DESIGN AND ANALYSIS OF A HYDRAULIC VALVE

Jackson Bryce Herb, Engineering Studies Program

Faculty Sponsor: Dr. Barry Hojjatie, Department of Physics, Astronomy, Geosciences and Engineering

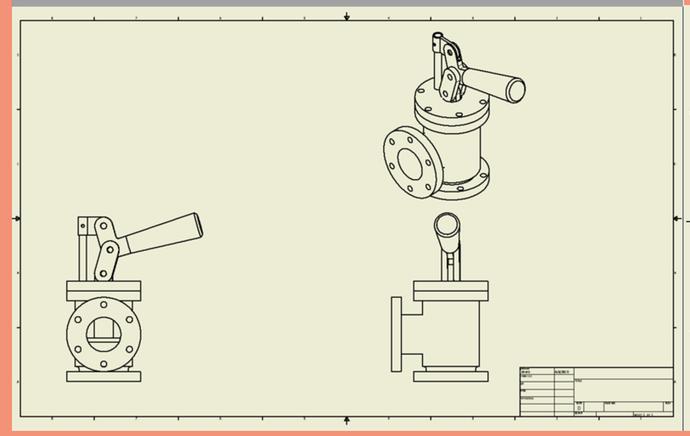
Abstract

This study focuses on the design process and analysis of stress testing on a hydraulic valve. Using Autodesk Inventor, a valve was designed to regulate the flow of fluids such as gasses or liquids with the use of mechanical levers and springs. While designing this valve, constraints and regulations were provided, along with the purpose of the machine, in order to ensure that the valve can withstand the stresses it experiences during its continuous use. Constraints included size, material cost, and the valve must have had the ability to direct and block the flow of fluids. A stress analysis and parametric analysis is used on the valve. The stress analysis is to see where stress will be applied to the valve and how well the materials that were chosen perform. The parametric analysis shows how the valve reacts to different tasks and conditions, such as, different fluid types, temperatures, and pressures.

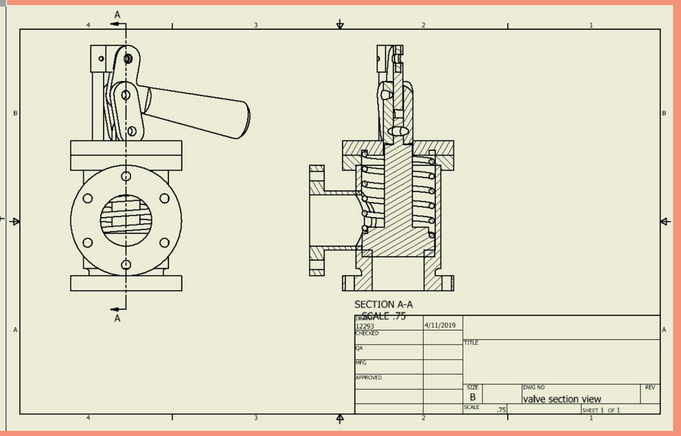
Objectives

- The main objective of this project is to use finite element analysis to test the performance of a hydraulic valve.
- The valve needs to be able to withstand fluid pressure without failing in order to complete its task. Failing the test includes buckling, breaking, bending, or being forced open by the pressure of the fluids.
- The stresses applied to the valve and how the materials used in the valve react to these loads are shown using ANSYS Workbench.

A 3D model of the valve on Autodesk Inventor along with a 2D layout drawing.



Section view of the valve shown in a 2D drawing.



Finite Element Analysis (FEA)

Finite element analysis is a tool used to break up a model or structure into many small parts called elements. Using ANSYS Workbench we can import the geometry of the valve, and the program will break it up into many finite elements. Each of these elements will have a force applied to them and will react based on their shape, surroundings, and the materials they consist of.

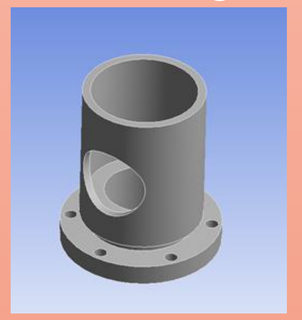
Stress Analysis Overview

- The main body of the valve along with the valve stem were analyzed in the stress tests. Theses parts were chosen because the fluid pressures are going to be acting on them more than any other part. This is so the reactions of the material and structure of the parts could be observed in the most critical areas.

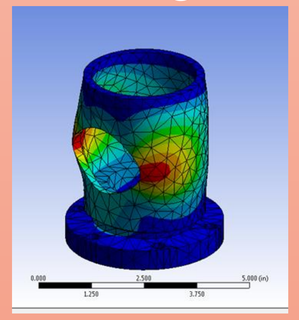
Testing

- When testing the main body of the valve with no support around the opening the pressure from the inside caused the steel body to buckle.
- When the side opening was added to the main body it provided enough support to prevent it from buckling under these pressures
- Under the stress tests the stem of the valve could withstand many times more pressure than it would experience during its use without bending, breaking, or buckling.

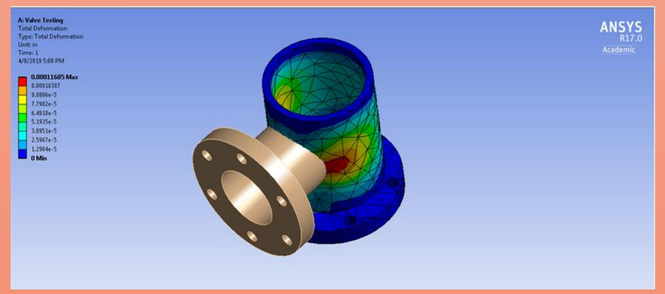
Before Testing



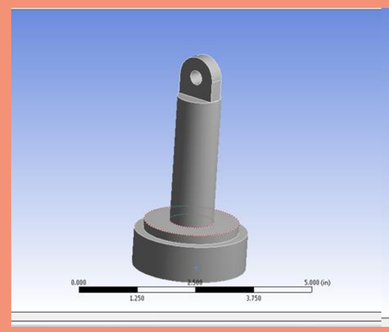
After Testing



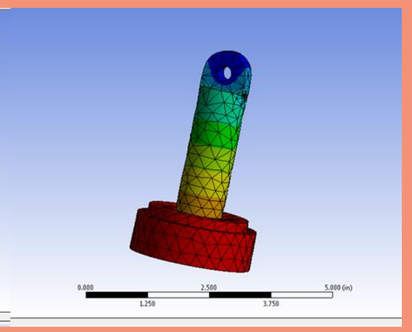
After the side opening was added



Before Testing



After Testing



Conclusion

Testing with ANSYS Workbench has revealed that the valve design is able to complete the tasks assigned to it with little effort. Pressures were applied to the main body of the valve and the valve stem. After testing these parts against the fluid pressure the material and structural reactions were observed. The main body would not retain its shape very well, under high loads, without extra support. The valve stem however was able to withstand the forces with no breaking, bending, or buckling because its cylindrical shape allows it to hold its form even under heavy loads from the top and bottom. The parts were not the only things analyzed in the tests; the material chosen for the valve, stainless steel, was also being tested for durability. The stainless steel preformed well and helped the parts retain their shape.