

# Ansys CFX Tutorial

## Simplified Aortic Valve Analysis

Developed and curated by the Ansys Academic Development Team

Michelle Boots, The University of Newcastle

[education@ansys.com](mailto:education@ansys.com)

## Summary

In this tutorial, we will explore the differences between steady-state and transient simulations in Ansys CFX, learn how to set up the boundary conditions, apply pulse flows and initiate expressions and create animations.

This tutorial assumes familiarity with running CFD simulations using Ansys Workbench.

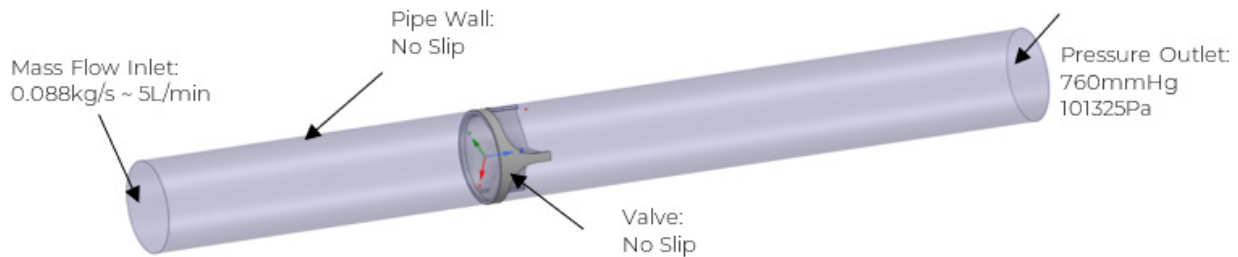
Tutorials on how to [define a mesh](#) and [run CFD simulations using Ansys Fluent](#), among others, can be found on the [Ansys Education Resources site](#).

## Table of Contents

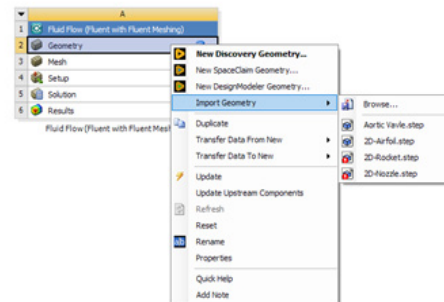
Exercise 1: Simplified 3D Aortic Valve Steady State:.....	3
Exercise 2: 3D Aortic Valve Transient Pulse Flow:.....	8

## Exercise 1: Simplified 3D Aortic Valve Steady State:

For this exercise, we will investigate and conduct a steady-state analysis on a fixed aortic valve prosthetic design. The primary objective of this initial lab is to familiarize you with various geometry editing techniques, including how to import and modify geometries, as well as how to establish fluid domains. The following image depicts the simulation conditions.

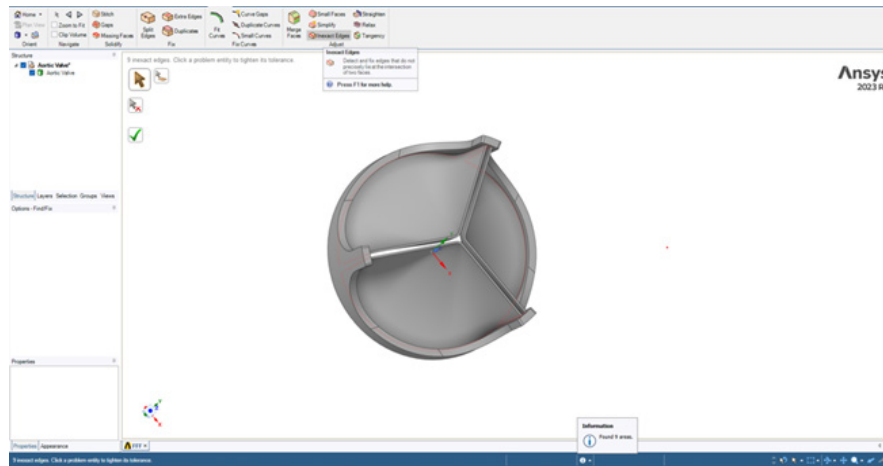


Begin by loading a new instance of Ansys CFX into Ansys Workbench. Retrieve the associated STEP file for this exercise and import the geometry into Ansys Fluent. You can do this by right-clicking on “Fluent” in Workbench and selecting ‘Import Geometry’.

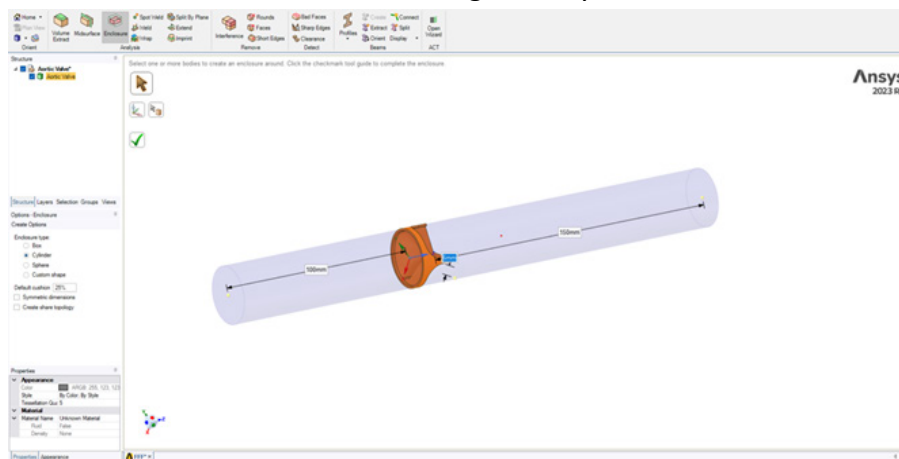


After importing the geometry, right-click once more and open it in ‘SpaceClaim.’ Ansys SpaceClaim<sup>1</sup> is a non-parametric CAD software with extensive capabilities for importing, editing, and preparing geometry for simulation. Once the geometry is loaded, navigate to the Repair tab. We will start by inspecting the geometry for any potential issues, such as the geometry loading as a series of surfaces, small gaps, voids, or interfering faces. Go through each duplicate, small curves, inexact edges, and interference, this will check and mark any bad faces red, if they appear click the green tick. This will improve the likelihood of meshing errors.

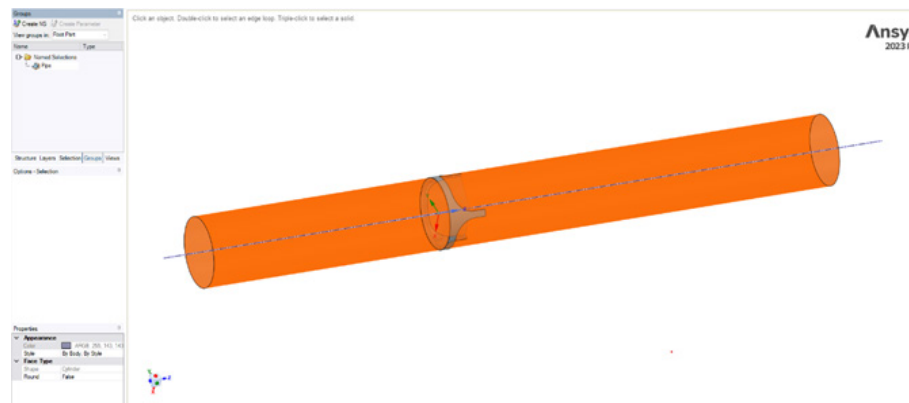
<sup>1</sup> After the 2023R1 release, Ansys SpaceClaim became a legacy product and Ansys Discovery (a simulation-driven design tool that combines instant physics simulation, high fidelity simulation and interactive geometry modeling in a single easy-to-use experience) became the primary built-in Geometry tool. If you want to learn more about specifically modeling in Ansys Discovery, check out this Ansys Innovation Course [“Learn Solid Modeling with Ansys Discovery”](#).



Next, we will create our fluid domain using the enclosure feature. Specify a thickness of -6mm, extending 100mm in front and 150mm behind the geometry, as illustrated in the following image.

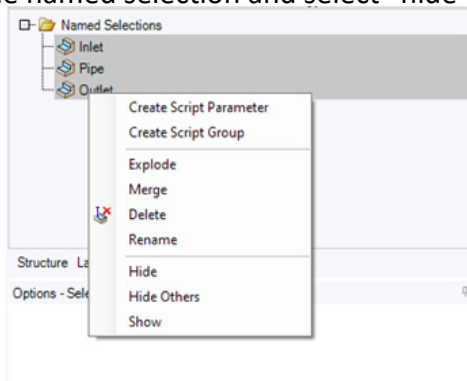


Now, we need to create our named selections. To do this, we will use the SpaceClaim interface. Click on the Select tool, then access the Group menu located just below your model tree. Select the Pipe wall faces and create a group, giving it a name. You should now observe the named selection in the top left corner.



Now, repeat the same process for the valve, inlet, and outlet. Create named selections for each of them using the SpaceClaim interface.

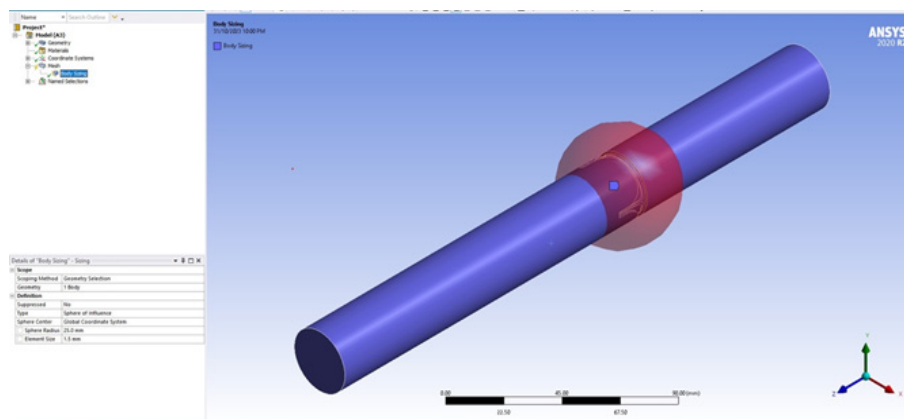
Note: You can hide faces by right-clicking on the face, selecting “Face,” and then choosing “Hide Face” or you can right click the named selection and select “hide”.



After completing the necessary tasks in SpaceClaim, close SpaceClaim and transition to the meshing environment.

For this exercise, we will implement a straightforward coarse mesh. Begin by setting the global mesh size to 5mm.

We will now add a sphere of influence, to do this, navigate to the ‘Mesh’ tab and select ‘Sizing’ from the menu. Right-click on ‘Mesh,’ then choose ‘Insert’ and select ‘Sizing.’ Under ‘Type,’ opt for ‘Sphere of Influence.’ Scope the ‘Geometry’ to the body of the fluid domain, and for ‘Sphere Center,’ select the global coordinate system. Specify the ‘Sphere Radius’ as 25mm and the ‘Element Size’ as 1.5mm. This configuration instructs Ansys to employ a 1.5mm element size within a 25mm radius of the coordinate system we’ve established.

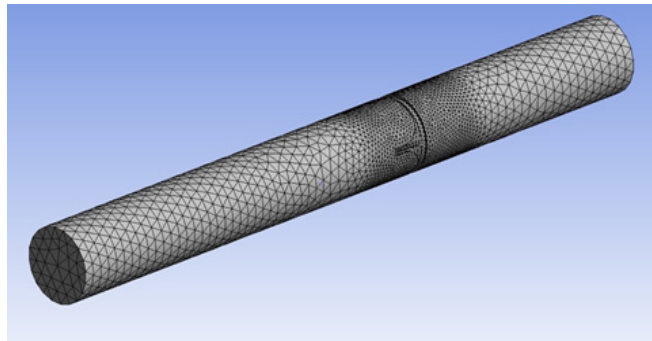


Now, let’s add several inflation layers around the valve to enhance our ability to capture boundary layer effects. To add an ‘Inflation’ layer, right-click on ‘Mesh,’ select ‘Insert,’ and choose ‘Inflation.’ Scope the ‘Geometry’ to the body of the fluid domain and the ‘Boundary’ to the named selection ‘Valve.’

Under ‘Inflation Options,’ change the setting to ‘Total Thickness’ so we can define the total thickness. Set the total thickness to 0.2mm and set the ‘Minimum Layers’ to 8, leaving the ‘Growth Rate’ as 1.2. These settings will ensure the creation of inflation layers around the valve, which is essential for accurately capturing boundary layer effects.

Details of "Inflation" - Inflation	
<div> <div>Scope</div> <div> <div>Scoping Method</div> <div>Geometry Selection</div> </div> <div> <div>Geometry</div> <div>1 Body</div> </div> </div>	
<div> <div>Definition</div> <div> <div>Suppressed</div> <div>No</div> </div> <div> <div>Boundary Scoping Method</div> <div>Named Selections</div> </div> <div> <div>Boundary</div> <div>Valve</div> </div> <div> <div>Inflation Option</div> <div>Total Thickness</div> </div> <div> <div>Number of Layers</div> <div>8</div> </div> <div> <div>Growth Rate</div> <div>1.2</div> </div> <div> <div>Maximum Thickness</div> <div>0.2 mm</div> </div> <div> <div>Inflation Algorithm</div> <div>Pre</div> </div> </div>	

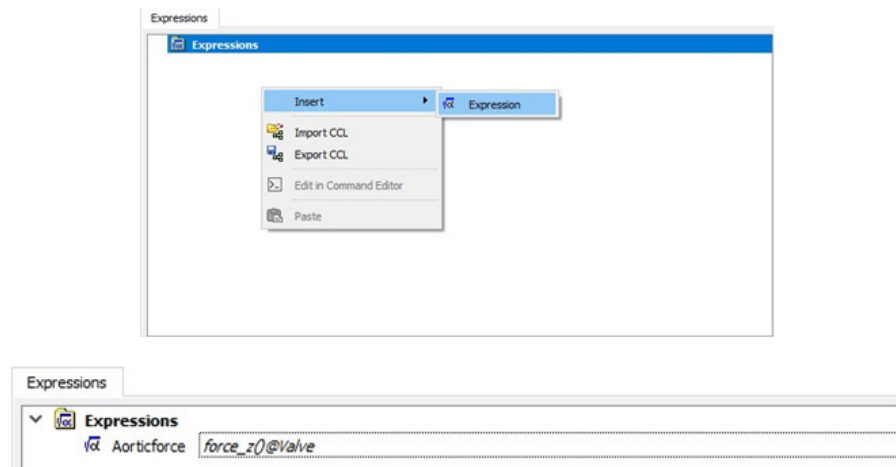
After completing the previous steps, generate and update the mesh. It should resemble the following:



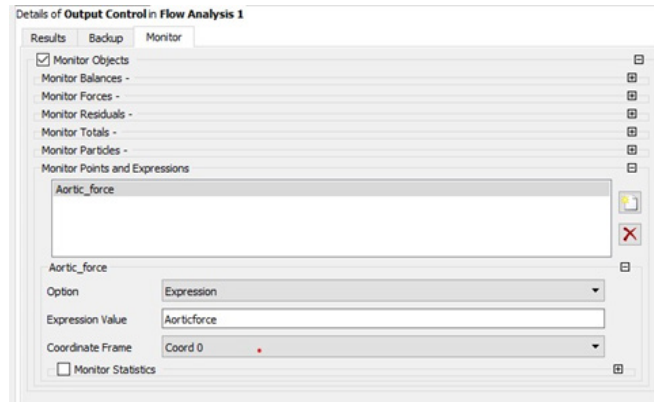
Now we can move over to CFX Pre, as you have done in previous labs set the simulation up with the following boundary conditions.

1. Set the turbulence model to Shear Stress Transport
2. Set the fluid domain to Water (Newtonian Blood,  $\rho = 1060 \text{ kg/m}^3$ ,  $\mu = 0.035 \text{ kg/ms}$ ).
3. Set the Inlet to Mass flow of  $0.088 \text{ kg/s}$ .
4. Set the Outlet to a Static Pressure of  $760 \text{ mmHg}$ .
5. Set the Pipe to a wall boundary.
6. Set the Valve to a Wall boundary.

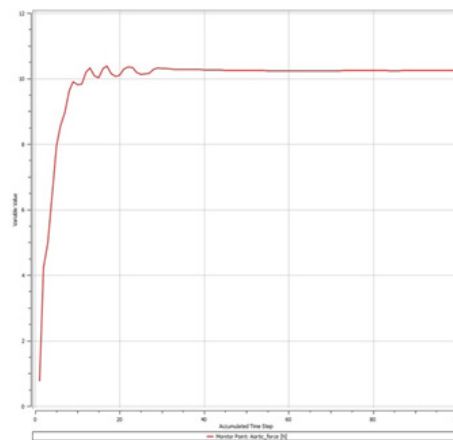
Now, configure a force output to understand the force acting on the valve. Click on 'Expressions' and create an expression called 'Aorticforce.' Use the expression 'force\_z()@Valve' to instruct the solver to determine the force in the z-direction on the named selection 'Valve.'



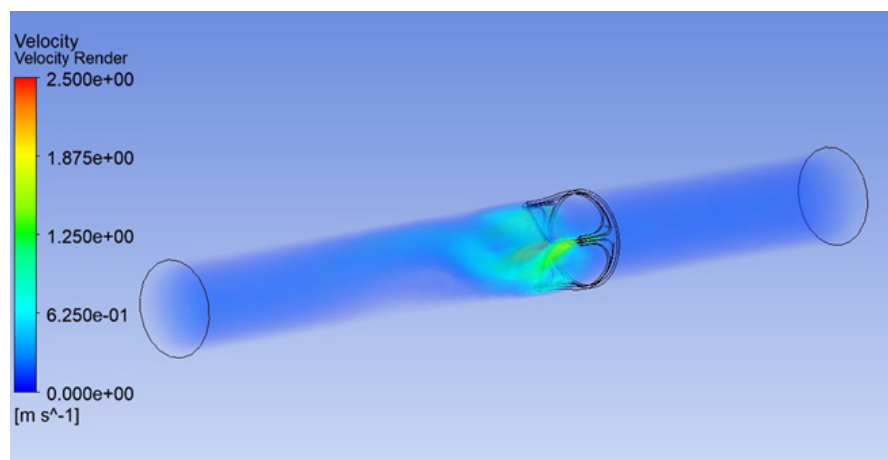
Before simulating, let's set up an output plot to visualize the convergence over iterations. Navigate to 'Solution Controls' and select 'Monitors.' Add a monitor for the new expression we've created. This will generate an output plot in our solver.



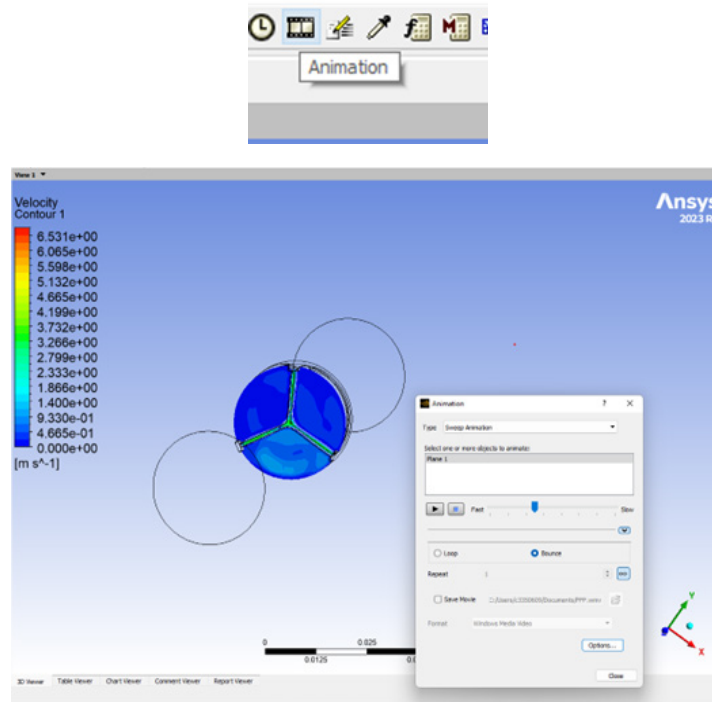
After setting up the output plot, proceed to run the calculation. The force plot should resemble the following:



Let's also look at the Velocity volumetric render.



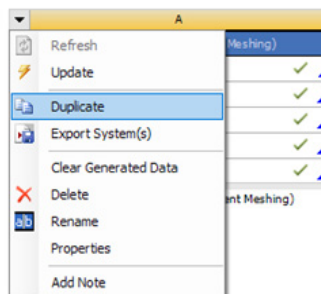
Now let create a sweep animation, first create plane on the XY-Plane. Create a velocity contour on this plain. Now click on the animation button in the top row, select "plane 1," and press the play button. This will generate a sweep animation based what your displaying on "plane 1".



Is this simulation valid? What simplifications have we made with this model?

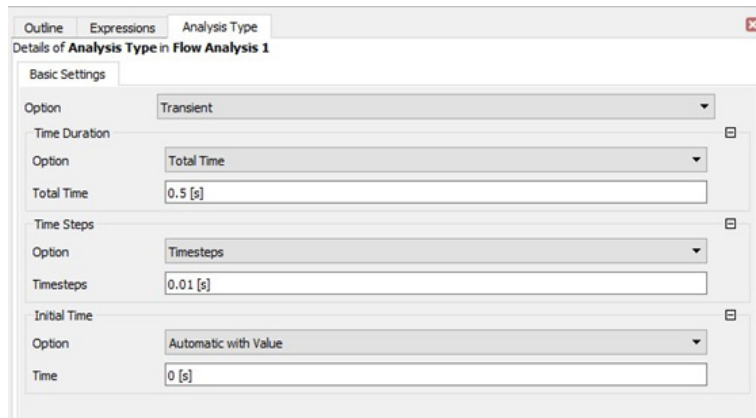
## Exercise 2: 3D Aortic Valve Transient Pulse Flow:

In this exercise, we're extending the previous simulation by examining a transient pulse simulation on the same geometry and comparing the flow. We'll begin by duplicating the workbench since most of our parameters will remain constant.

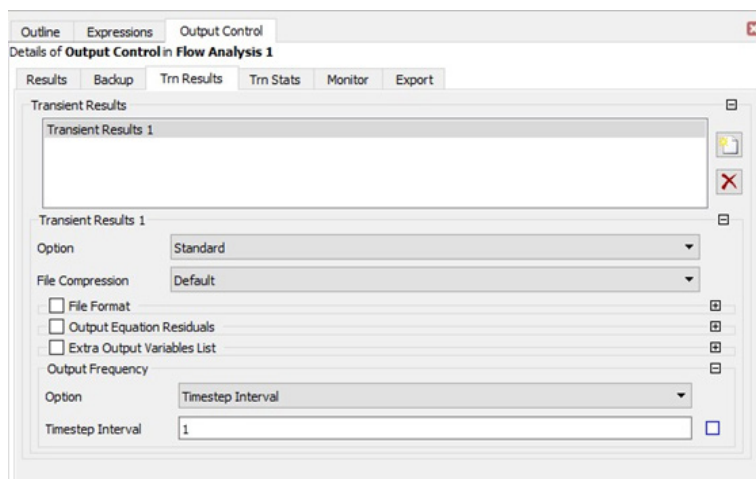


After updating the mesh and reopening CFX Pre, switch the simulation from steady state to transient. You can do this in the 'Analysis Settings' section. Set the run time to 0.5 seconds and the time step size to 0.01. We will initiate the simulation from time zero.

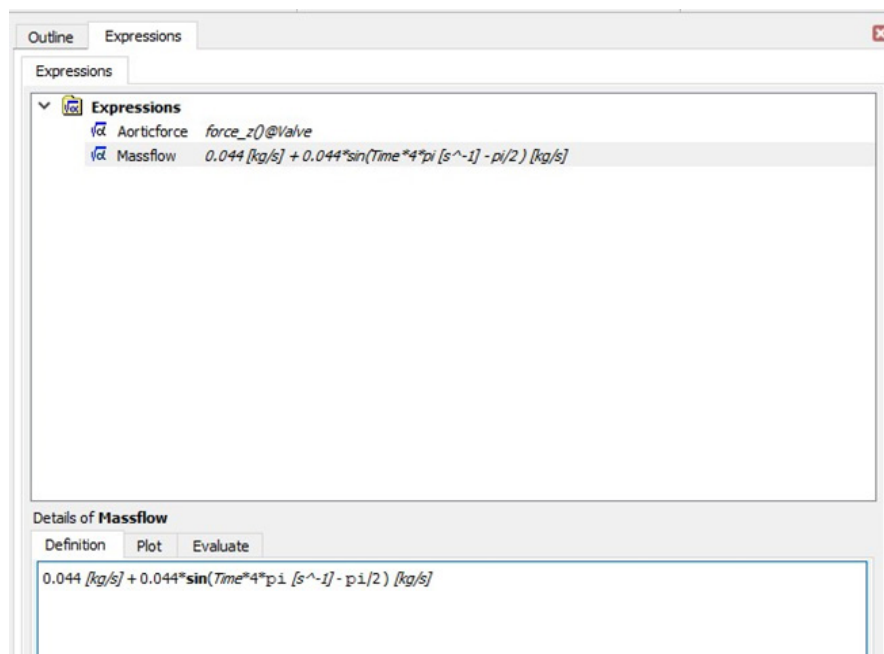




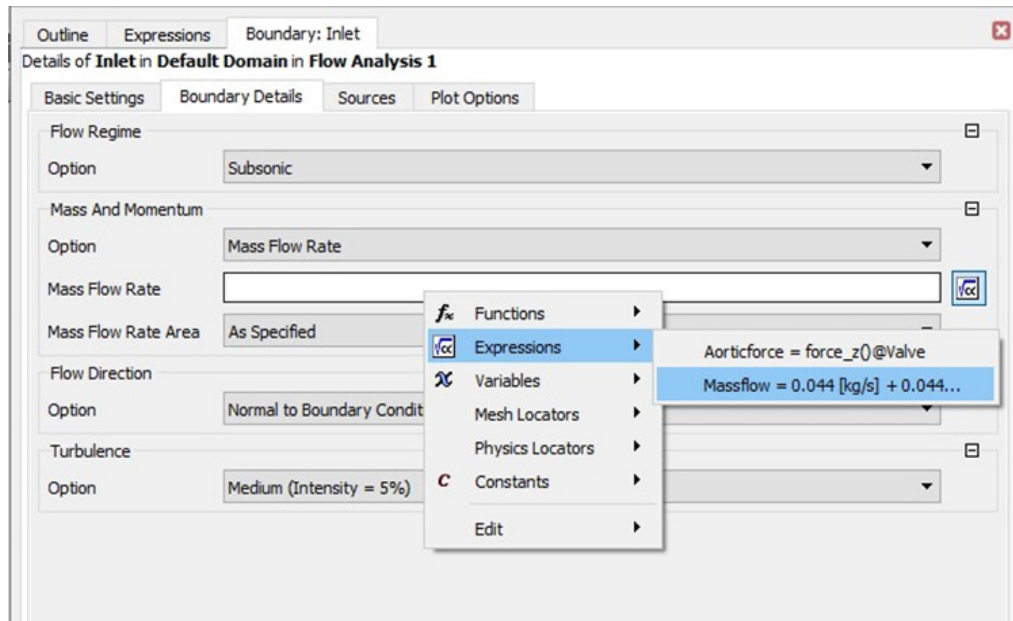
Now, enable transient results to save your data. Under 'Output Control,' click 'TRN Results' and create a new results setup with the following settings.



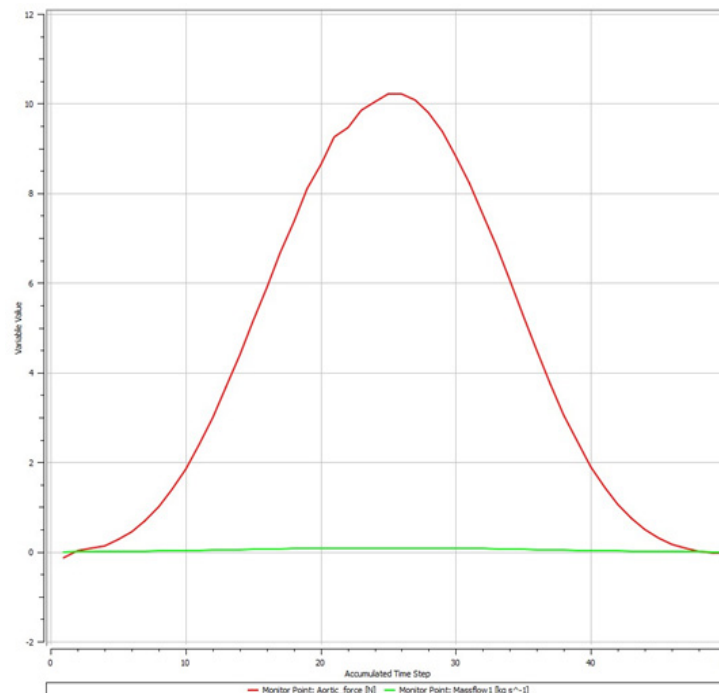
To initiate our pulsating flow, create a new expression called 'massflow' with the following expression that pulses between 0.088 and 0 every 0.5 seconds:  $0.044 + 0.044 \cdot \sin(4 \cdot \pi \cdot \text{Time} - \pi/2)$ . Set it up as follows:



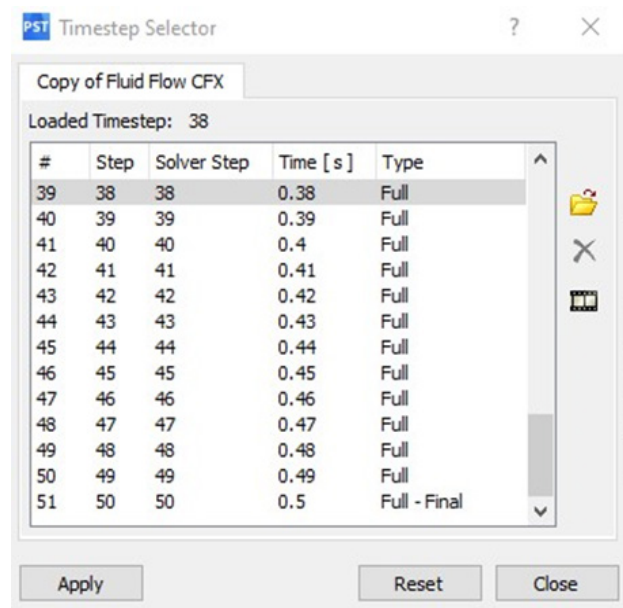
Set the massflow inlet to the expression we just created.



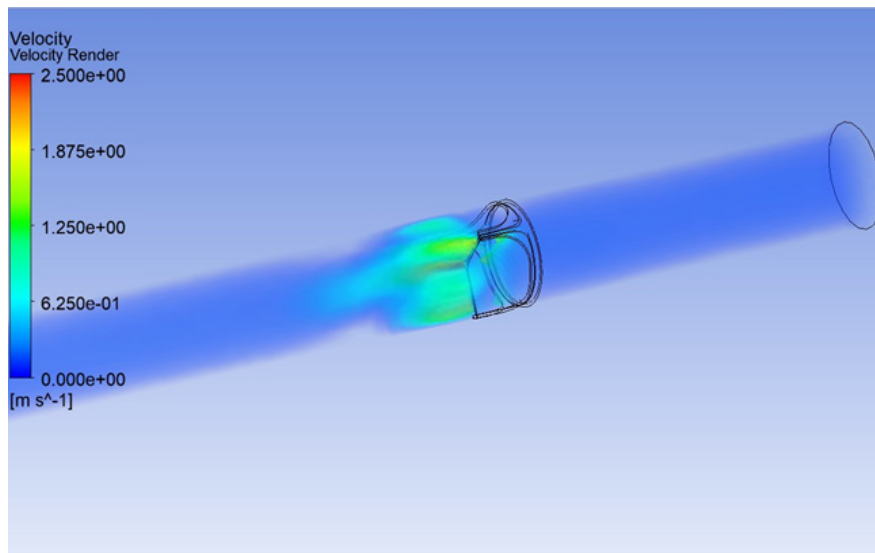
After configuring the solution, run the calculation. You should observe a force plot as follows:



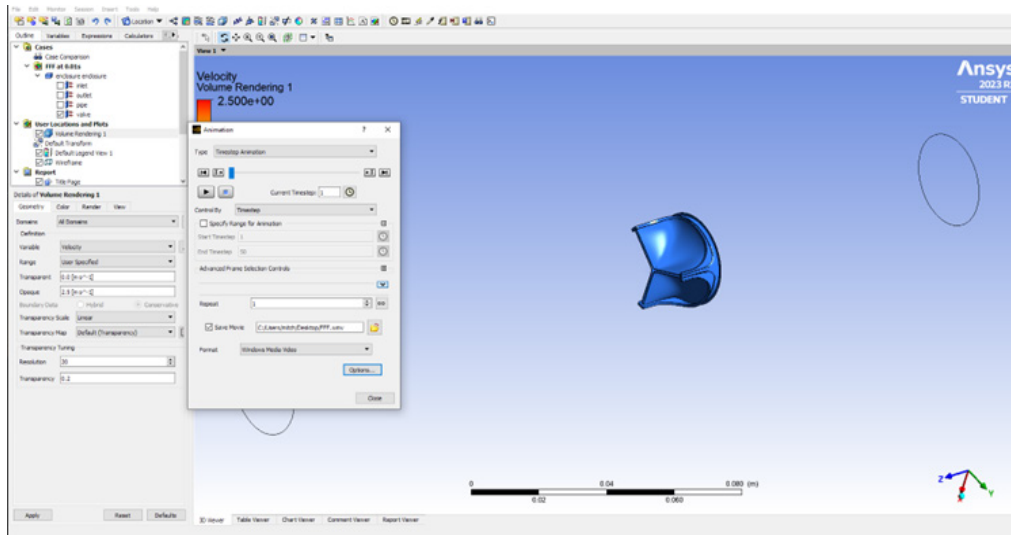
Now, switch to CFD-Post and validate that the timesteps have loaded correctly by selecting the clock icon in the top row. This will allow you to ensure that the timesteps are properly loaded and available for analysis.



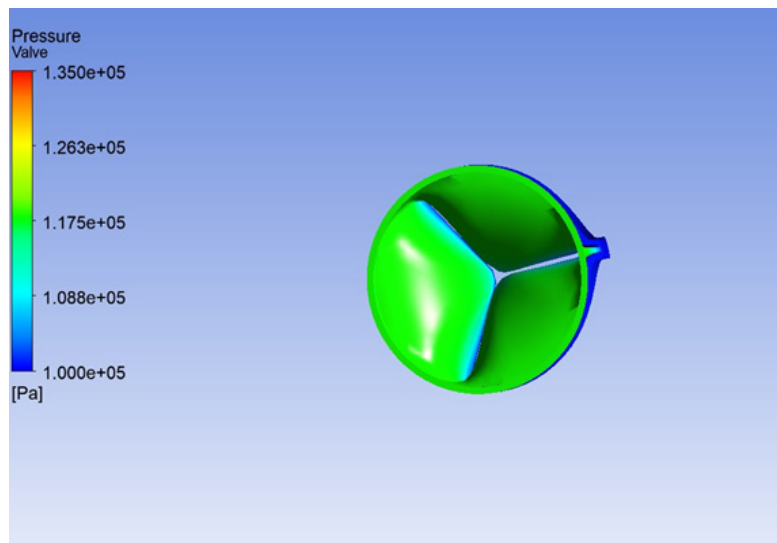
When you display any contour, vector, etc the flow will initially show us zero, we can click on one of those timesteps and select that timestep to display. Display a velocity volume render.



Let's create a timestep animation of this render. Click on the film reel again, select type timestep animation, begin from timestep 1 and press the play button. You can save by clicking on the tick box and selecting your desired file path location.



We can also look at pressure contours of the surface of the valve.



Compare the force plots are they the same, if not how do they differ?

Challenge: Attempt to reduce the mesh size, how does this influence your results.

© 2024 ANSYS, Inc. All rights reserved.

## Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

## Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

## Ansyes Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit [www.ansys.com/education-resources](http://www.ansys.com/education-resources).

## Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

**ANSYS, Inc.**  
Southpointe  
2600 Ansys Drive  
Canonsburg, PA 15317  
U.S.A.  
724.746.3304  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)

If you've ever seen a rocket launch, flown on an airplane, driven a car, used a computer, touched a mobile device, crossed a bridge or put on wearable technology, chances are you've used a product where Ansys software played a critical role in its creation. Ansys is the global leader in engineering simulation. We help the world's most innovative companies deliver radically better products to their customers. By offering the best and broadest portfolio of engineering simulation software, we help them solve the most complex design challenges and engineer products limited only by imagination.

visit [www.ansys.com](http://www.ansys.com) for more information

Any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. All other brand, product, service and feature names or trademarks are the property of their respective owners.

© 2024 ANSYS, Inc. All Rights Reserved.