Summary of Findings for Thermal Stress Analysis

Introduction and Materials:

The goal of this project was to evaluate thermal strain and stress across several different materials typically used in heat transfer applications. The analysis was inspired by a theoretical nuclear steam supply system (NSSS). The materials used were:

- Inconel 690 – An alloy meant for corrosive and high temperature environments. Commonly used in nuclear applications.
- SS304 – An industry standard stainless steel. While highly durable for most applications, its lower yield strength at high temperatures would not be ideal for extreme thermal environments. It was included as a reference.
- P91 Steel Alloy – An alloy steel specifically designed to deal with high temperature environments while retaining strength. Commonly used in power generation.

Assumptions and Simplifications:

To simplify the analysis, several assumptions were made regarding material properties, geometry, and boundary conditions. Generally, the model was assumed to be steady state and have constant material properties, i.e. Young's Modulus and yield strength were assumed constant across the temperature range and the value used was the typical value associated with the secondary side environment. In Python, it was assumed that the material would be 100% constrained and that the heat transfer was only one dimensional. The model also did not account for localized stresses such as those caused by bolt holes. In ANSYS Mechanical, the pipe was constrained via bolt holes to simulate how it might be mounted in a real system.

Findings:

After running the Python code and the simulation in ANSYS Mechanical, the models produced similar results which validated the code. Exhibit 1 shows the stress of each material at varying temperature differentials. For Inconel 690 at $\Delta T = 50°C$, the approximate stress was 130 MPa. The ANSYS model (Exhibit 2) shows the piping part made of Inconel 690 with stresses around 130-180 MPa and peak values of about 240 MPa. These peak values are most likely attributable to stress concentrations around the edges of the bolt holes, or possibly mesh artifacts. Overall, both Python and ANSYS values are the same order of magnitude and Exhibits 3 and 4 also show close alignment between Python and ANSYS results in terms of strain, lending further confidence to the model validity.

Future Updates:

Future updates to this project could include parameterizing material properties in Python to account for the variation of temperatures. Additionally, more models could be run in ANSYS with the other materials and parameterized temperature differentials. In this iteration, Inconel 690 was the only material modeled to verify the baseline simulation.

Exhibit 1:

Graph of thermal stress vs. the temperature differential for three different materials generated from Python. Yield strengths shown in dashed lines for each material.

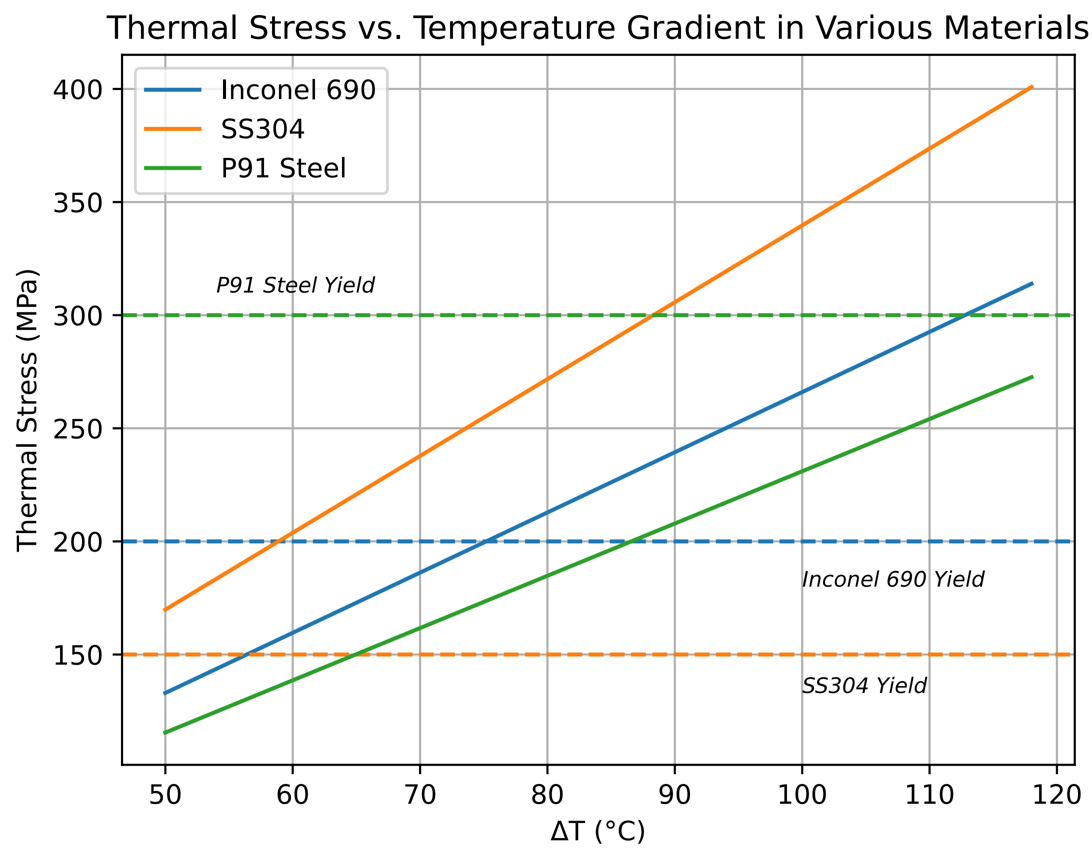

Exhibit 2:

ANSYS model of pipe made of Inconel 690 with a 50 degree temperature difference between the inside and outside. The bolt holes are fixed.

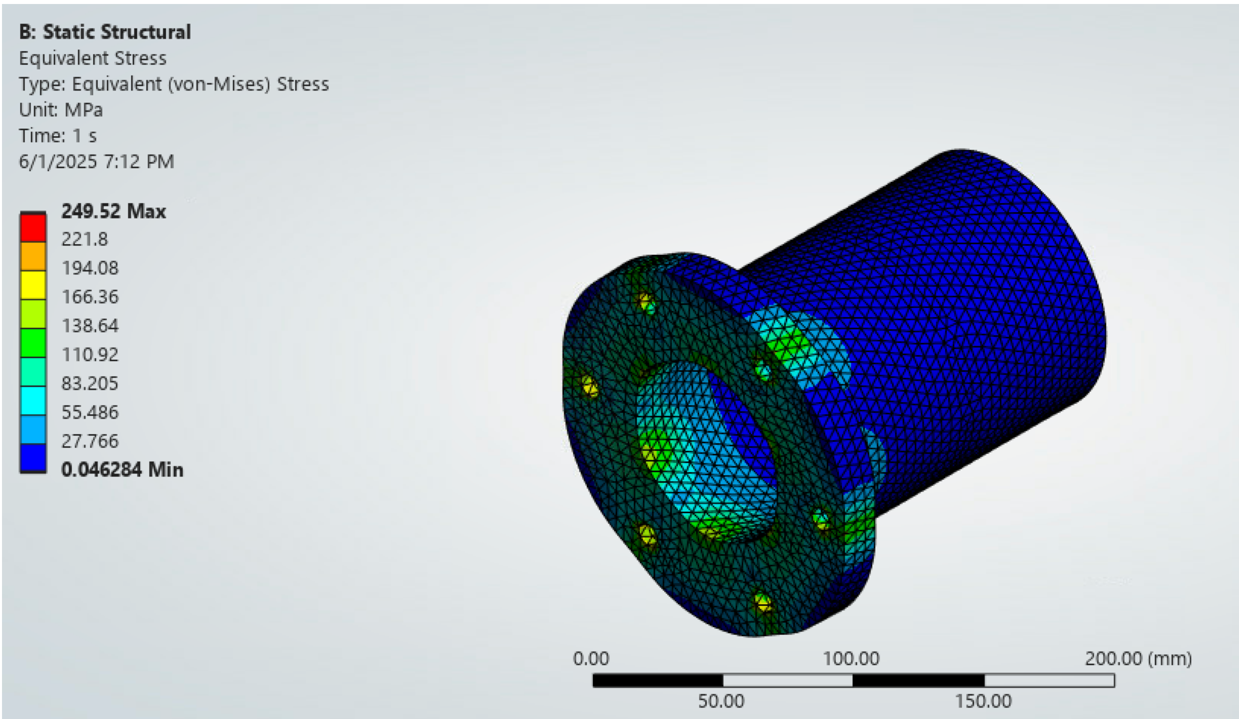

Exhibit 3:

Thermal strain vs. Temperature differential in the three different materials. Results verified by ANSYS model in Exhibit 4.

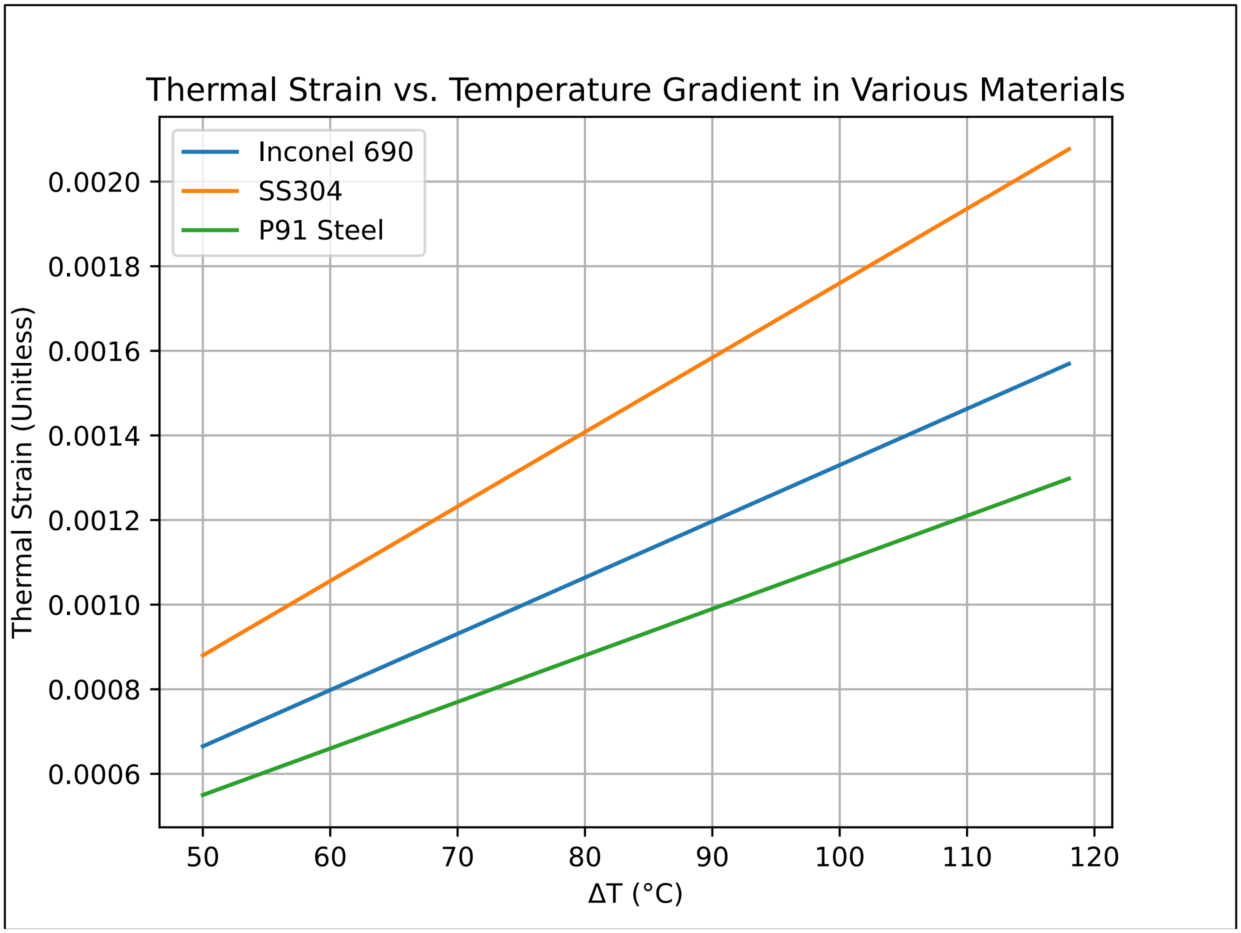

Exhibit 4:

Strain on pipe made of Inconel 690. Results align with graph in Exhibit 3.