

25-650: Applied FEA

Assignment 3

Ryan Nagle

Due: 2/21/2024

Overall Objective

The goal was to analyze a valve loaded with an internal cyclic pressure. The location and value of the peak stress must be accurately determined to ensure the structural integrity of the valve and avoid fatigue failure.

Assumptions

For all situations below, it is assumed that the bottom face of the valve is constrained in the normal direction (frictionless support). The pressure load is assumed to be uniformly distributed and constant. End effects at the pipe ends and stress risers near the frictionless support are to be ignored.

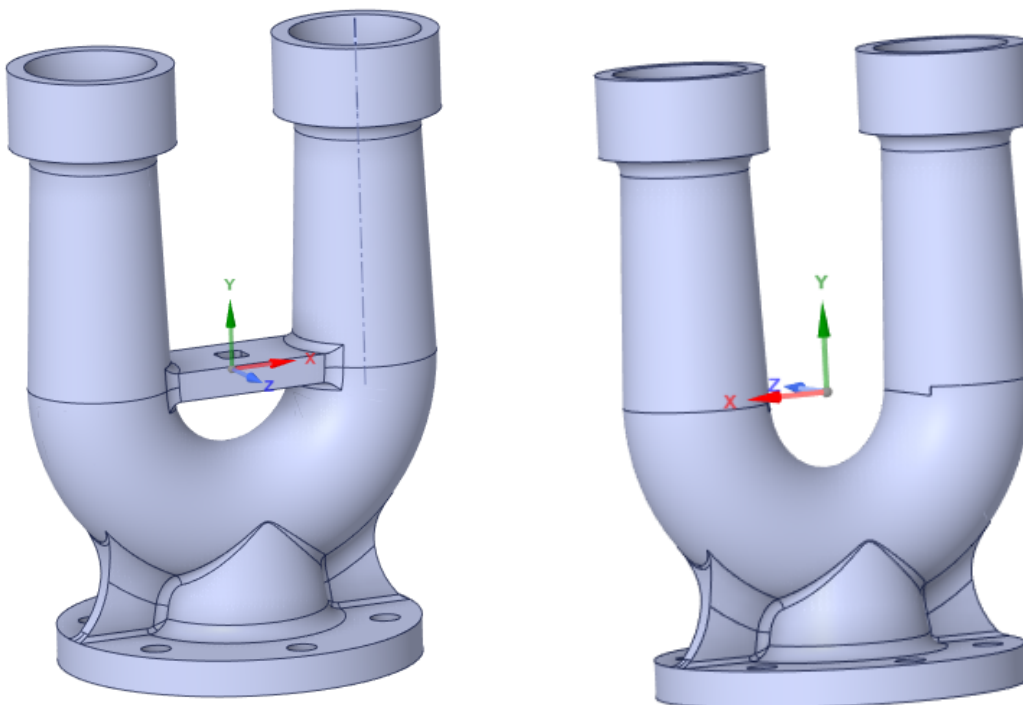
Geometry

For Part A (left), the valve was created in SpaceClaim with these dimensions:

- Height (Y): 425mm; Depth (Z): 250mm; Width (X): 315mm
- Pipe of varying internal diameter: 62mm-90mm

For Part B (right), the brace was removed from the valve

The resulting geometries are shown below:



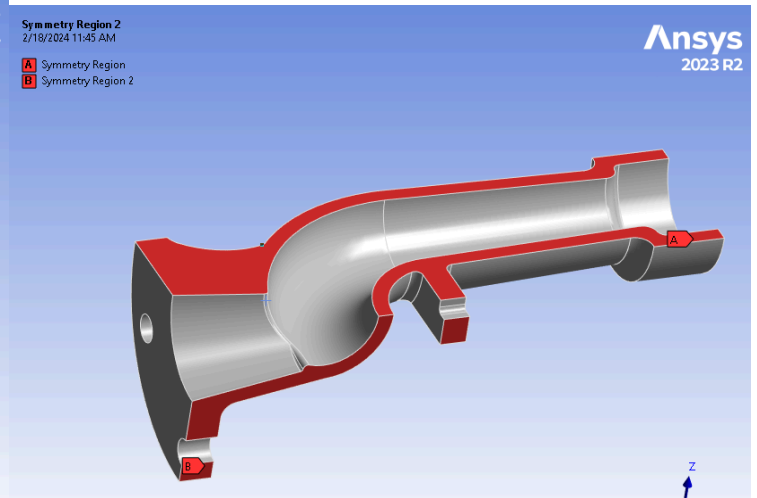
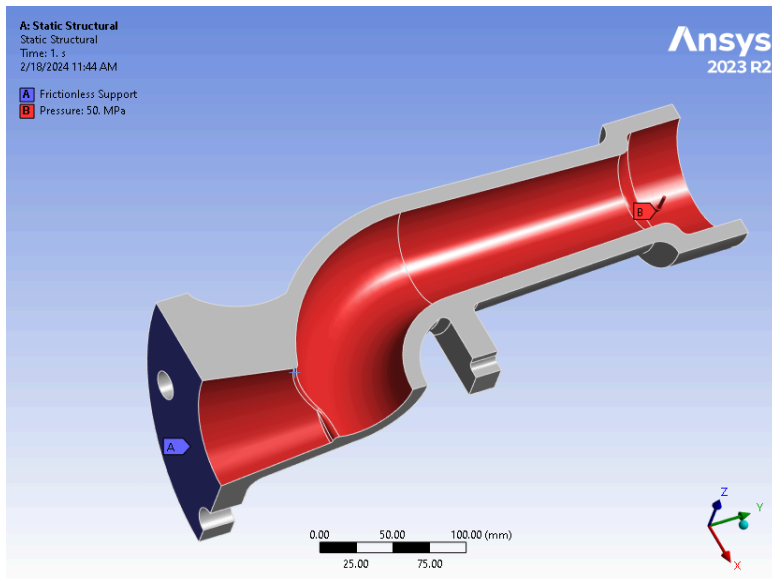
Material Data

The valve is made of structural steel (default material) from the engineering data within Ansys Workbench.

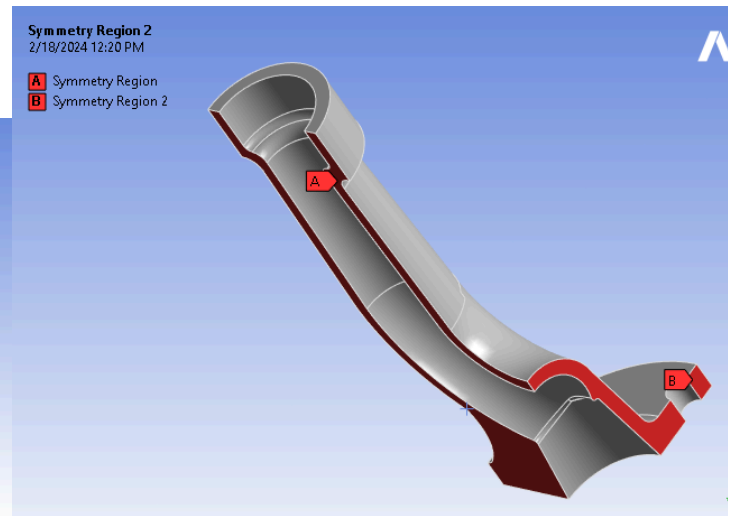
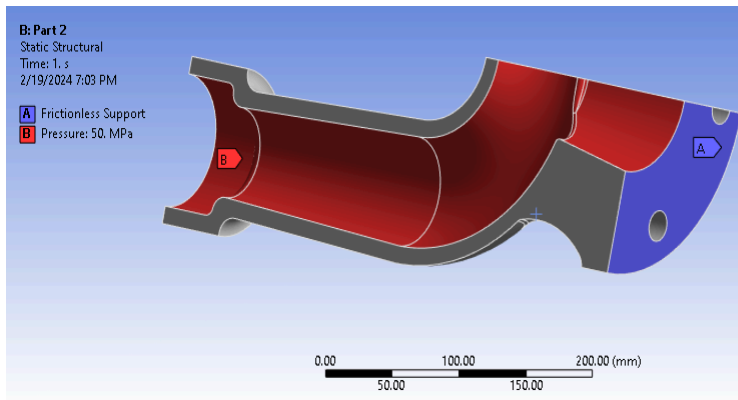
Boundary Conditions

For both Part A (shown below) and B, $\frac{1}{4}$ symmetry model was used:

- Frictionless support along bottom face of the valve
- Internal pressure of 50MPa along inner faces of the valve
- Symmetry along the X-Y and Y-Z planes



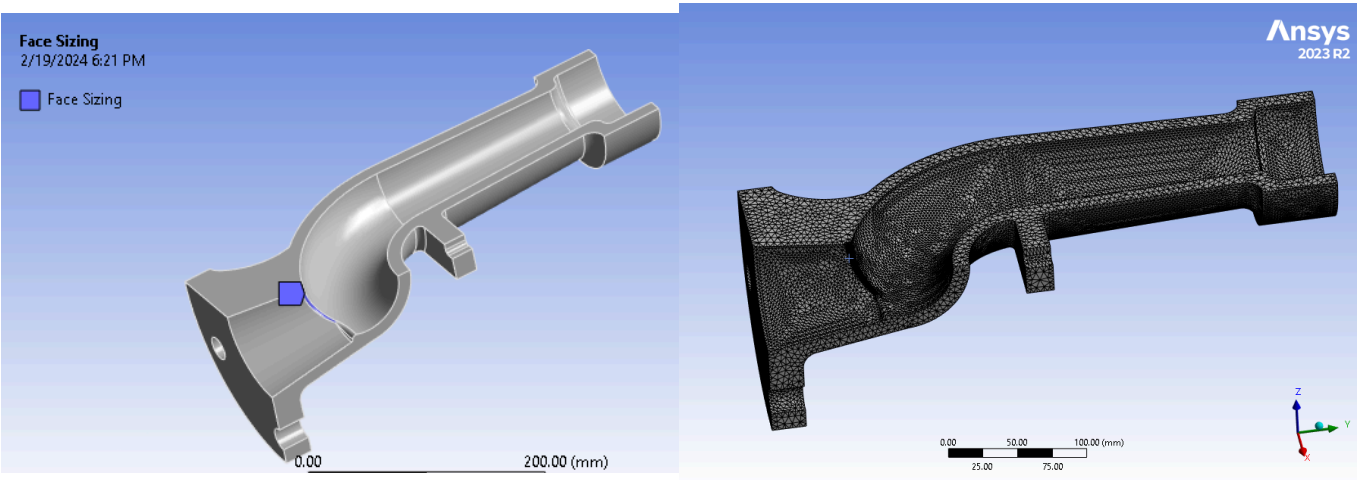
Part B (shown below):



Mesh and Solution Setup

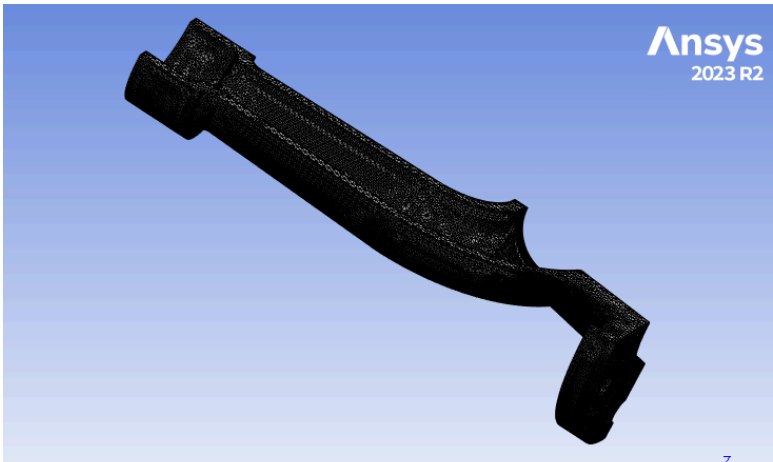
For Part A, three meshes were generated for Part A with element sizes of 10mm with 1mm along face (shown below), 5mm with 1mm, and 2.5mm with 0.5mm. There were small observed differences between results as shown below, thereby indicating that the mesh is converged. 2.5mm with 0.5mm shown below.

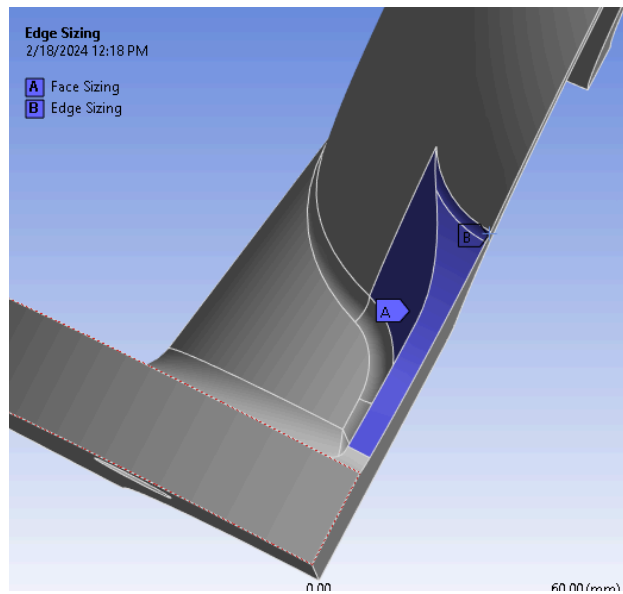
	<input checked="" type="checkbox"/> Nodes	<input checked="" type="checkbox"/> Elements	Equivalent Stress (MPa)	
			<input type="checkbox"/> Minimum	<input type="checkbox"/> Maximum
Solution 2: 02/18/2024 11:33 AM	15,700	10,254	2.0044	637.82
Solution 3: 02/18/2024 11:35 AM	57,593	38,522	2.137	638.07
Solution 4: 02/18/2024 11:36 AM	201,506	133,111	1.3314	638.21



For Part B, two meshes were generated for Part A with element sizes of 5mm with 1mm along faces and 0.25mm along the edge (shown below) and the other with element sizes of 2.5mm with 0.5mm along faces and 0.1mm along the edge. There were small observed differences between results as shown below, thereby indicating that the mesh is converged. 2.5mm with 0.5mm faces and 0.1mm edges shown below.

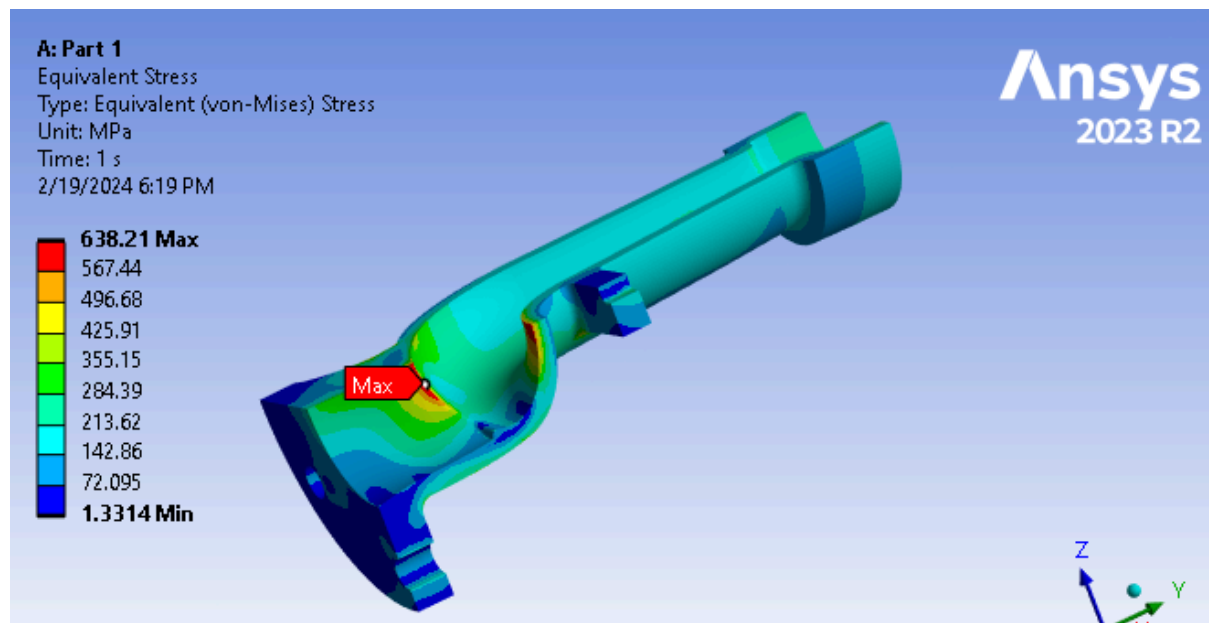
	<input checked="" type="checkbox"/> Nodes	<input checked="" type="checkbox"/> Elements	Equivalent Stress (MPa)	
			<input type="checkbox"/> Minimum	<input type="checkbox"/> Maximum
Solution 1: 02/18/2024 12:17 PM	338,397	217,776	1.3266	711.61
Solution 2: 02/19/2024 07:00 PM	55,657	35,522	2.4886	708.87





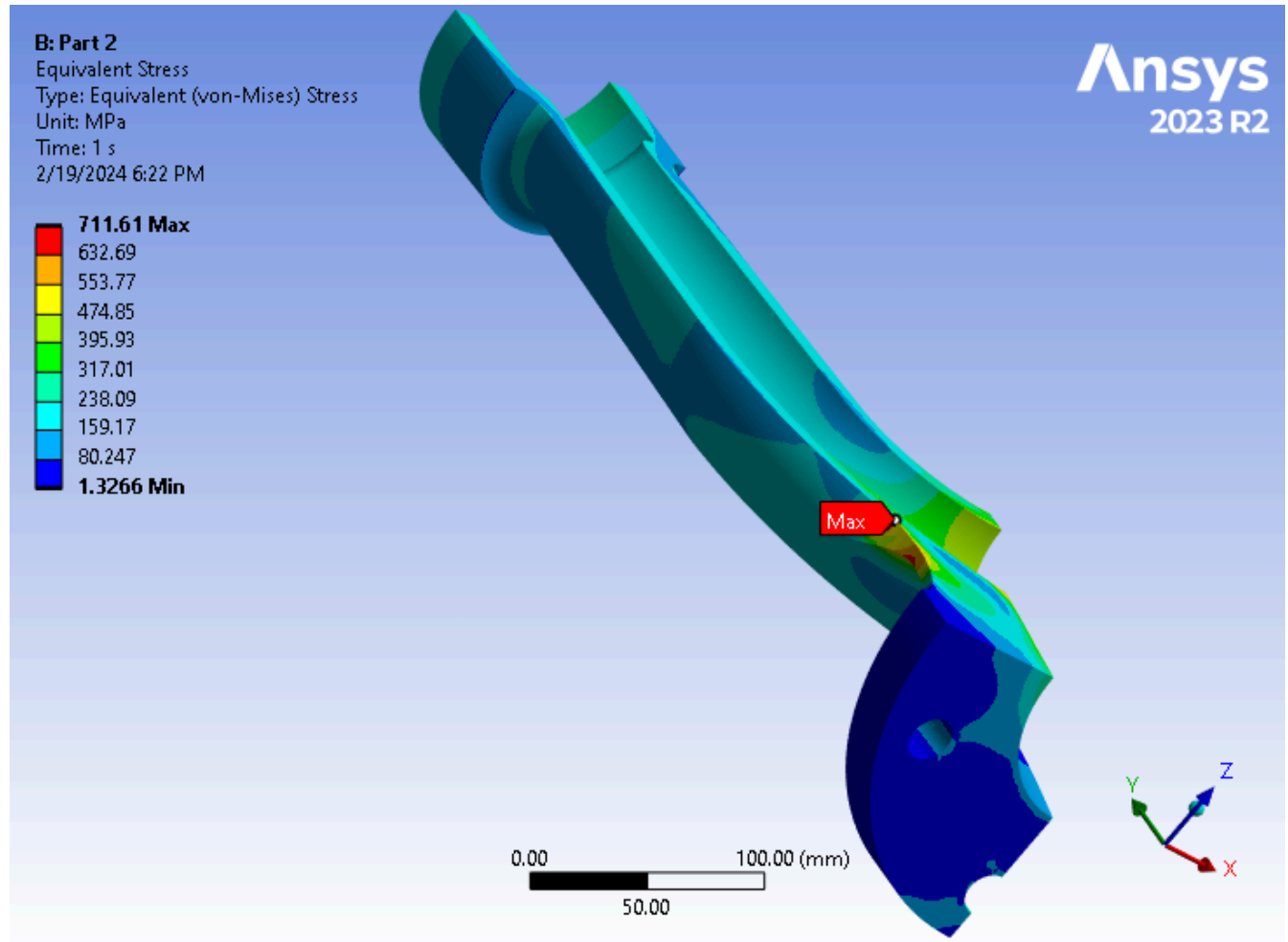
Results

Part A:



The peak stress was located as shown with a value of 638.2 MPa.

Part B:



The peak stress was located as shown with a value of 711.6 MPa.

Conclusion

Overall, the results of the simulations make sense given the conditions used. It makes sense that the max stress would be on a relatively sharp corner in part A. It also makes sense that once the brace was removed that the max stress would be higher and on the lower outside of the pipe because there is less material to resist outward expansion.