Maryam Hakimzadeh **Due: 2/21/2024**

Overall Objective

The objective is to accurately identify the peak stress location and magnitude within the valve body.

Assumptions

Thermal effects are disregarded in this analysis.

The analysis assumes linear elasticity.

A quarter-symmetry model has been utilized for all analyses within this report.

End effects at the pipe ends are not considered.

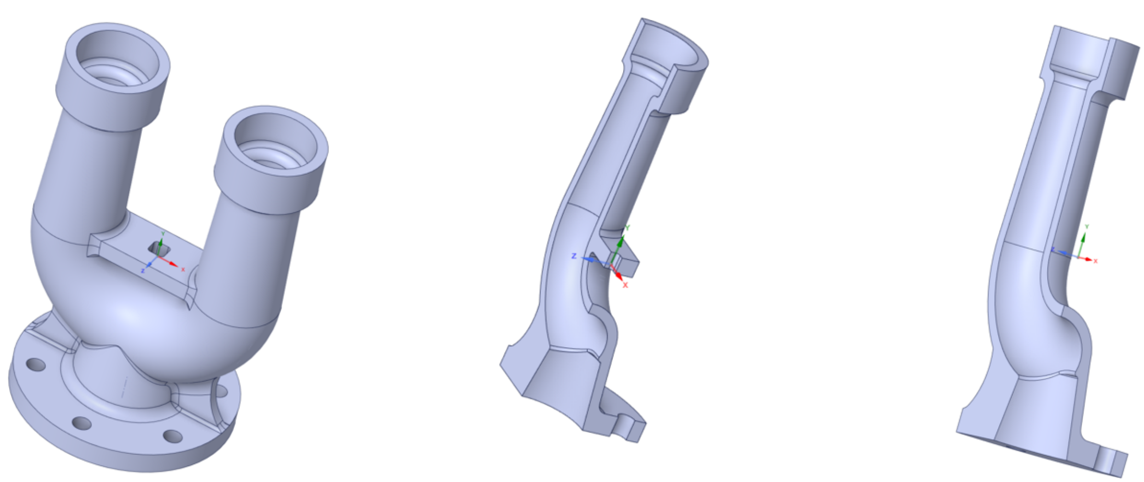
Stress risers on and near the frictionless support are overlooked.

The choice of frictionless support implies that the valve's bottom is free to slide along the bottom face, an idealization not reflective of real-world conditions where attachment to another part introduces resistance.

Geometry

The geometry of the valve, and the symmetry used in part A and part B of the analysis (explained later) is given as below:

Total body | ¼ symmetry body | ¼ symmetry body with brace removed



Material Data

The valve is constructed from structural steel, as specified in the Engineering Data tool of Ansys.

Boundary Conditions

The valve is subjected to an internal pressure of 50 MPa.

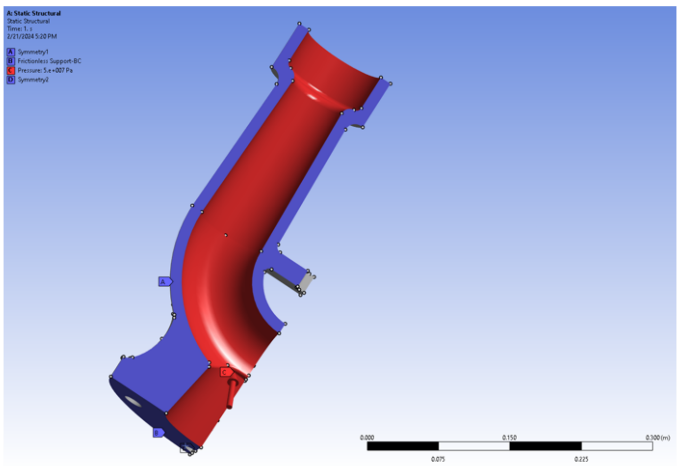
A frictionless support is implemented on the bottom surface.

In the analysis for part A, brace support is included between the pipes.

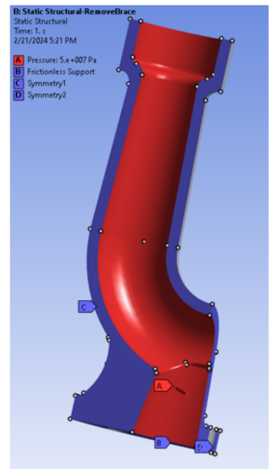
The analysis for part B mirrors that of part A, with the sole difference being the removal of the brace support between the pipes.

The accompanying figure illustrates the boundary conditions for both part A and part B.

Part A boundary conditions



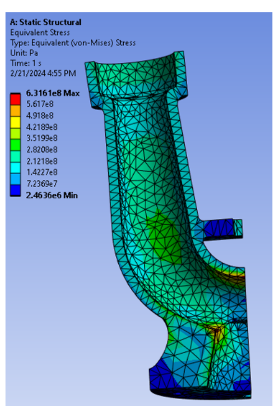
Part B boundary conditions



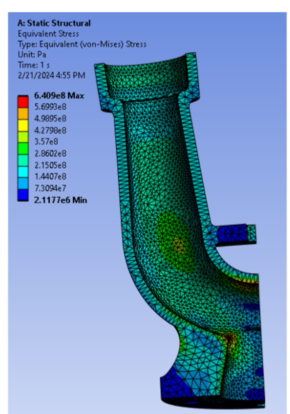
Mesh Convergence Study

A mesh convergence study was performed to ensure the mesh is sufficiently refined for accurate convergence.

Mesh size = 0.01 m



Mesh size = 0.005 m



Given that the difference in maximum stress exceeds 1 MPa, the analysis proceeded with further mesh refinement in areas exhibiting higher stress. The table below summarizes all analyses conducted:

Mesh size(mm)	10	5	2.5	1.25
Maximum Stress (MPa)	631.6	640.9	638.53	637.74

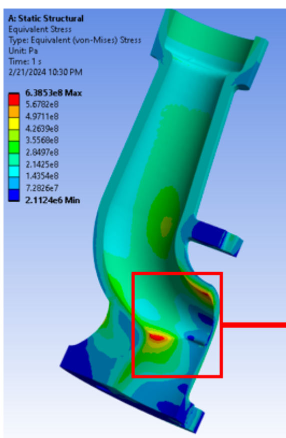
By comparing the maximum stress values between a mesh size of 2.5 mm and a refined mesh size of 1.25 mm, we observe that the difference in maximum stress is less than 1 MPa. Hence, we can conclude that the results have converged at a mesh size of 2.5 mm. This mesh size has been adopted for the remainder of the analysis for part A.

Results

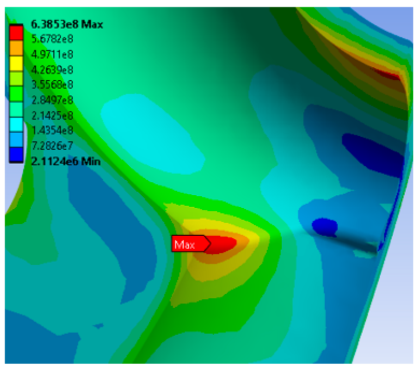
Part A:

Utilizing a mesh size of 2.5 mm, as determined from the mesh convergence study, the maximum stress value and the location of the maximum stress are as follows:

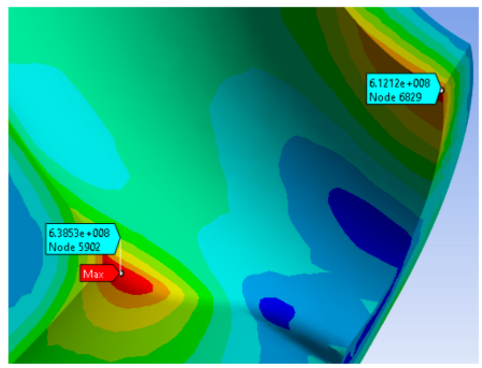
Plot of equivalent stress



The location with maximum equivalent stress



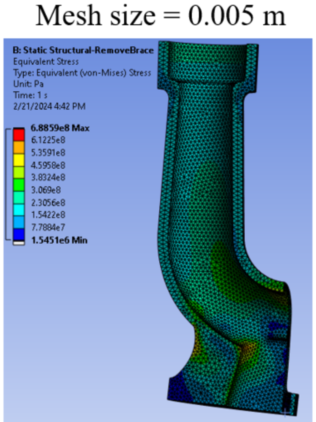
Locations of highest stress points, and values of stress on those points



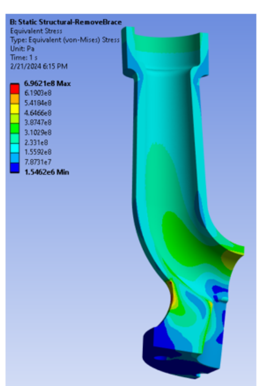
Part B:

With the brace removed, the analysis is conducted to assess the impact on the system.

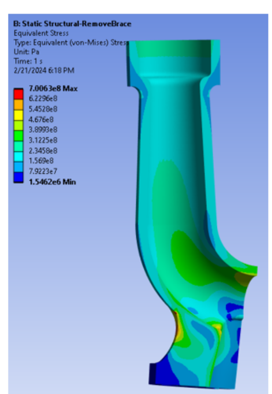
Mesh size = 0.005 m



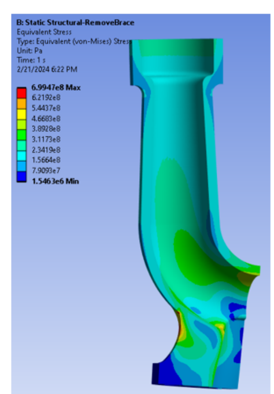
Refinement 1



Refinement 2



Refinement 3



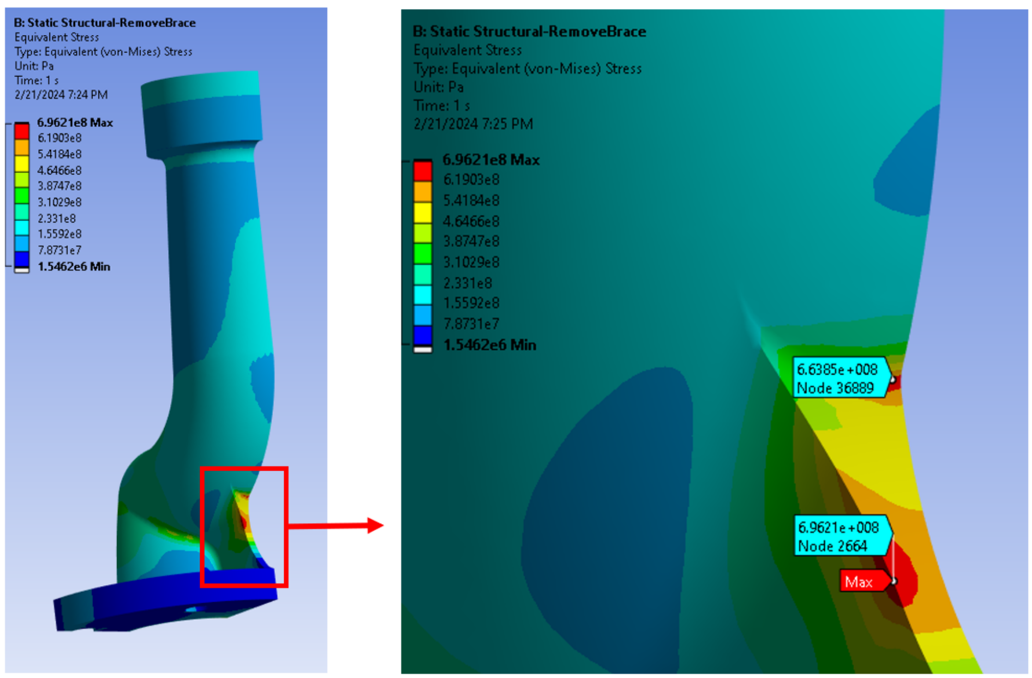
The table below summarizes the results of the mesh convergence study:

Mesh size (mm)	5	2 .5	1.25	0.625
Maximum Stress (MPa)	688.59	696.21	700.63	699.47

By using the mesh size = 5 mm with refinement 1 (2.5 mm in the refined regions):

Plot of equivalent stress

Locations of highest stress points, and values of stress on those points



After the removal of the brace, there was an increase in the maximum stress value from 639 MPa to 696 MPa. Moreover, the location of the maximum stress has changed.

Upon reviewing the material properties of structural steel, it is noted that the tensile yield strength is 460 MPa. Consequently, the material has undergone yielding in both Part A and Part B of the analysis. Considering the cyclic loading conditions of the valve, it faces a significantly high risk of fatigue and eventual failure.

Furthermore, given that the analysis was performed under the assumption of linear elasticity, the reported maximum stress values may not accurately reflect the material's response. Therefore, an elastoplastic analysis is recommended for obtaining more precise results.

Conclusion

This report aimed to precisely determine the peak stress location and magnitude within a valve body, under specific assumptions and conditions. Through meticulous analysis, including mesh convergence studies and the application of linear elasticity principles, significant insights were obtained regarding the valve's structural integrity when subjected to internal pressures.

The findings reveal that removing the brace support between the pipes (transitioning from Part A to Part B) resulted in an increase in maximum stress from 639 MPa to 696 MPa, surpassing the tensile yield strength of the structural steel material (460 MPa) in both Part A and Part B. This indicates that the material has yielded under the given conditions in both parts of the analysis, highlighting a substantial risk of fatigue and potential failure due to cyclic loading.

Moreover, the analysis, based on linear elasticity, suggests that the obtained stress values might not fully capture the material's response under such conditions. This underscores the necessity for an elastoplastic analysis to achieve a more accurate representation of the valve's behavior under stress.

In conclusion, while the initial objective of identifying peak stress locations and magnitudes was achieved, the outcomes signal a critical need for further investigation into the valve's design and material selection. Considering the valve's high risk of fatigue and failure, it is imperative to explore alternative designs or materials that can withstand the identified stresses more effectively. Additionally, conducting an elastoplastic analysis is recommended to refine the understanding of the valve's structural performance, ensuring its reliability and safety in practical applications.