cell (**D3**) of **System Coupling** and select **Clear Generated Data** from the context menu.

This will set both solvers back to the initial time-zero condition so you can repeat the computation.

Often one of the key challenges is to get the correct dynamic mesh settings in Fluent. A useful technique is to ask Fluent to write a TIFF image at every timestep so that you can monitor the solution in progress. For this case, you could create four **Execute Commands** (under **Calculation Activities** in Fluent) to run each time step as follows:

```
/dis/set/colors/csclassic :- Sets the background to black (easier to see mesh)
```

```
/dis/sw2 /dis/surface-mesh (symmetry_1) :- Plots mesh in new graphic window
```

```
/dis/views/restore front /dis/views/camera/zoom 2.5 :- Sets camera angle and zoom
```

```
/dis/sp "mesh-%t.tif" :- Write image to disk
```

4. Summary

- This tutorial has demonstrated how to set up and run a full 2-way coupled FSI simulation.
- All the coupling is done within Workbench, using ANSYS Mechanical (Transient Structural) and ANSYS Fluent.
- The motion of the flap is significant, and therefore it was necessary to use the dynamic re-meshing tools in Fluent to add and remove grid cells to prevent them becoming distorted.
- At present, System Coupling can only be performed in 3D. However this need not add to the complexity of the model setup. This tutorial has shown how to create a 3D domain, 1 cell thick. The 2.5D re-meshing scheme in Fluent allows for re-meshing on these triangular prism cells (normally in 3D re-meshing could only happen with tetrahedral grid cells).
- CFD-Post has been used to simultaneously post-process results from the CFD and FEA simulation, and use these to generate a movie file of this transient deformation.