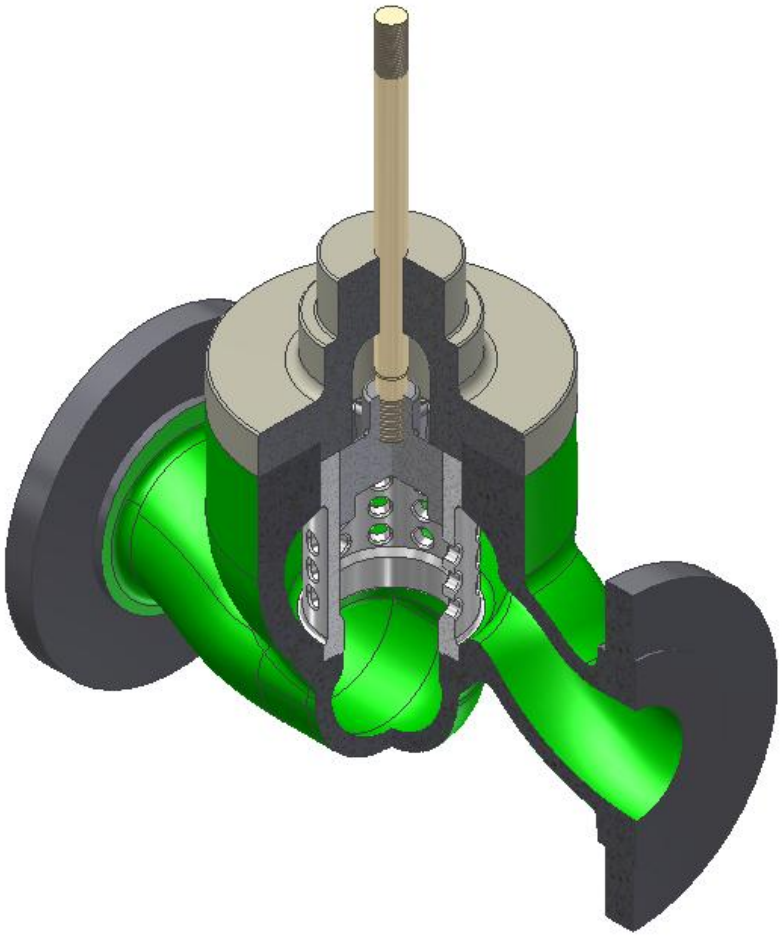
ME 24-688 – Week 14

Problem Set Assignment

1 Valve CFD Analysis Validation Project

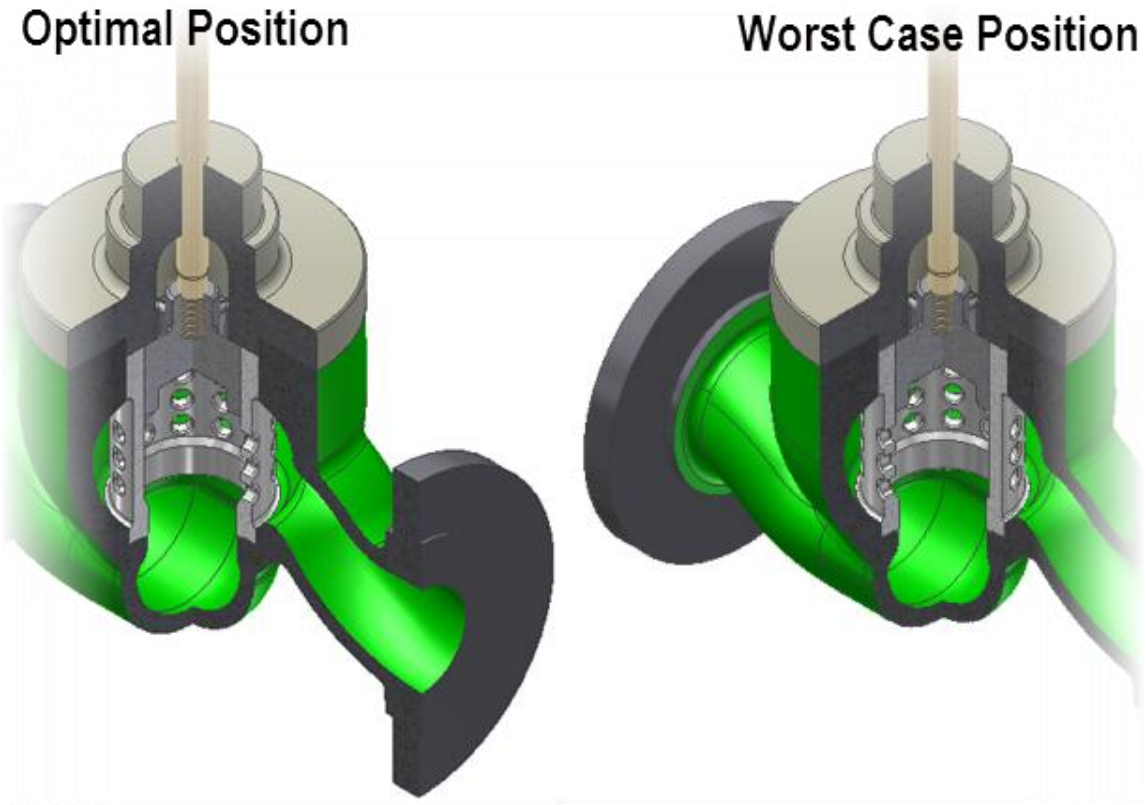
A local valve company has contracted you to analyze if impeller assembly of a valve is causing a loss in velocity in their valve. Run an Unsteady Fluid Flow analysis within Autodesk Simulation to determine the impact that improper assembly has on the outlet velocity of the valve. The goal is to determine if the valve manufacturer needs to modify the design of the valve to ensure proper alignment of the guide part with respect to the valve body. Additional information about the project assignment is listed in the following sections.



1.1 Provided Items

The following items are provided to complete the assignment.

Autodesk Inventor 2012 Assembly File
The "*Valve.iam"* file is provided as a starting point to complete the analysis. Open this provided 3D model file in Autodesk Inventor 2012. The "*GuideOrientation*" parameter within the "*Valve.iam*" will rotate the "*guide_02.ipt*" between the optimal position and the worst case position.



Fluid Material
The following material will be applied to the fluid passing through the valve.

- Water

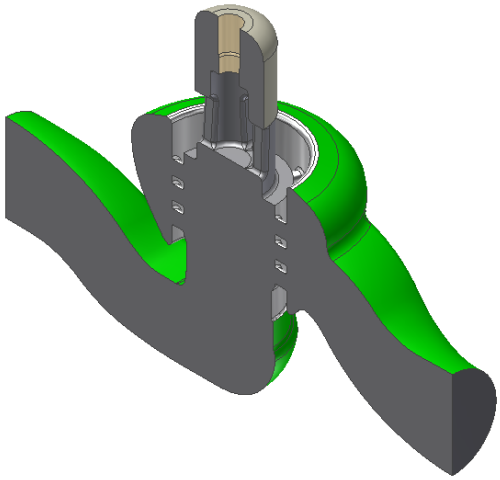
1.2 Requirements

Outlined below are the assignment requirements for the analysis. Each of the noted items is required to complete the setup for proper results.

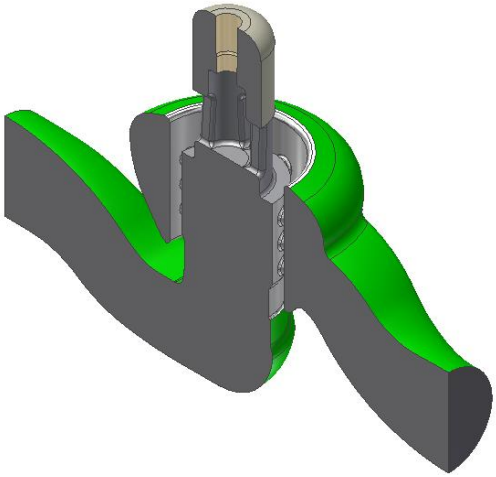
Fluid Model

Using techniques learned in the guided projects model the fluid that will pass through the valve in both the optimal position and the worst case position. To simplify the model and reduce processing time split the model as shown and apply symmetry.

Optimal Position

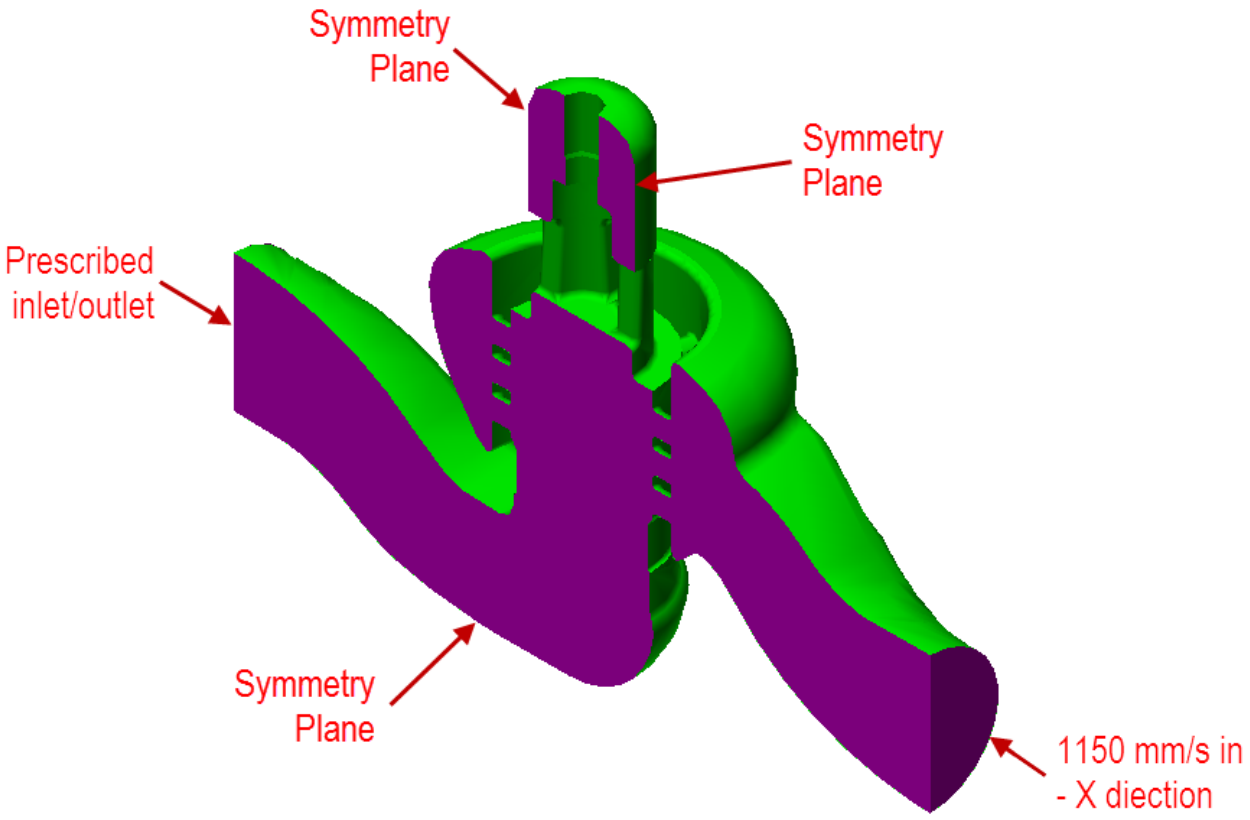


Worst Case Position



Setup

All symmetry, inlet/outlet and prescribed velocity surfaces need to be excluded from the boundary layer.



1.3 Instructions

Outlined below is a series of different instructions and guides for completing the project.

- Create a step file for fluid in both the Optimal and worst case position.
- The mesh size should be set to an absolute value of 2.5 mm.
- Set the Solid mesh to Terahedra and wedges (boundary layer)
- Analysis total time duration is 1.0 seconds.
- Apply the following load curve:

Time	Multiplier	Steps
0.0	0.0	1
1.0	1.0	20

- Each run of the analysis will take approximately 15-20 minutes to solve depending on your computer resources.
- The inlet of the valve needs a prescribed velocity applied of 1150 mm/s in the – x direction.
- Generate a step file of the fluid in both the optimal position and the worst case position.
- Run analysis for both the optimal position and the worst case position.

Tips

- Watch the first couple time steps as the analysis solves to determine if everything looks setup correctly before continue. If you happen to notice an issue you can stop the analysis and return to the FEA Editor environment to make the required updates.

1.4 Final Deliverable

To complete the problem set provide a completed Autodesk Simulation report in PDF format with your recommendations on if a design changes is warranted to prevent a loss of velocity in this valve in the executive summary section. Also please include your name in the author field of the PDF report files.

Report 1
Simulation completed using the *Optimal Position* including the following:

- A graph showing the velocity at the outlet.
- A Contour presentation of the Maximum Velocity
- A Contour presentation of the Maximum Pressure.
- A Streamline flow visualization at step 1 of the analysis with a least 80 nodes from the inlet
- A Particle path flow visualization at step 7 of the analysis with a least 80 nodes from the inlet selected and the number of particles to introduce set to 80

In addition to the report the step file used for the analysis also need to be submitted.

Report 2
Simulation completed using the *Worst Case Position* including the following:

- A graph showing the velocity at the outlet.
- A Contour presentation of the Maximum Velocity
- A Contour presentation of the Maximum Pressure.
- A Streamline flow visualization at step 1 of the analysis with a least 80 nodes from the inlet
- A Particle path flow visualization at step 7 of the analysis with a least 80 nodes from the inlet selected and the number of particles to introduce set to 80

In addition to the report the step file used for the analysis also need to be submitted.

1.5 Grading

The problem set grading level will be established from the following items:

- 40% = Correct Report 1 Values
- 40% = Correct Report 2 Values
- 20% = Provided both report files in PDF format