MIXING TEE ANALYSIS WITH THERMAL CFD

A Thermal CFD simulation utilizing ANSYS Fluent

Orientation: **The goal of this project was to show capability in ANSYS Fluent with a coupled CFD and Thermal problem. The object to analyze is a simple T-shaped pipe joint. With a hot and cold fluid coming into the system while we observe velocity, pressure and temperature data on the output side of the system.**

The Analysis

The Model

The part was modeled in Solidworks and imported into ANSYS via IGES file. It consists of a pipe tee with a 3 inch main tube with an auxiliary 2 inch 90 degree input as shown.

Note that this is the fluid domain the actual part has been subtracted from the model, as ANSYS does not need it to perform a CFD analysis.

No fillets were implemented at the seams for simplicity.

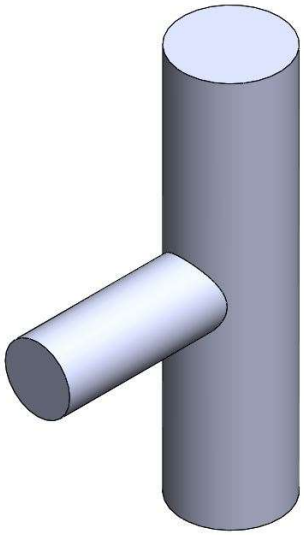

Figure 1 Solidworks Model

Parameters and Boundary Conditions

The 2 inch inlet and bottom 3 inch inlet are defined as velocity inlets. With the smaller inlet having an input velocity defined at 6 in/sec with a temperature of 559.67 R (100 F) and the larger inlet having a 3 in/sec velocity and a temperature of 491.67 R (32 F). The remaining outlet has a boundary condition defined as a 0 gage pressure outlet. This is to prevent back flow and simulate an uninfluenced input flow.

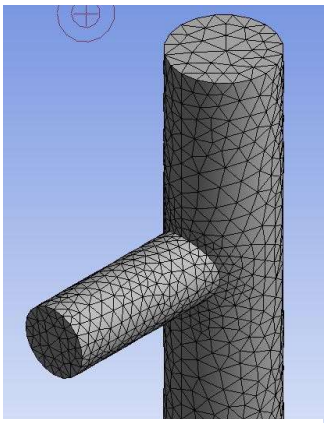

Figure 2 Mesh completed in ANSYS Mechanical

Mesh

A curvature based mesh was implemented due to the cylindrical nature of the model. A curvature based mesh works well for these applications because the algorithm moves to equally space triangular elements radially, while maximizing the angle between them. This in theory will contribute to a better overall quality mesh because it will ensure the most elements have an aspect ratio closer to 1.

Sensors

We will monitor the outlet for temperature.

Results

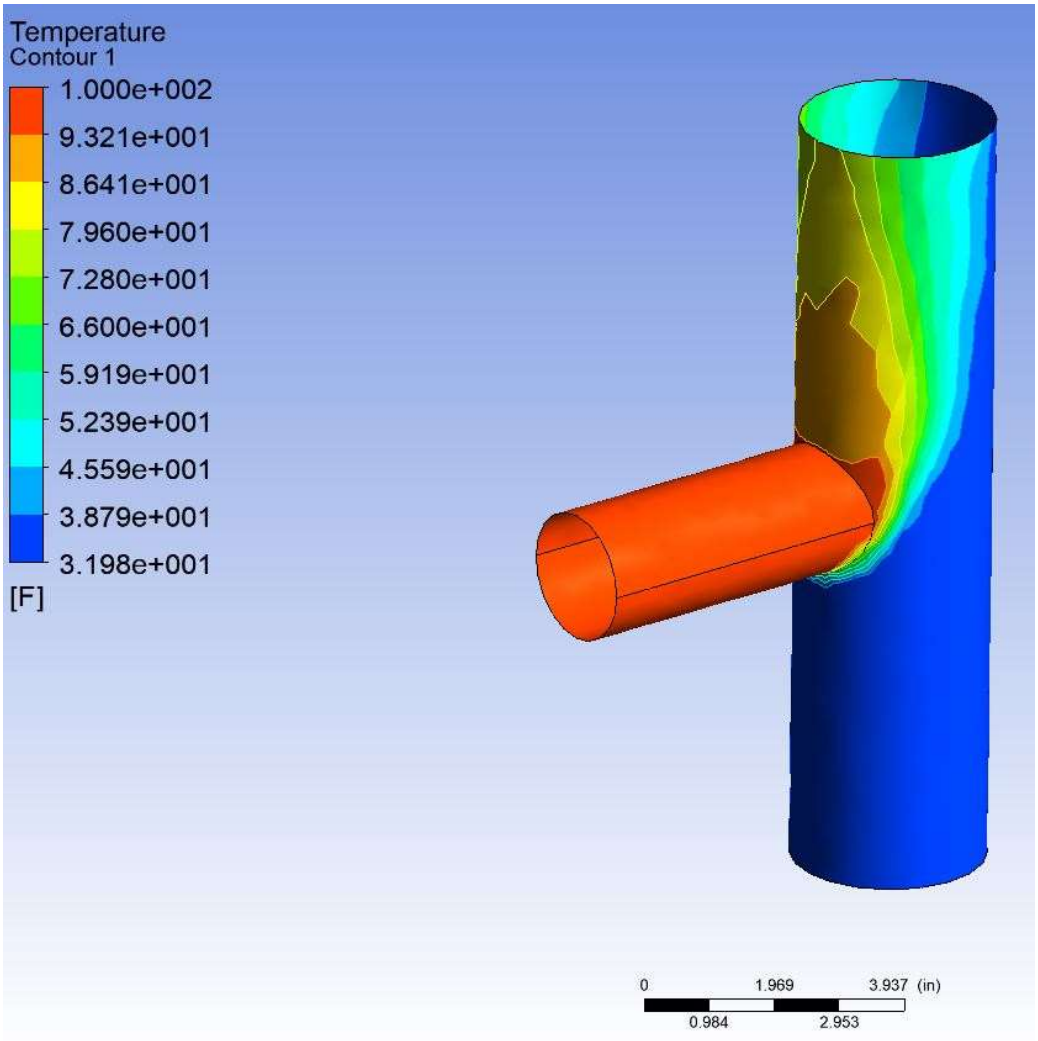

Figure 3 Temperature contour plot at the wall

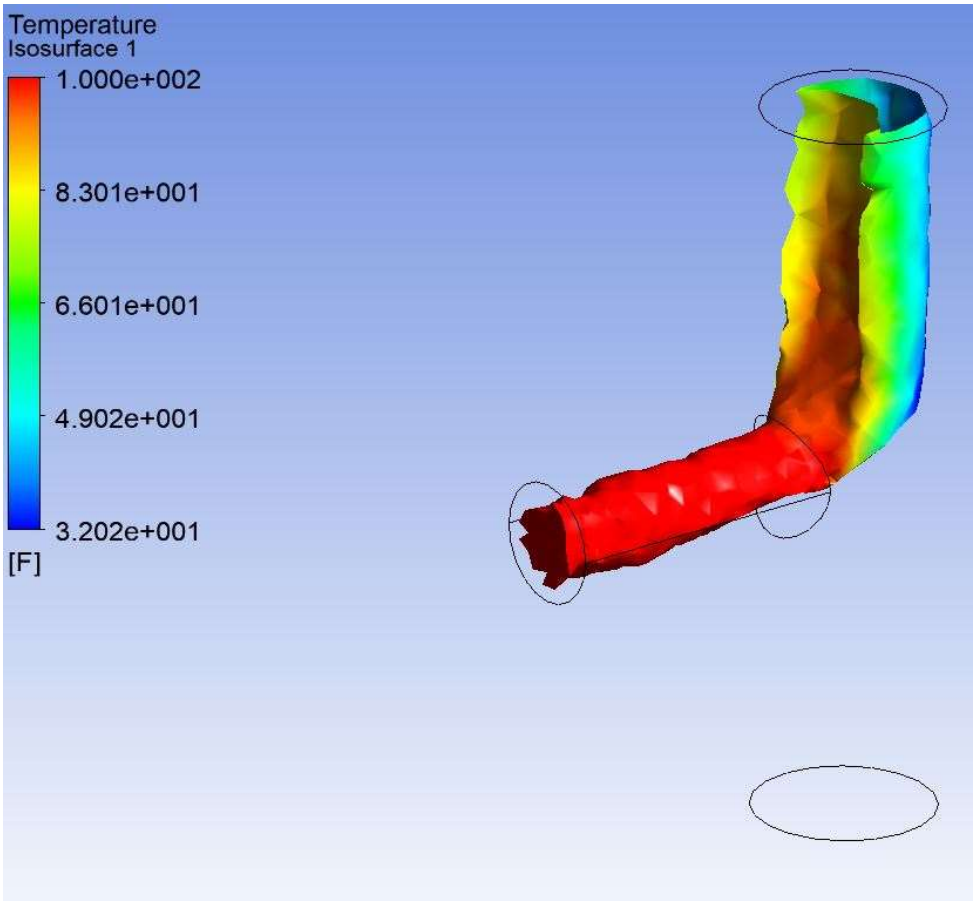

Figure 4 Isosurface plot of all the fluid above 32 F

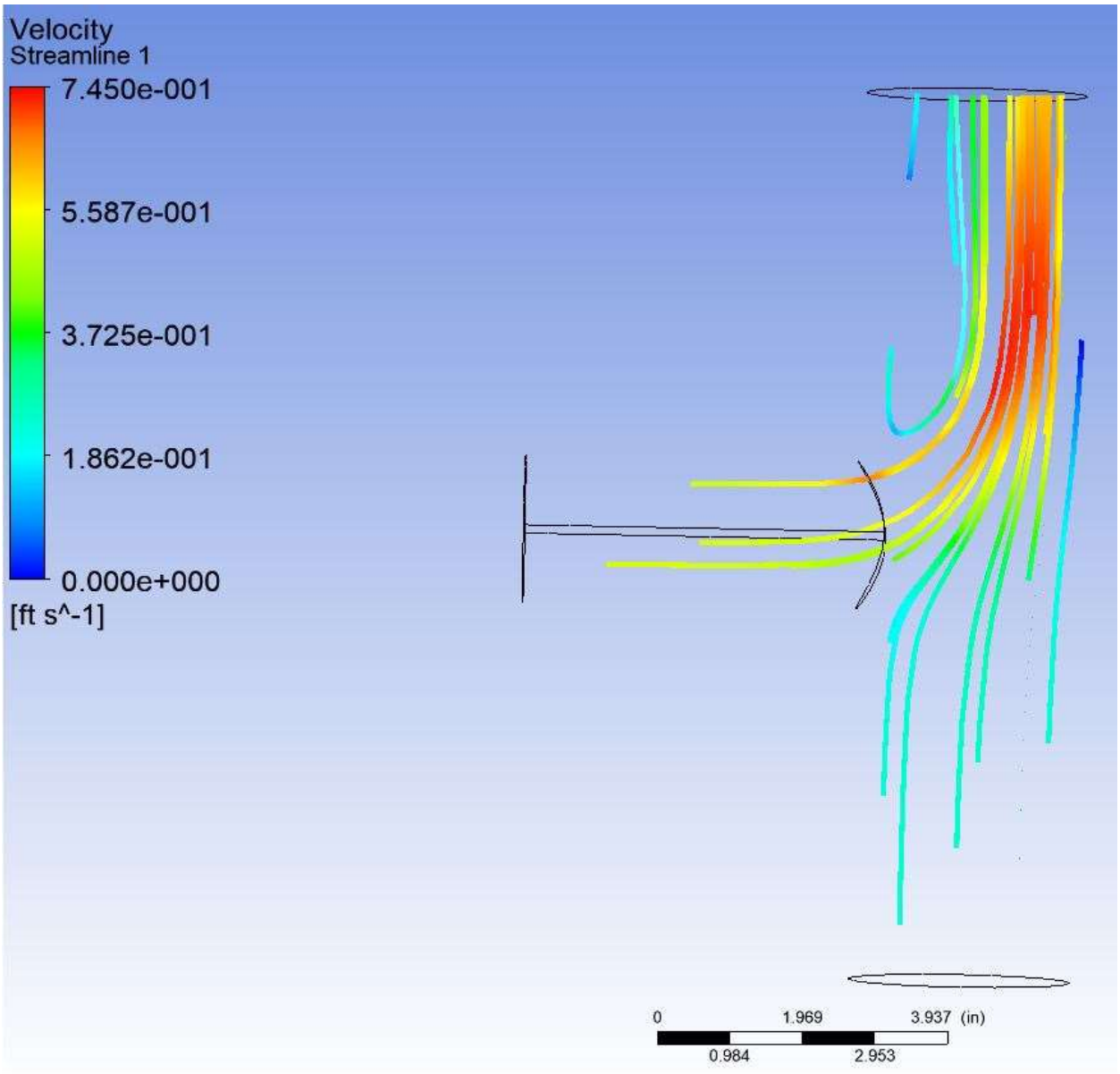

Figure 5 Streamline plot