# **Chapter 15: Using Dynamic Meshes**

This tutorial is divided into the following sections:

15.1. Introduction
15.2. Prerequisites
15.3. Problem Description
15.4. Setup and Solution
15.5. Summary
15.6. Further Improvements

## 15.1. Introduction

In ANSYS Fluent the dynamic mesh capability is used to simulate problems with boundary motion, such as check valves and store separations. The building blocks for dynamic mesh capabilities within ANSYS Fluent are three dynamic mesh schemes, namely, smoothing, layering, and remeshing. A combination of these three schemes is used to tackle the most challenging dynamic mesh problems. However, for simple dynamic mesh problems involving linear boundary motion, the layering scheme is often sufficient. For example, flow around a check valve can be simulated using only the layering scheme. In this tutorial, such a case will be used to demonstrate the layering feature of the dynamic mesh capability in ANSYS Fluent.

Check valves are commonly used to allow unidirectional flow. For instance, they are often used to act as a pressure-relieving device by only allowing fluid to leave the domain when the pressure is higher than a certain level. In such a case, the check valve is connected to a spring that acts to push the valve to the valve seat and to shut the flow. But when the pressure force on the valve is greater than the spring force, the valve will move away from the valve seat and allow fluid to leave, thus reducing the pressure upstream. Gravity could be another factor in the force balance, and can be considered in ANSYS Fluent. The deformation of the valve is typically neglected, thus allowing for a rigid body Fluid Structure Interaction (FSI) calculation, for which a user-defined function (UDF) is provided.

This tutorial provides information for performing basic dynamic mesh calculations by demonstrating how to do the following:

- Use the dynamic mesh capability of ANSYS Fluent to solve a simple flow-driven rigid-body motion problem.
- Set boundary conditions for internal flow.
- · Compile a User-Defined Function (UDF) to specify flow-driven rigid-body motion.
- Calculate a solution using the pressure-based solver.

## 15.2. Prerequisites

This tutorial is written with the assumption that you have completed one or more of the introductory tutorials found in this manual:

• Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 1)

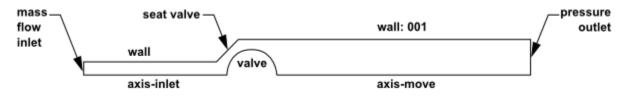
- Parametric Analysis in ANSYS Workbench Using ANSYS Fluent (p. 73)
- Introduction to Using ANSYS Fluent: Fluid Flow and Heat Transfer in a Mixing Elbow (p. 123)

and that you are familiar with the ANSYS Fluent navigation pane and menu structure. Some steps in the setup and solution procedure will not be shown explicitly.

## **15.3. Problem Description**

The check valve problem to be considered is shown schematically in Figure 15.1: Problem Specification (p. 632). A 2D axisymmetric valve geometry is used, consisting of a mass flow inlet on the left, and a pressure outlet on the right, driving the motion of a valve. In this case, the transient motion of the valve due to spring force, gravity, and hydrodynamic force is studied. Note, however, that the valve in this case is not completely closed. Since dynamic mesh problems require that at least one layer remains in order to maintain the topology, a small gap will be created between the valve and the valve seat.

### Figure 15.1: Problem Specification



# 15.4. Setup and Solution

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

15.4.1. Preparation
15.4.2. Mesh
15.4.3. General Settings
15.4.4. Models
15.4.5. Materials
15.4.6. Boundary Conditions
15.4.7. Solution: Steady Flow
15.4.8. Time-Dependent Solution Setup
15.4.9. Mesh Motion
15.4.10. Time-Dependent Solution
15.4.11. Postprocessing

### 15.4.1. Preparation

To prepare for running this tutorial:

- 1. Set up a working folder on the computer you will be using.
- 2. Go to the ANSYS Customer Portal, https://support.ansys.com/training.

### Note

If you do not have a login, you can request one by clicking **Customer Registration** on the log in page.

- 3. Enter the name of this tutorial into the search bar.
- 4. Narrow the results by using the filter on the left side of the page.
  - a. Click ANSYS Fluent under Product.
  - b. Click 15.0 under Version.
- 5. Select this tutorial from the list.
- 6. Click **Files** to download the input and solution files.
- 7. Unzip dynamic\_mesh\_R150.zip to your working folder.

The mesh and source files valve.msh and valve.c can be found in the dynamic\_mesh directory created after unzipping the file.

A user-defined function will be used to define the rigid-body motion of the valve geometry. This function has already been written (valve.c). You will only need to compile it within ANSYS Fluent.

8. Use the Fluent Launcher to start the **2D** version of ANSYS Fluent.

Fluent Launcher displays your **Display Options** preferences from the previous session.

Note that this tutorial has been generated using single precision, so you should ensure that **Double Precision** is disabled if you want to match the tutorial setup exactly.

For more information about Fluent Launcher, see Starting ANSYS Fluent Using Fluent Launcher in the User's Guide.

- 9. Ensure that the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options are enabled.
- 10. Run in Serial under Processing Options.

### 15.4.2. Mesh

1. Read the mesh file valve.msh.

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Mesh...}$ 

### 15.4.3. General Settings

1. Check the mesh.



#### Note

You should always make sure that the cell minimum volume is not negative, since ANSYS Fluent cannot begin a calculation if this is the case.

2. Change the display units for length to mm.

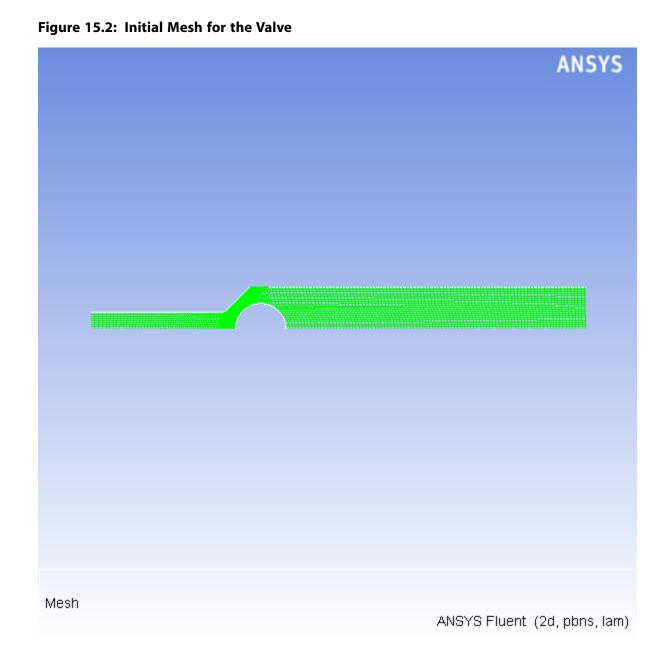
**♀**General → Units...

- a. In the Set Units dialog box select length under Quantities and mm under Units.
- b. Close the **Set Units** dialog box.
- 3. Display the mesh (Figure 15.2: Initial Mesh for the Valve (p. 635)).

# $\clubsuit General \rightarrow Display...$

💶 Mesh Displa	у		<b>—</b>
Options           Nodes           Edges           Faces           Partitions	Edge Type a All Feature Outline	Surfaces axis-inlet axis-move default-interior default-interior:012 inlet	
0	Feature Angle	int-layering outlet	-
Outline Inter	Match	New Surface  Surface Types axis clip-surf exhaust-fan fan	
Display Colors Close Help			

- a. Deselect axis-inlet, axis-move, inlet, and outlet from the Surfaces selection list.
- b. Click **Display**.



- c. Close the **Mesh Display** dialog box.
- 4. Enable an axisymmetric steady-state calculation.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation Absolute     Relative
Time	2D Space
<ul> <li>Steady</li> <li>Transient</li> </ul>	<ul> <li>Axisymmetric</li> </ul>
- Hanalent	Axisymmetric Swirl
Gravity	Units
Help	

a. Select Axisymmetric from the 2D Space list.

## 15.4.4. Models

*					
4 1					
Ŷ	м	n	d	ρ	I٩
		~	~	-	

Models
Models
Multiphase - Off
Energy - Off
Viscous - Laminar
Radiation - Off
Heat Exchanger - Off
Species - Off Discrete Phase - Off
Solidification & Melting - Off
Acoustics - Off
Edit
Lucin
Help

1. Enable the standard k-  $\varepsilon$  turbulence model.



Viscous Model	
Model         Inviscid         Laminar         Spalart-Allmaras (1 eqn)         k-epsilon (2 eqn)         Transition k-kl-omega (3 eqn)         Transition SST (4 eqn)         Reynolds Stress (5 eqn)         Scale-Adaptive Simulation (SAS)         k-epsilon Model         Standard         RNG         Realizable         Near-Wall Treatment         Standard Wall Functions         Scalable Wall Functions         Scalable Wall Functions         Non-Equilibrium Wall Functions         Enhanced Wall Treatment         User-Defined Wall Functions         Enhanced Wall Treatment         Pressure Gradient Effects         Options         Production Kato-Launder         Production Limiter	Model Constants         Cmu         0.09         C1-Epsilon         1.44         C2-Epsilon         1.92         TKE Prandtl Number         1         User-Defined Functions         Turbulent Viscosity         none         Prandtl Numbers         TKE Prandtl Number         IDR Prandtl Number         IDR Prandtl Number         Inone
OK	Cancel Help

a. Select **k-epsilon (2 eqn)** from the **Model** list and retain the default selection of **Standard** in the **k-epsilon Model** group box.

### b. Select Enhanced Wall Treatment for the Near-Wall Treatment.

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

## 15.4.5. Materials

## Materials

### Materials

Materials
Fluid
air
Solid aluminum
Create/Edit
Create/Edit Delete
Help

1. Apply the ideal gas law for the incoming air stream.

```
 \diamondsuit \mathsf{Materials} \to \mathbf{Fluid} \to \mathsf{Create/Edit...}
```

1		Order Materials by
lame	Material Type	Order Materials by
air	fluid	O Name     O Chemical Formula
Chemical Formula	Fluent Fluid Materials	
	air	Fluent Database
	Mixture	User-Defined Database
	none	-
roperties		
Density (kg/m3)	ideal-gas 🗸 Edit	
Cp (Specific Heat) (j/kg-k)	constant 👻 Edit	
	1006.43	
Thermal Conductivity (w/m-k)		
mermai Conductivity (w/m-k)	constant v Edit	
	0.0242	
Viscosity (kg/m-s)		
viscosity (kg/m-s/	constant	
	1.7894e-05	
	· ·	

- a. Select ideal-gas from the Density drop-down list.
- b. Click Change/Create.
- c. Close the Create/Edit Materials dialog box.

### **15.4.6. Boundary Conditions**

Dynamic mesh motion and all related parameters are specified using the items in the **Dynamic Mesh** task page, not through the **Boundary Conditions** task page. You will set these conditions in a later step.

1. Set the conditions for the mass flow inlet (inlet).



Since the inlet boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the inlet boundary to a mass flow inlet boundary type in ANSYS Fluent.

Boundary	Conditions
Dogingary	vonutions

Zone		
axis-inlet axis-move default-interior default-interior:0	12	
inlet int-layering outlet seat-valve valve wall wall:001		
Phase mixture	Type ▼ mass-flow-inlet ▼	ID 11
Edit Parameters Display Mesh	Copy Profiles Operating Conditions Periodic Conditions	
Help		

- a. Select mass-flow-inlet from the Type drop-down list in the Boundary Conditions task page.
- b. Click Yes when ANSYS Fluent asks you if you want to change the zone type.

The *Mass-Flow Inlet* boundary condition dialog box will open.

Mass-Flow Inlet	×			
Zone Name				
inlet				
Momentum Thermal Radiation Species DPM Multiphase U	DS			
Reference Frame Absolute	•			
Mass Flow Specification Method Mass Flow Rate				
Mass Flow Rate (kg/s) 0.0116	constant 🔻			
Supersonic/Initial Gauge Pressure (pascal)	constant 🔹			
Direction Specification Method Normal to Boundary				
Turbulence				
Specification Method Intensity and Hydraulic Diam	eter 👻			
Turbulent Intensity (%	6) 5 P			
Hydraulic Diameter (mr	n) 20			
OK Cancel Help				

- i. Enter 0.0116 kg/s for Mass Flow Rate.
- ii. Select Normal to Boundary from the Direction Specification Method drop-down list.
- iii. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- iv. Retain 5% for Turbulent Intensity.
- v. Enter 20 mm for the Hydraulic Diameter.
- vi. Click **OK** to close the **Mass-Flow Inlet** dialog box.
- 2. Set the conditions for the exit boundary (**outlet**).

# $\clubsuit Boundary \ Conditions \rightarrow \overleftarrow{E} \ outlet$

#### **Boundary Conditions**

Zone		
axis-inlet axis-move default-interior default-interior:0: inlet int-layering outlet	12	
seat-valve valve wall wall:001		
Phase mixture	Type pressure-outlet	ID 10
Edit	Copy Profiles	
Parameters	Operating Conditions	
Display Mesh	Periodic Conditions	
Help		

Since the **outlet** boundary is assigned to a wall boundary type in the original mesh, you will need to explicitly assign the outlet boundary to a pressure outlet boundary type in ANSYS Fluent.

- a. Select pressure-outlet from the Type drop-down list in the Boundary Conditions task page.
- b. Click Yes when ANSYS Fluent asks you if you want to change the zone type.

The **Pressure Outlet** boundary condition dialog box will open.

Pressure Outlet	×
Zone Name	
outlet	
Momentum Thermal Radiation Species DPM Multiphase UDS	
Gauge Pressure (pascal) 0 constant	•
Backflow Direction Specification Method From Neighboring Cell	-
Average Pressure Specification	_
Target Mass Flow Rate	
Turbulence	1
Specification Method Intensity and Hydraulic Diameter	-
Backflow Turbulent Intensity (%) 5	e
Backflow Hydraulic Diameter (mm) 50	P
OK Cancel Help	

- i. Select **From Neighboring Cell** from the **Backflow Direction Specification Method** drop-down list.
- ii. Select Intensity and Hydraulic Diameter from the Specification Method drop-down list in the Turbulence group box.
- iii. Retain 5% for **Backflow Turbulent Intensity**.
- iv. Enter 50 mm for Backflow Hydraulic Diameter.
- v. Click OK to close the Pressure Outlet dialog box.
- 3. Set the boundary type to **axis** for both the **axis-inlet** and the **axis-move** boundaries.

### **Organization** Boundary Conditions

Since the **axis-inlet** and the **axis-move** boundaries are assigned to a wall boundary type in the original mesh, you will need to explicitly assign these boundaries to an axis boundary type in ANSYS Fluent.

- a. Select axis-inlet from the Zone list and select axis from the Type list.
- b. Click **Yes** when ANSYS Fluent asks you if you want to change the zone type.
- c. Retain the default Zone Name in the Axis dialog box and click OK to close the Axis dialog box.
- d. Select axis-move from the Zone list and select axis from the Type list.
- e. Click Yes when ANSYS Fluent asks you if you want to change the zone type.
- f. Retain the default **Zone Name** in the **Axis** dialog box and click **OK** to close the **Axis** dialog box.

# 15.4.7. Solution: Steady Flow

In this step, you will generate a steady-state flow solution that will be used as an initial condition for the time-dependent solution.

1. Set the solution parameters.

# Solution Methods

Solution Methods	
Pressure-Velocity Coupling	
Scheme	
Coupled	
Spatial Discretization	
Gradient	<u>.</u>
Least Squares Cell Based 💌	
Pressure	
PRESTO!	Ξ
Density	
Second Order Upwind 🔹	
Momentum	-
Second Order Upwind 🔹	
Turbulent Kinetic Energy	
First Order Upwind 🔹	Ŧ
Transient Formulation	
Non-Iterative Time Advancement	
Frozen Flux Formulation  Frozen Transient	
High Order Term Relaxation Options	
Default	
Help	

- a. Select **Coupled** from the **Scheme** drop-down list.
- b. Select PRESTO! from the Pressure drop-down list.
- c. Retain the default of Second Order Upwind in the Density drop-down list.
- d. Retain the default of Second Order Upwind in the Momentum drop-down list.
- e. Retain the defaults of First Order Upwind in the Turbulent Kinetic Energy and Turbulent Dissipation Rate drop-down lists.
- f. Retain the default of Second Order Upwind in the Energy drop-down list.
- 2. Set the relaxation factors.

# **Solution** Controls

Solution Controls	
Flow Courant Number	
200	٦
Explicit Relaxation Factors	
Momentum 0.75	]
Pressure 0.75	
Under-Relaxation Factors	
Density	4
1	
Body Forces	
1	
Turbulent Kinetic Energy	=
0.8	
Turbulent Dissipation Rate	
0.8	
Turbulent Viscosity	
1	
Default	
Equations Limits Advanced	
Help	

a. Retain the default values for **Under-Relaxation Factors** in the **Solution Controls** task page.

3. Enable the plotting of residuals during the calculation.



Residual Monitors					<b>-X</b>
Options	Equations				
Print to Console	Residual	Monitor	Check Convergence	Absolute Criteria	<u> </u>
V Plot	continuity	<b>V</b>		0.001	
Window	x-velocity			0.001	E
Iterations to Plot	y-velocity			0.001	
1000	energy	<b>V</b>		1e-06	<b>.</b>
	Residual Values			Convergence Cr	riterion
Iterations to Store	Normalize		Iterations	absolute	•
1000			5		
	V Scale				
Compute Local Scale					
OK Plot Renormalize Cancel Help					

- a. Ensure that **Plot** is enabled in the **Options** group box.
- b. Click **OK** to close the **Residual Monitors** dialog box.
- 4. Initialize the solution.

## **Over Solution** Initialization

Solution Initialization			
Initialization Methods <ul> <li>Hybrid Initialization</li> <li>Standard Initialization</li> </ul>			
More Settings Initialize			
Patch			
Reset DPM Sources Reset Statistics			
Help			

- a. Retain the default Hybrid Initialization in the Initialization Methods group box.
- b. Click Initialize in the Solution Initialization task page.

#### Note

A warning is displayed in the console stating that the convergence tolerance of 1.000000e-06 not reached during Hybrid Initialization. This means that the default number of iterations is not enough. You will increase the number of iterations and

re-initialize the flow. For more information refer to Hybrid Initialization in the User's Guide.

c. Click More Settings....

U Hybrid Initialization	x
General Settings   Turbulence Settings   Species Settin	gs   I
Number of Iterations 20	
Explicit Under-Relaxation Factor	
Scalar Equation-0	
Scalar Equation-1	
Reference Frame	
<ul> <li>Relative to Cell Zone</li> <li>Absolute</li> </ul>	
Initialization Options	
Use Specified Initial Pressure on Inlets Use External-Aero Favorable Settings Maintain Constant Velocity Magnitude	
OK Cancel Help	

- i. Increase the Number of Iterations to 20.
- ii. Click OK to close the Hybrid Initialization dialog box.
- d. Click Initialize once more.

#### Note

Click **OK** in the **Question** dialog box, where it asks to discard the current data. The console displays that hybrid initialization is done.

#### Note

For flows in complex topologies, hybrid initialization will provide better initial velocity and pressure fields than standard initialization. This will help to improve the convergence behavior of the solver.

5. Save the case file (valve\_init.cas.gz).

File  $\rightarrow$  Write  $\rightarrow$  Case...

6. Start the calculation by requesting 150 iterations.

# Calculation

Run Calculation	
Check Case	review Mesh Motion
Number of Iterations Re 150	eporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

### Click Calculate.

The solution converges in approximately 115 iterations.

- 7. Save the case and data files (valve\_init.cas.gz and valve\_init.dat.gz).
  - File  $\rightarrow$  Write  $\rightarrow$  Case & Data...

## 15.4.8. Time-Dependent Solution Setup

1. Enable a time-dependent calculation.



General	
Mesh	
Scale	Check Report Quality
Display	
Solver	
Type Pressure-Based Density-Based	Velocity Formulation
Time Steady Transient	2D Space Planar Axisymmetric Axisymmetric Swirl
Gravity	Units
Help	

a. Select Transient from the Time list in the General task page.

## 15.4.9. Mesh Motion

1. Select and compile the user-defined function (UDF).

 $\textbf{Define} \rightarrow \textbf{User-Defined} \rightarrow \textbf{Functions} \rightarrow \textbf{Compiled...}$ 

Compiled UDFs	<b>—</b> ×-
Source Files	Header Files
Library Name libudf	Build
Load	Help

a. Click Add... in the Source Files group box.

The **Select File** dialog box will open.

- i. Select the source code valve.c in the Select File dialog box, and click OK.
- b. Click **Build** in the **Compiled UDFs** dialog box.

The UDF is already defined, but it must be compiled within ANSYS Fluent before it can be used in the solver. Here you will create a library with the default name of <code>libudf</code> in your working folder. If you want to use a different name, you can enter it in the **Library Name** field. In this case you need to make sure that you will open the correct library in the next step.

A dialog box will appear warning you to make sure that the UDF source files are in the directory that contains your case and data files. Click **OK** in the warning dialog box.

c. Click Load to load the UDF library you just compiled.

When the UDF is built and loaded, it is available to hook to your model. Its name will appear as **valve::libudf** and can be selected from drop-down lists of various dialog boxes.

2. Hook your model to the UDF library.

💶 User-Define	d Function Hooks	<b>-</b>
Initialization	none	Edit
Adjust	none	Edit
Execute at End	none	Edit
Read Case	none	Edit
Write Case	none	Edit
Read Data	reader::libudf	Edit
Write Data	writer::libudf	Edit
Execute at Exit	none	Edit
Wall Heat Flux none -		
OK Cancel Help		

**Define**  $\rightarrow$  **User-Defined**  $\rightarrow$  **Function Hooks...** 

- a. Click the Edit... button next to Read Data to open the Read Data Functions dialog box.
  - i. Select reader::libudf from the Available Read Data Functions selection list.
  - ii. Click Add to add the selected function to the Selected Read Data Functions selection list.
  - iii. Click OK to close the Read Data Functions dialog box.
- b. Click the Edit... button next to Write Data to open the Write Data Functions dialog box.
  - i. Select writer::libudf from the Available Write Data Functions selection list.
  - ii. Click Add to add the selected function to the Selected Write Data Functions selection list.
  - iii. Click OK to close the Write Data Functions dialog box.

These two functions will read/write the position of the center of gravity (CG) and velocity in the X direction to the data file. The location of the CG and the velocity are necessary for restarting a case. When starting from an intermediate case and data file, ANSYS Fluent needs to know the location of the CG and velocity, which are the initial conditions for the motion calculation. Those values are saved in the data file using the writer UDF and will be read in using the reader UDF when reading the data file.

- c. Click OK to close the User-Defined Function Hooks dialog box.
- 3. Enable dynamic mesh motion and specify the associated parameters.

## Dynamic Mesh

Dynamic Mesh		
V Dynamic Mesh		
Mesh Methods	Options	
Smoothing	In-Cylinder	
Layering	Six DOF	
Remeshing	Implicit Update	
Settings	Contact Detection	
	Settings	
Events		
Dynamic Mesh Zones		
Create/Edit	elete Delete All	
Display Zone Motion		
Preview Mesh Motion		
(Tever near modo)		
Help		

### a. Enable Dynamic Mesh in the Dynamic Mesh task page.

For more information on the available models for moving and deforming zones, see Modeling Flows Using Sliding and Dynamic Meshes in the User's Guide.

b. Disable Smoothing and enable Layering in the Mesh Methods group box.

ANSYS Fluent will automatically flag the existing mesh zones for use of the different dynamic mesh methods where applicable.

c. Click the **Settings...** button to open the **Mesh Method Settings** dialog box.

💶 Mesh Method Settings 🛛 💽
Smoothing Layering Remeshing
Options       Options       Height Based       Ratio Based       Split Factor       0.4
Collapse Factor 0.2
OK Cancel Help

- i. Click the **Layering** tab.
- ii. Select Ratio Based in the Options group box.
- iii. Retain the default settings of 0.4 and 0.2 for Split Factor and Collapse Factor, respectively.
- iv. Click OK to close the Mesh Method Settings dialog box.
- 4. Specify the motion of the fluid region (fluid-move).

## Dynamic Mesh → Create/Edit...

The valve motion and the motion of the fluid region are specified by means of the UDF valve.

Dynamic Mesh Zones	×
Zone Names	Dynamic Mesh Zones
fluid-move 🔹	
Туре	
Stationary	
Rigid Body	
Deforming	
<ul> <li>User-Defined</li> <li>System Coupling</li> </ul>	
System Coupling	
Motion Attributes Geometry Definition Meshin	g Options Solver Options
Motion UDF/Profile	
valve::libudf	
Center of Gravity Location	Center of Gravity Orientation
X (mm) 0	Theta_Z (deg)
~ (m) 0	meta_z (deg) 0
Y (mm) 0	
Create Draw Delete All	Delete Close Help

- a. Select **fluid-move** from the **Zone Names** drop-down list.
- b. Retain the default selection of Rigid Body in the Type group box.
- c. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab to hook the UDF to your model.
- d. Retain the default settings of (0, 0) mm for **Center of Gravity Location**, and 0 for **Center of Gravity Orientation**.

Specifying the CG location and orientation is not necessary in this case, because the valve motion and the initial CG position of the valve are already defined by the UDF.

- e. Click Create.
- 5. Specify the meshing options for the stationary layering interface (**int-layering**) in the **Dynamic Mesh Zones** dialog box.

Dynamic Mesh Zones		<b>—</b>
Zone Names int-layering Type © Stationary © Rigid Body © Deforming © User-Defined © System Coupling	Dynamic Mesh Zones fluid-move	
Motion Attributes Geometry Definition Meshin	g Options Solver Options	
Adjacent Zone fluid-move Co	ell Height (mm) 0.5	constant 👻
Adjacent Zone fluid-inlet Co	ell Height (mm)	constant 🔹
Create Draw	Delete All Delete	Close Help

- a. Select int-layering from the Zone Names drop-down list.
- b. Select Stationary in the Type group box.
- c. Click the **Meshing Options** tab.
  - i. Enter 0.5 mm for Cell Height of the fluid-move Adjacent Zone.
  - ii. Retain the default value of 0 mm for the **Cell Height** of the fluid-inlet **Adjacent zone**.
- d. Click Create.
- 6. Specify the meshing options for the stationary outlet (outlet) in the Dynamic Mesh Zones dialog box.
  - a. Select outlet from the Zone Names drop-down list.
  - b. Retain the previous selection of **Stationary** in the **Type** group box.
  - c. Click the **Meshing Options** tab and enter 1.9 mm for the **Cell Height** of the fluid-move **Adjacent Zone**.
  - d. Click Create.
- 7. Specify the meshing options for the stationary seat valve (**seat-valve**) in the **Dynamic Mesh Zones** dialog box.
  - a. Select **seat-valve** from the **Zone Names** drop-down list.

- b. Retain the previous selection of Stationary in the Type group box.
- c. Click the **Meshing Options** tab and enter 0.5 mm for **Cell Height** of the fluid-move **Adjacent Zone**.
- d. Click Create.
- 8. Specify the motion of the valve (valve) in the Dynamic Mesh Zones dialog box.
  - a. Select **valve** from the **Zone Names** drop-down list.
  - b. Select **Rigid Body** in the **Type** group box.
  - c. Click the **Motion Attributes** tab.
    - i. Ensure that **valve::libudf** is selected from the **Motion UDF/Profile** drop-down list to hook the UDF to your model.
    - ii. Retain the default settings of (0, 0) mm for **Center of Gravity Location**, and 0 for **Center of Gravity Orientation**.
  - d. Click the **Meshing Options** tab and enter 0 mm for the **Cell Height** of the fluid-move **Adjacent zone**.
  - e. Click Create and close the Dynamic Mesh Zones dialog box.

In many MDM problems, you may want to preview the mesh motion before proceeding. In this problem, the mesh motion is driven by the pressure exerted by the fluid on the valve and acting against the inertia of the valve. Hence, for this problem, mesh motion in the absence of a flow field solution is meaningless, and you will not use this feature here.

### **15.4.10. Time-Dependent Solution**

1. Set the solution parameters.

Solution Methods

Pressure-Velocity Coupling	
Scheme	
PISO 👻	
Skewness Correction	
0	
Neighbor Correction	
1	
Skewness-Neighbor Coupling	
Spatial Discretization	
Gradient	-
Least Squares Cell Based 🗸	
Pressure	
PRESTO!	Ξ
Density	
Second Order Upwind 🗸	
Momentum	
Second Order Upwind 🗸	
Turbulent Kinetic Energy	
First Order Upwind 🔹	Ŧ
Transient Formulation	
First Order Implicit 👻	
Non-Iterative Time Advancement	
Frozen Flux Formulation	
High Order Term Relaxation Options	
Default	
Help	

- a. Select **PISO** from the **Scheme** drop-down list in **Pressure-Velocity Coupling** group box.
- b. Enter 0 for Skewness Correction.
- c. Retain all of the other previously set schemes and defaults.
- 2. Set the relaxation factors.

# Solution Controls

Solution Controls	
Under-Relaxation Factors	
Pressure	<b>_</b>
0.6	
Density	
1	Ξ
Body Forces	
1	
Momentum	-
0.7	
Turbulent Kinetic Energy	
0.4	
	Ŧ
Default	
Equations Limits Advanced	
Help	

- a. Enter 0.6 for Pressure in the Under-Relaxation Factors group box.
- b. Enter 0.4 for Turbulent Kinetic Energy.
- c. Enter 0.4 for **Turbulent Dissipation Rate**.
- 3. Request that case and data files are automatically saved every 50 time steps.

 $\textcircled{Calculation Activities (Autosave Every (Time Steps)) \rightarrow Edit...}$ 

💶 Autosave 📃
Save Data File Every (Time Steps) 50
Data File Quantities
Save Associated Case Files
<ul> <li>Only if Modified</li> <li>Each Time</li> </ul>
File Storage Options
Retain Only the Most Recent Files Maximum Number of Data Files Only Associated Case Files are Retained
File Name
C:\dynamic_mesh\valve_tran.gz Browse
Append File Name with flow-time
Decimal Places in File Name 6
OK Cancel Help

- a. Enter 50 for Save Data File Every (Time Steps).
- b. Enter valve\_tran.gz in the File Name text box.
- c. Select flow-time from the Append File Name with drop-down list.

When ANSYS Fluent saves a file, it will append the flow time value to the file name prefix (valve\_tran). The gzipped standard extensions (.cas.gz and .dat.gz) will also be appended.

- d. Click **OK** to close the **Autosave** dialog box.
- 4. Create animation sequences for the static pressure contour plots and velocity vectors plots for the valve.

# $\bigcirc$ Calculation Activities (Solution Animations) $\rightarrow$ Create/Edit...

Use the solution animation feature to save contour plots of temperature every five time steps. After the calculation is complete, you use the solution animation playback feature to view the animated temperature plots over time.

💶 Solu	tion Animation				<b>-X</b>
Animatio	n Sequences 2				
Active	Name	Every	When		*
	pressure	5	Time Step	▼ Defin	e
	vv	5	Time Step	▼ Defin	e
	sequence-3	1	Iteration	▼ Defin	e
	sequence-4		Iteration	▼ Defin	e
	sequence-5		Iteration	▼ Defin	e
OK Cancel Help					

- a. Set Animation Sequences to 2.
- b. Enter pressure in the Name text box for the first animation.
- c. Enter vv in the **Name** text box for the second animation.
- d. Set **Every** to 5 for both animation sequences.

The default value of 1 instructs ANSYS Fluent to update the animation sequence at every time step. For this case, this would generate a large number of files.

- e. Select **Time Step** from the **When** drop-down list for pressure and vv.
- f. Click the **Define...** button next to pressure to open the **Animation Sequence** dialog box.

Animation Sequence	<b>—</b> ו	
Sequence Parameters          Storage Type       Name         In Memory       pressure         Metafile       Window         PPM Image       Storage Directory	Display Type Mesh Contours Pathlines Particle Tracks Vectors XY Plot Monitor Monitor Monitor Type Residuals Create Tedit	
OK Cancel Help		

i. Retain the default selection of **Metafile** in the **Storage Type** group box.

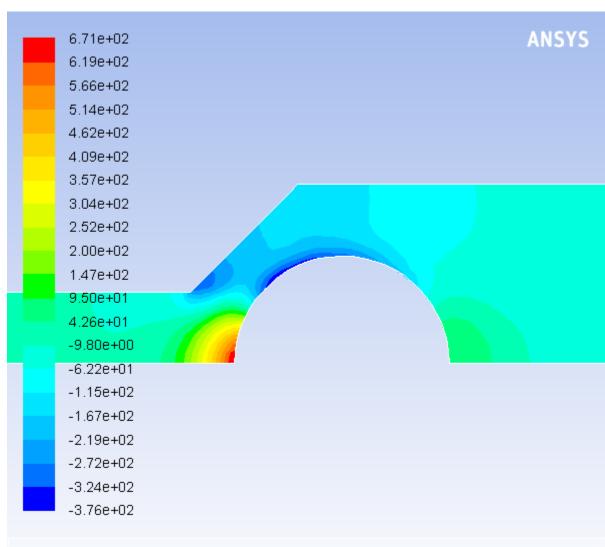
#### Note

If you want to store the plots in a folder other than your working folder, enter the folder path in the **Storage Directory** text box. If this field is left blank (the default), the files will be saved in your working folder (that is, the folder where you started ANSYS Fluent).

- ii. Set **Window** number to 1 and click **Set**.
- iii. Select **Contours** in the **Display Type** group box to open the **Contours** dialog box.

Contours	×
Options	Contours of
V Filled	Pressure
Node Values	Static Pressure
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>	Min Max
Clip to Range	0
Draw Profiles	
Draw Mesh	Surfaces 🔋 🗏 🚍
	axis-inlet
Levels Setup	axis-move
20 ( 1	default-interior:012
	inlat
Surface Name Pattern	New Surface 🕶
Match	Surface Types
	axis
	clip-surf
	exhaust-fan
	fan 👻
Display	Compute Close Help

- A. Enable **Filled** in the **Options** group box.
- B. Retain the default selection of **Pressure...** and **Static Pressure** from the **Contours of** dropdown lists.
- C. Click **Display** (Figure 15.3: Contours of Static Pressure at t=0 s (p. 661)).
- D. Close the **Contours** dialog box.



### Figure 15.3: Contours of Static Pressure at t=0 s

Contours of Static Pressure (pascal) (Time=0.0000e+00) ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

iv. Click **OK** in the **Animation Sequence** dialog box.

The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **pressure** in the **Solution Animation** dialog box will be enabled.

- g. Click the **Define...** button next to vv to open the **Animation Sequence** dialog box.
  - i. Retain the default selection of **Metafile** in the **Storage Type** group box.
  - ii. Set Window to 2 and click Set.
  - iii. Select **Vectors** in the **Display Type** group box to open the **Vectors** dialog box.

<b>Vectors</b>		×
Options	Vectors of	
	Velocity	
<ul> <li>Global Range</li> <li>Auto Range</li> </ul>		•
Clip to Range	Color by	
Auto Scale	Velocity	•
Draw Mesh	Velocity Magnitude	•
Style	Min Max	
arrow	0	
Scale Skip		
	Surfaces	
	axis-inlet	<u>^</u>
Vector Options	axis-move default-interior	=
	default-interior:012	
Custom Vectors	inlet	
	int-layering	-
Surface Name Pattern		
Match	New Surface 💌	
	Surface Types	
	axis	*
	clip-surf	
	exhaust-fan	
	fan	-
Display	Compute Close Help	

- A. Retain all the other default settings.
- B. Click **Display** (Figure 15.4: Vectors of Velocity at t=0 s (p. 663)).
- C. Close the **Vectors** dialog box.



Velocity Vectors Colored By Velocity Magnitude (m/s) (Time=0.0000e+00) ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

iv. Click **OK** in the **Animation Sequence** dialog box.

The **Animation Sequence** dialog box will close, and the check box in the **Active** column next to **vv** in the **Solution Animation** dialog box will be enabled.

- h. Click **OK** to close the **Solution Animation** dialog box.
- 5. Set the time step parameters for the calculation.

## **C**Run Calculation

Run Calculation	
Check Case	Preview Mesh Motion
Time Stepping Method	Time Step Size (s)
Settings	Number of Time Steps
0-6-6-6	150
Options Extrapolate Variables Data Sampling for Time S Sampling Interval 1 Time Sampled (s	Sampling Options
Max Iterations/Time Step	Reporting Interval
Profile Update Interval	
Data File Quantities	Acoustic Signals
Calculate	
Help	

- a. Enter 0.0001 s for Time Step Size.
- b. Retain 20 for Max Iterations/Time Step.

In the accurate solution of a real-life time-dependent CFD problem, it is important to make sure that the solution converges at every time step to within the desired accuracy. Here the first few time steps will only come to a reasonably converged solution.

6. Save the initial case and data files for this transient problem (valve\_tran-0.000000.cas.gz and valve\_tran-0.000000.dat.gz).

```
\textbf{File} \rightarrow \textbf{Write} \rightarrow \textbf{Case \& Data...}
```

7. Request 150 time steps and calculate a solution.

### Run Calculation

### Extra

If you decide to read in the case file that is provided for this tutorial on the Customer Portal, you will need to compile the UDF associated with this tutorial in your working folder. This is necessary because ANSYS Fluent will expect to find the correct UDF libraries in your working folder when reading the case file.

The UDF (valve.c) that is provided can be edited and customized by changing the parameters as required for your case. In this tutorial, the values necessary for this case were preset in the source code. These values may be modified to best suit your model.

## 15.4.11. Postprocessing

- 1. Inspect the solution at the final time step.
  - a. Inspect the contours of static pressure in the valve (Figure 15.5: Contours of Static Pressure After 150 Time Steps (p. 666)).

Graphics and Animations →  $\stackrel{\frown}{=}$  Contours → Set Up...

#### Note

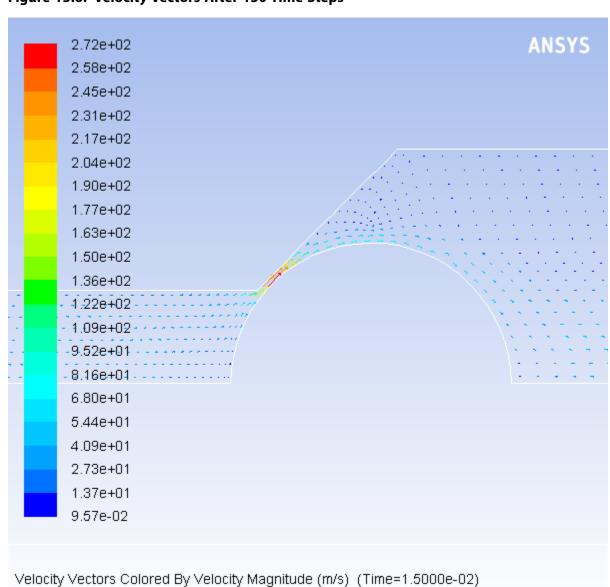
You may need to switch to Window 1 (using the drop-down list at the upper left corner of the graphics window) to view the contour plot.

1.41e+04	ANSYS
1.21e+04	
1.01e+04	
8.07e+03	
6.08e+03	
4.09e+03	
2.10e+03	
1.06e+02	
-1.89e+03	
-3.88e+03	
-5.87e+03	
-7.86e+03	
-9.85e+03	
-1.18e+04	
-1.38e+04	
-1.58e+04	
-1.78e+04	
-1.98e+04	
-2.18e+04	
-2.38e+04	
-2.58e+04	
Contours of Static Pressure (pascal) (Time=1.5000e-02) ANSYS Fluent (axi,	pbns, dynamesh, ske, transient)

### Figure 15.5: Contours of Static Pressure After 150 Time Steps

b. Inspect the velocity vectors near the point where the valve meets the seat valve (Figure 15.6: Velocity Vectors After 150 Time Steps (p. 667)).

Graphics and Animations →  $\overline{\equiv}$  Vectors → Set Up...



ANSYS Fluent (axi, pbns, dynamesh, ske, transient)

Figure 15.6: Velocity Vectors After 150 Time Steps

2. Play the animation of the pressure contours.

a.  $\bigcirc$  Graphics and Animations  $\rightarrow \stackrel{\frown}{\equiv}$  Solution Animation Playback  $\rightarrow$  Set Up...

Playback	<b>EX</b>
Playback	Animation Sequences
Playback Mode Play Once	Sequences pressure vv
1 Frame	
Keplay Speed Fast	Delete Delete All
Write/Record Format Animation Frames	Picture Options
Write Read Close Help	

b. Select pressure from the **Sequences** list in the **Animation Sequences** box of the **Playback** dialog box.

If the **Sequences** list is empty, click **Read...** to select the pressure.cxa sequence file from your working directory.

The playback control buttons will become active.

- c. Set the slider bar above **Replay Speed** about halfway in between **Slow** and **Fast**.
- d. Retain the default settings in the rest of the dialog box and click the 📩 button.

You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 1, which corresponds to the **Window** number for pressure that you set in the **Animation Sequence** dialog box.

- 3. Play the animation of the velocity vectors.
  - a. Select vv from the Sequences list in the Animation Sequences box of the Playback dialog box.

If the **Sequences** list does not contain vv, click **Read...** to select the vv. cxa sequence file from your working directory.

b. Retain the default settings in the rest of the dialog box and click the 上 button.

You may have to change the Viewer window to see the animation. In the drop-down menu at the top of the Viewer, set the window number to 2, which corresponds to the **Window** number for vv that you set in the **Animation Sequence** dialog box.

For additional information on animating the solution, see Modeling Transient Compressible Flow (p. 257) and see Animating the Solution of the User's Guide.

c. Close the **Playback** dialog box.

- 4. You can also inspect the solution at different intermediate time steps.
  - a. Read the corresponding case and data files (for example, valve\_tran-1-0.010000.cas.gz and valve\_tran-1-0.010000.dat.gz).

 $\textbf{File} \rightarrow \textbf{Read} \rightarrow \textbf{Case \& Data...}$ 

b. Display the desired contours and vectors.

## 15.5. Summary

In this tutorial, a check valve is used to demonstrate the dynamic layering capability within ANSYS Fluent, using one of the three dynamic mesh schemes available. You were also shown how to perform a one degree of freedom (1DoF) rigid body FSI by means of a user-defined function (UDF). ANSYS Fluent can also perform a more general six degrees of freedom (6DoF) rigid body FSI using a built-in 6DoF solver.

If you decide to run this tutorial in parallel, make sure you use **Principal Axes** as the partitioning method.

## **15.6. Further Improvements**

This tutorial guides you through the steps to generate an initial first-order solution. You may be able to increase the accuracy of the solution further by using an appropriate higher-order discretization scheme. For a more accurate solution, you can increase the number of layers across the valve seat area. This can be achieved either by using a finer mesh at the valve seat area and/or using a non-constant layer height instead of a constant layer height, as demonstrated in this tutorial.