



MECH 410 Lab 4: Flow Simulation Using Computational Fluid Dynamics (CFD)

Aidan Sarkozy - V00937139
Colm Molder - V00937879
Blaine Tubungbanua - V00918128

Oct 10, 2023

Table of Contents

1 Objective	2
2 Description of Assignment	2
3 Illustrations and Procedure	3
3.1 Initial Setup	3
3.2 Simulation Results	7
Discussion	16
Summary	16
References	18

1 Objective

The objective of this lab session is to gain an understanding of CFD simulations and become familiar with solving these types of simulations using SIEMENS NX. Through this lab, students will study the impact of part geometry on fluid flow patterns and pressure distributions. By the end of the lab session, students should have a working proficiency in setting up CFD simulations in NX and recognize the solution's impact on the simulated part.

2 Description of Assignment

This assignment explores the concept of CFD by conducting a flow simulation on a bent steel cast iron pipe that is meant to connect to the inlet of the pump casing from previous assignments. The pipe model's diameter is based on an actual straight pipe sold by McMaster-Carr, and the overall dimensions are provided in a drawing [1]. The model is then copied using the Wave Function to be able to create two bodies. Where one is assigned water for its material and it appears as the inside of the pipe. The other body is the pipe itself and it is assigned a cast iron material. The fluid body is then meshed at 10mm with an internal mesh gradation of 1. The next step was to provide boundary conditions at the inlet and outlet of the pipe. The inlet flow rate is assigned a value of 100 m³/h and the outlet is assigned as an opening. The overall convection coefficient of the water is 3000 W/m²°C. Given that the pipe is rough, the flow surface is set to 2mm of sand grain roughness height. Once all the boundary conditions and constraints are set, the simulation can be solved. The static pressure and velocity throughout the pipe are noted and discussed.

CFD employs the finite volume method and conservation equations to solve for relevant variables such as temperature, velocity, and pressure[2]. By splitting an overall shape into a finite number of control volumes, a computer is able to compute many conservation equations at once to solve for the interactions at the boundary conditions of the control volumes.

CFD analysis is a useful initial tool to validate ideas, but should always be verified through physical experimentation. Scaled models of surfaces and bodies are used for physical testing along with sensors and probes to measure required variables. These variables are then compared with the CFD to verify its effectiveness and to continue to optimize the model.

3 Illustrations and Procedure

3.1 Initial Setup

To start the CFD analysis, the part was modeled based on a McMaster available pipe seen below.

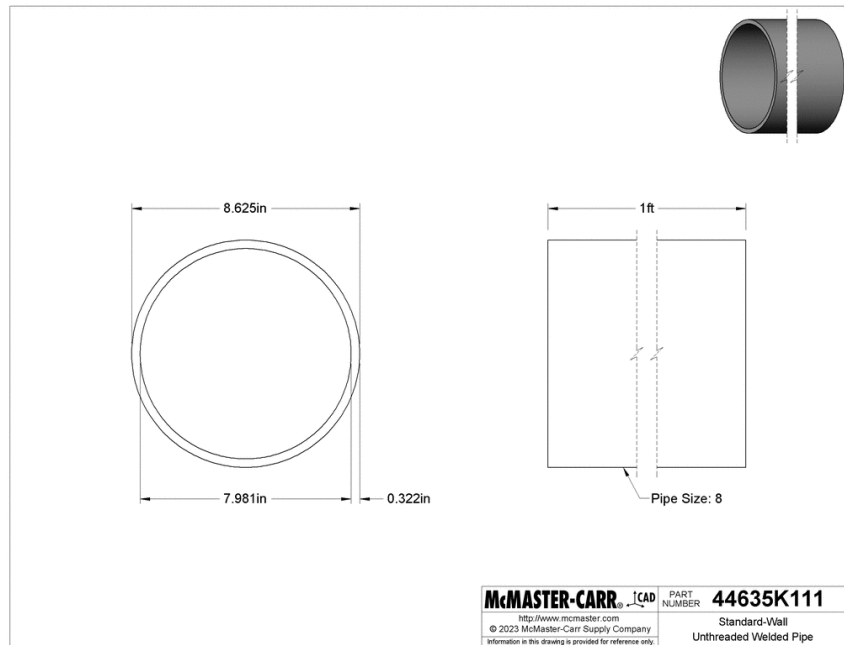


Figure 1: McMaster-Carr Stainless Steel Pipe

The modeled pipe with the specified 750mm straight and 150mm, 90 degree bend radius is seen below.



Figure 2: Model of Pipe

The pipe is to be attached to the housing with a stainless steel flange such as the one shown in figure 2.2. The pipe would be welded to the flange, and the flange would be bolted to the housing. However, since the housing's bolt hole diameter is 380mm metric, and there are no matching flanges available on McMaster-Carr, a custom flange would have to be machined to match the bolt hole circle. A water-resistant gasket such as McMaster Part 6032N53 (Figure 2.3) would then be used in the slot machined in the housing to ensure a sealing surface. As there are no 150mm OD gaskets available on McMaster to accommodate the housing dimensions, a custom gasket will have to be manufactured.

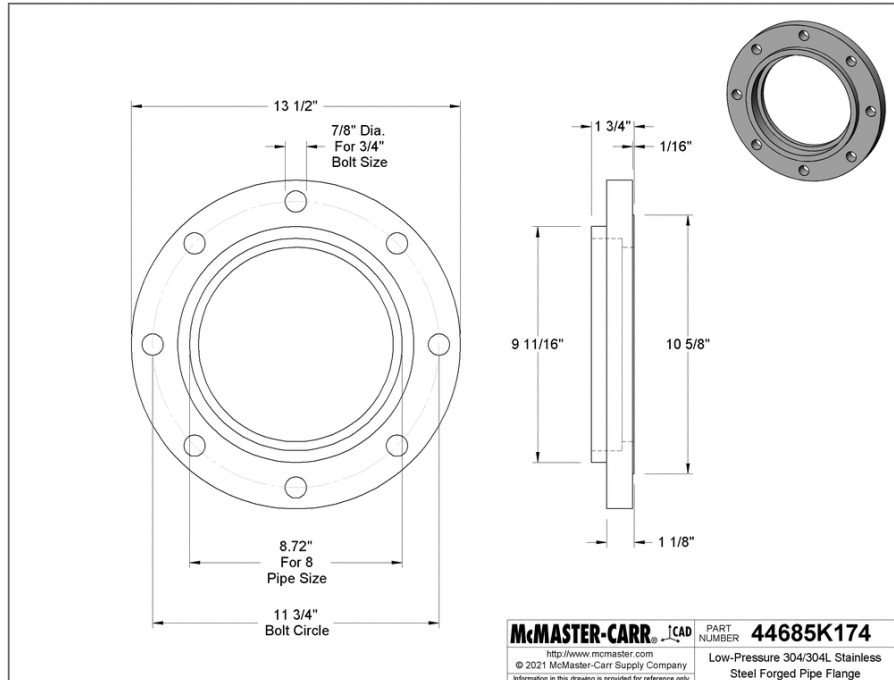


Figure 2.2: McMaster-Carr Stainless Steel Pipe Flange

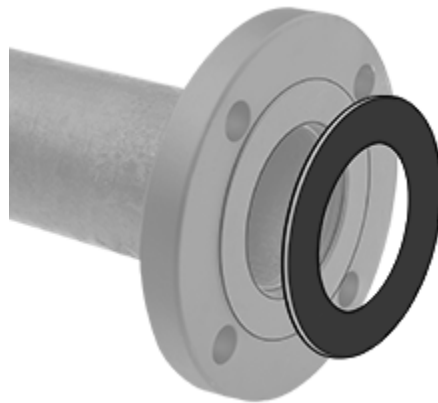


Figure 2.3: McMaster-Carr Water Resistant Rubber Gasket

With the pipe modeled, the body was wave-linked. This was done to create two separate overlapping bodies. The ideal pipe body had the outer faces deleted to leave only the internal water volume. This is seen below in Figure 3 with the water volume coloured blue.

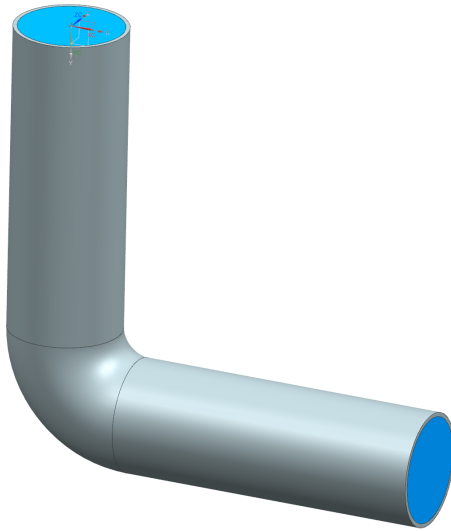


Figure 3: Wave linked Water Volume

After this, the internal volume had its material set to water and the outer pipe was hidden. Then, a mesh was created on the internal water volume. The mesh chosen was a 10mm mesh with 1 internal gradation. Internal gradation was set to 1 in order to create a consistent mesh throughout the water volume. This will result in more accurate flow profiles when analyzing cross-sections. Mesh control was initially used on the inlet, outlet, and corner faces to increase accuracy, but it was opted to reduce the mesh size of the full volume instead as simulation times were short. A photo of the mesh is seen below.



Figure 4: Mesh Created

Once meshing was completed, the simulation parameters could be set. First, the fluid inlet was set to a volume flow of $100\text{m}^3/\text{h}$.

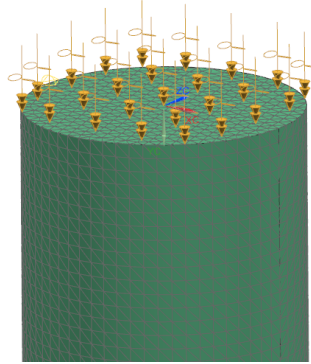


Figure 5: Setting Inlet

After setting the inlet, the opening on the other end was set. An opening was used as it will automatically determine the volume flow rate at the exit.

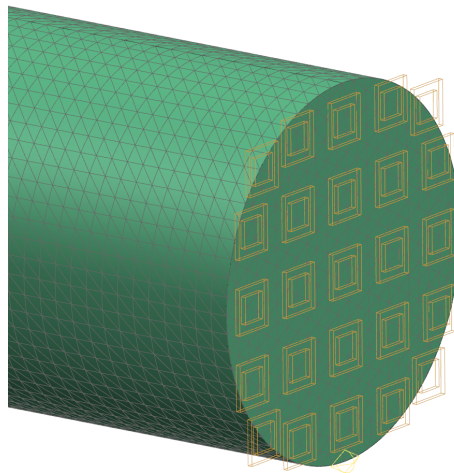


Figure 6: Setting opening

Next, a no slip surface with a surface roughness of 2mm was set the water volume that contacts the pipe.

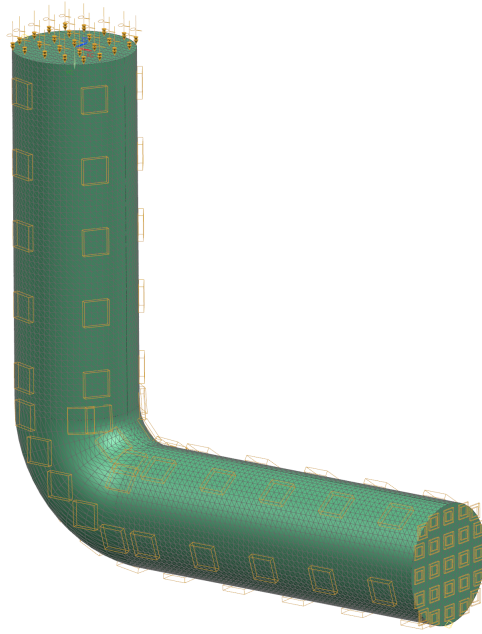


Figure 7: Setting surface roughness

Finally, a surface convection with environment of $3000 \text{ W/m}^2\text{-C}$ was set to the same surface as above.

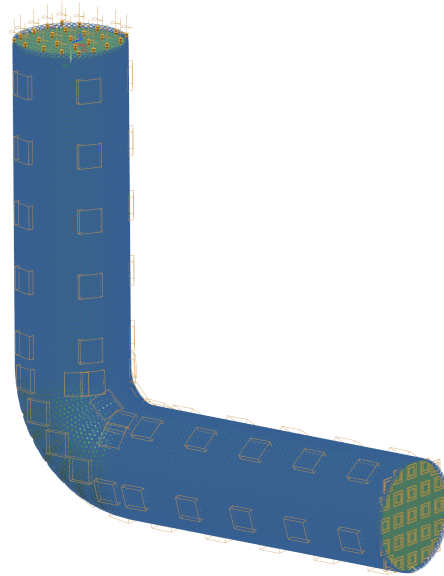


Figure 8: Setting convection to environment

3.2 Simulation Results

After setting up the simulation, a solution was created. The convergence plots of this simulation are seen below.

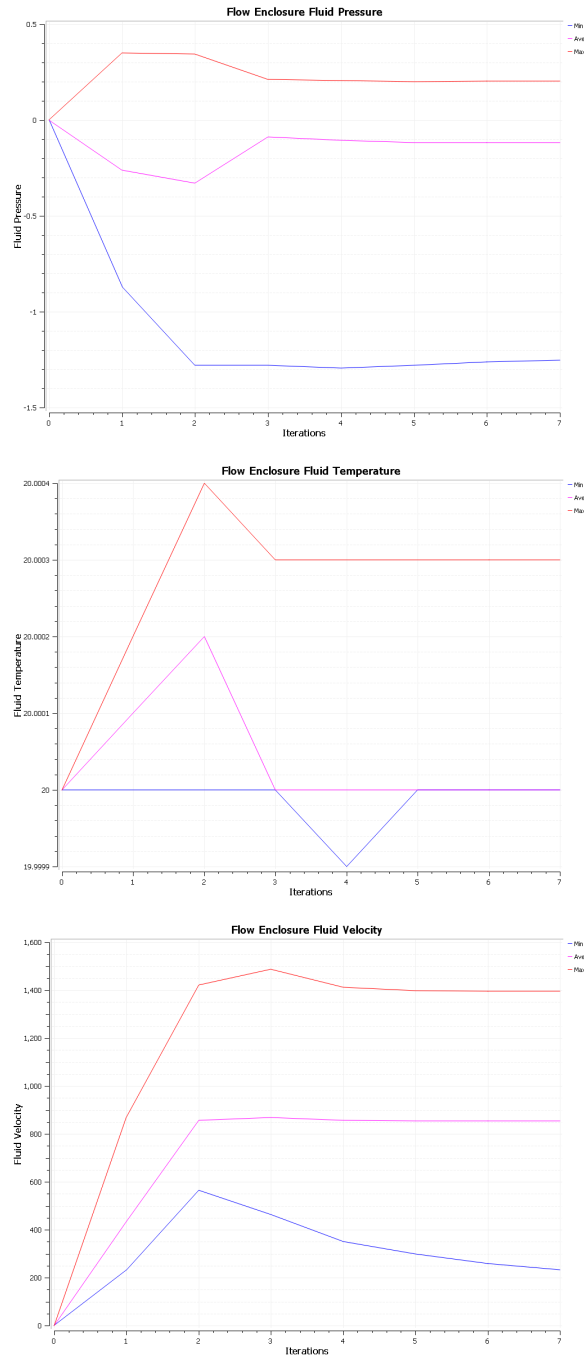


Figure 9: Convergence Plots

The resulting static pressure distribution is seen below.

Bent_Pipe_sim1 : Solution 1 Result
Load Case 1, Static Step 1
Static Pressure - Element-Nodal, Unaveraged, Scalar
Min : -0.0012528, Max : 0.000202378, Units = MPa

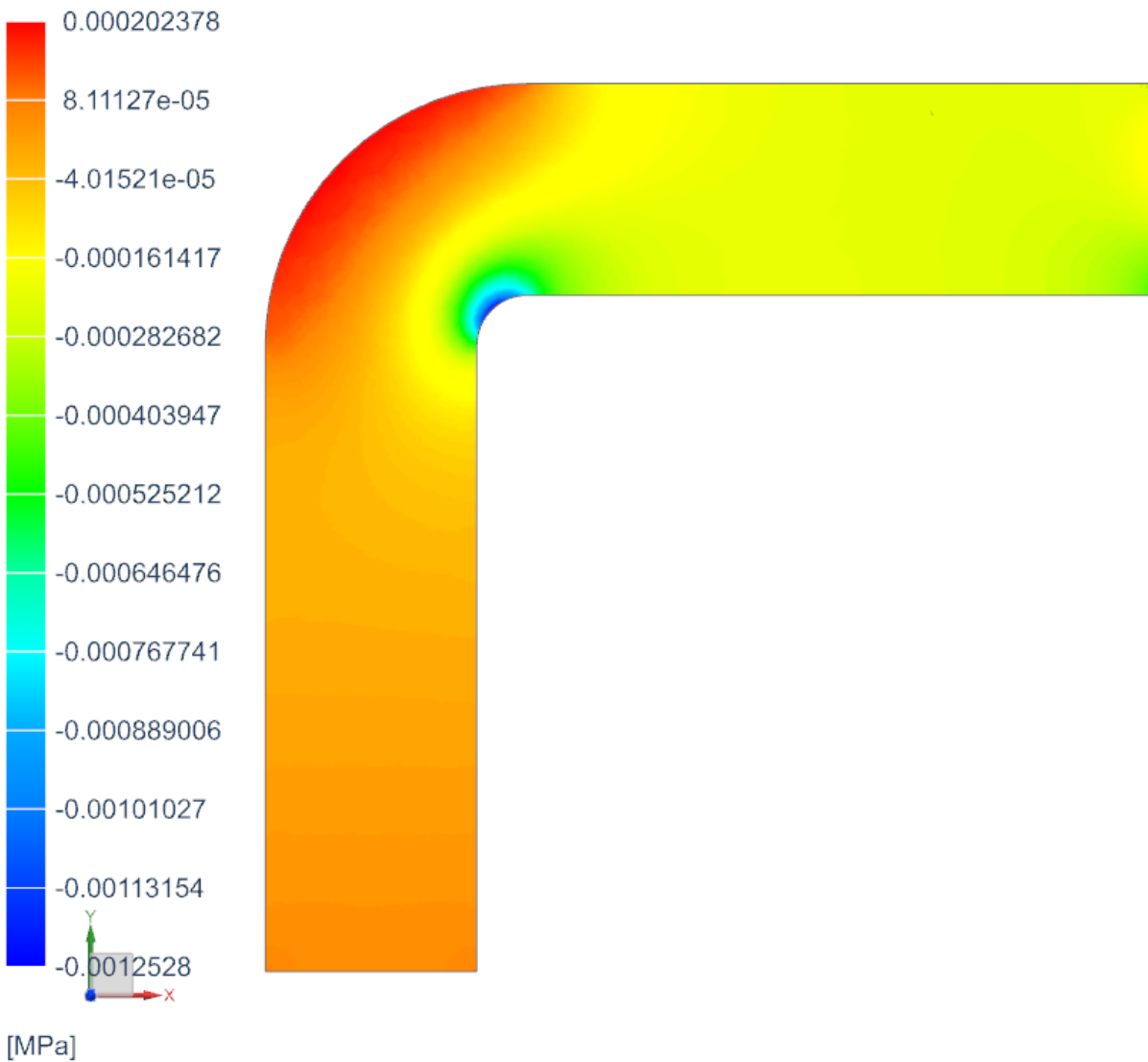


Figure 10: Static Pressure Gradient (Cross Section)

When the static and dynamic pressures are summed together it results in the total pressure which is shown in the figure below.

Bent_Pipe_sim1 : Solution 1 Result
 Load Case 1, Static Step 1
 Total Pressure - Element-Nodal, Unaveraged, Scalar
 Min : -0.000612829, Max : 0.000459138, Units = MPa

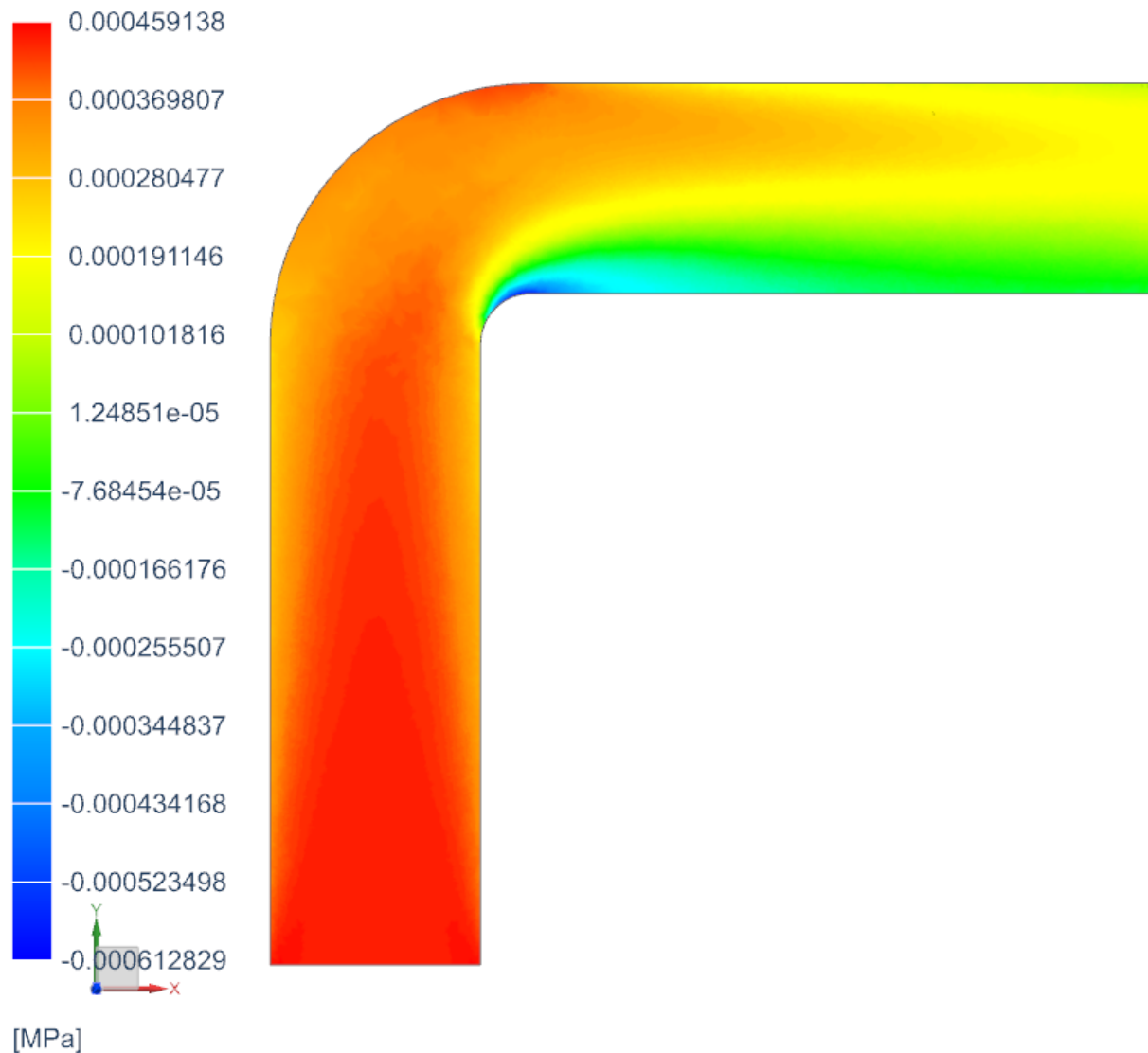


Figure 11: Total Pressure Gradient (Cross Section)

The resulting velocity plots (shown with colour gradation and then arrows) are seen below.

Bent_Pipe_sim1 : Solution 1 Result
Load Case 1, Static Step 1
Velocity - Element-Nodal, Unaveraged, Magnitude
Min : 231.87, Max : 1394.14, Units = mm/s

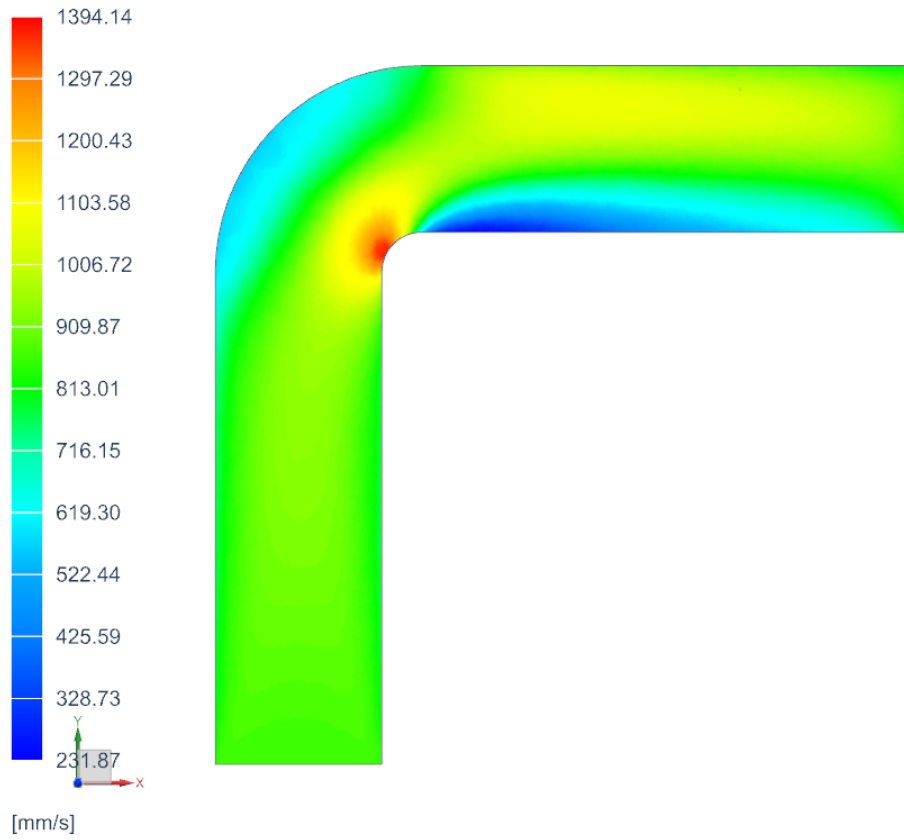


Figure 12: Velocity With Color Gradation (Cross Section)

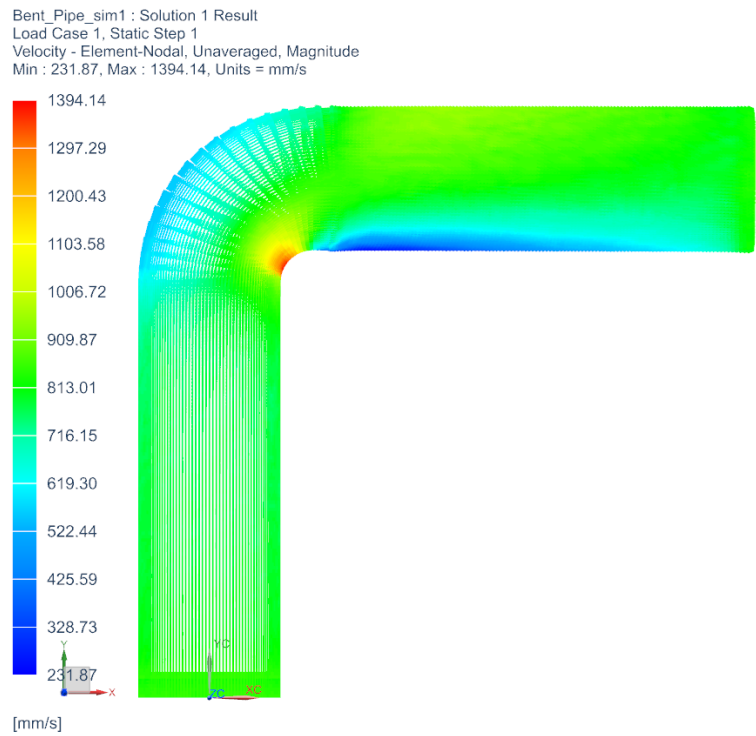


Figure 13: Velocity with Arrows

The temperature distribution is seen below.

Bent_Pipe_sim1 : Solution 1 Result
Load Case 1, Static Step 1
Temperature - Nodal, Scalar
Min : 20, Max : 20.0003, Units = °C

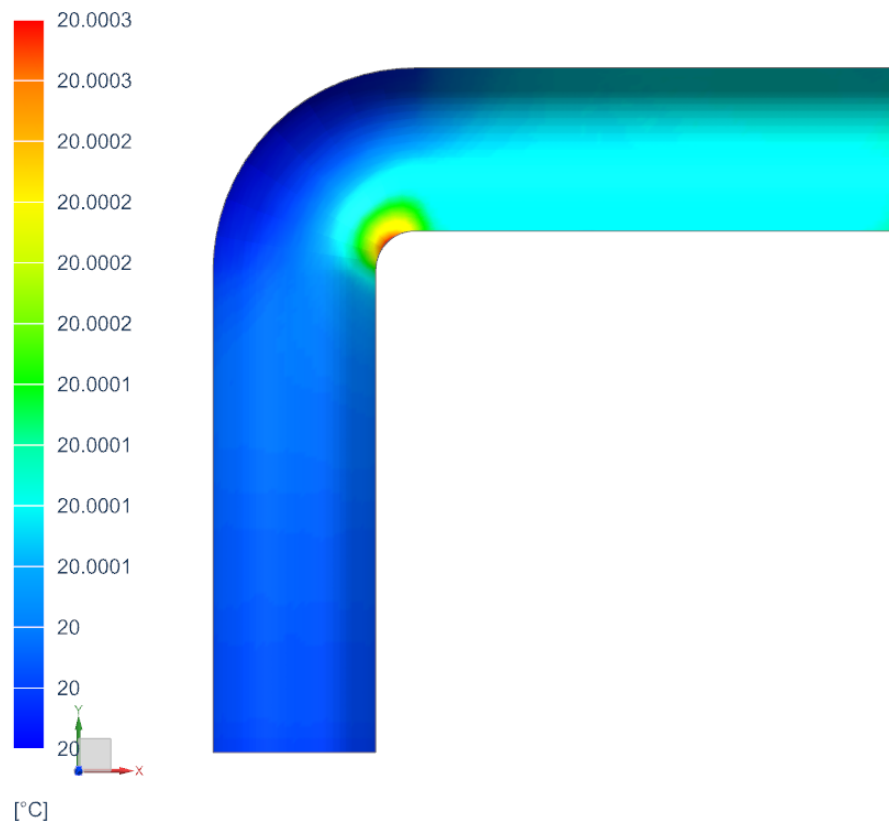


Figure 14: Temperature

Creating a cross section 250mm from the outlet, the following flow distribution is observed.

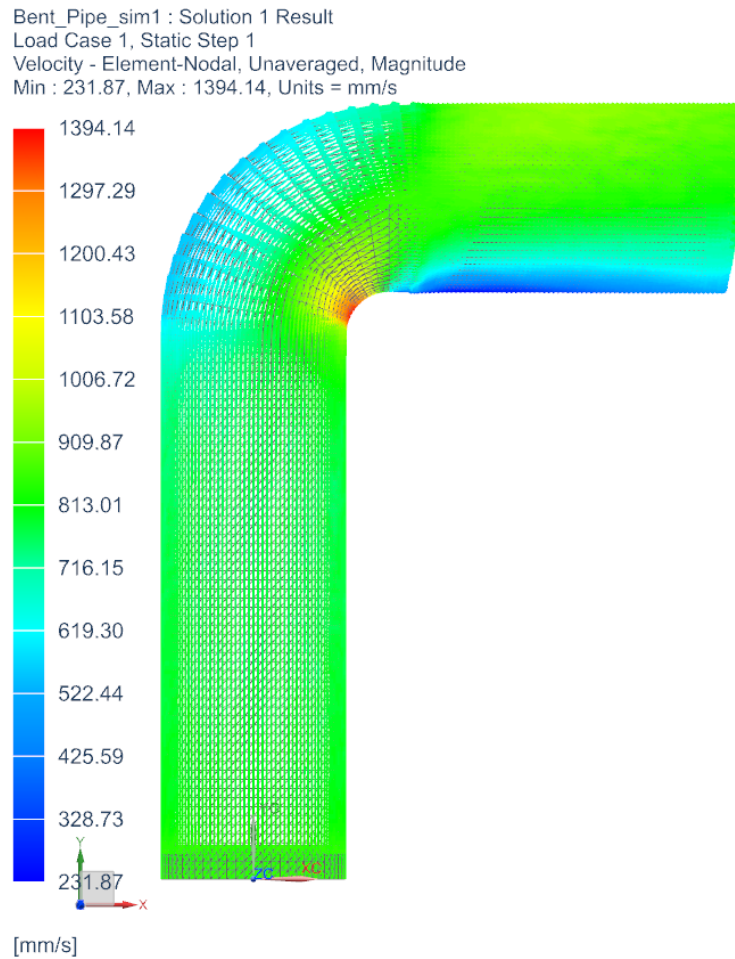
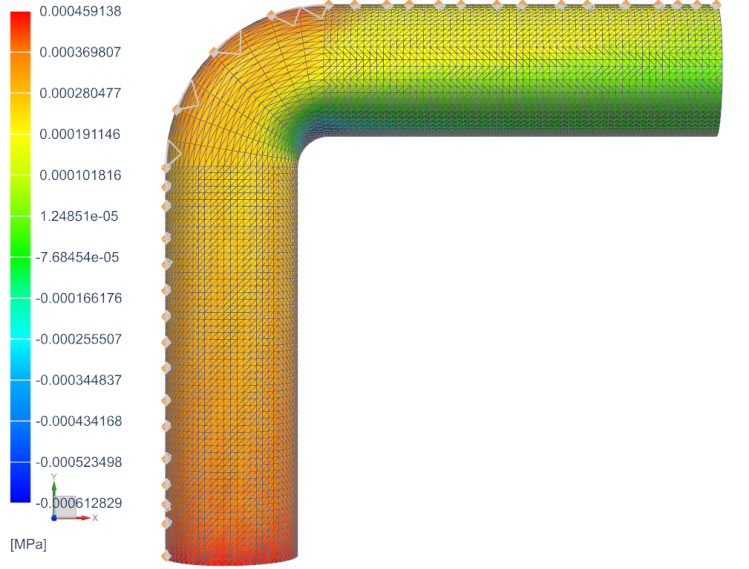


Figure 15: Velocity profile at section C-C

Next, pressure plots along the inner and outer sides of the piper were created. The elements selected to create these two plots are seen below.

Bent_Pipe_sim1 : Solution 1 Result
 Load Case 1, Static Step 1
 Total Pressure - Element-Nodal, Unaveraged, Scalar
 Min : -0.000612829, Max : 0.000459138, Units = MPa



Bent_Pipe_sim1 : Solution 1 Result
 Load Case 1, Static Step 1
 Total Pressure - Element-Nodal, Unaveraged, Scalar
 Min : -0.000612829, Max : 0.000459138, Units = MPa

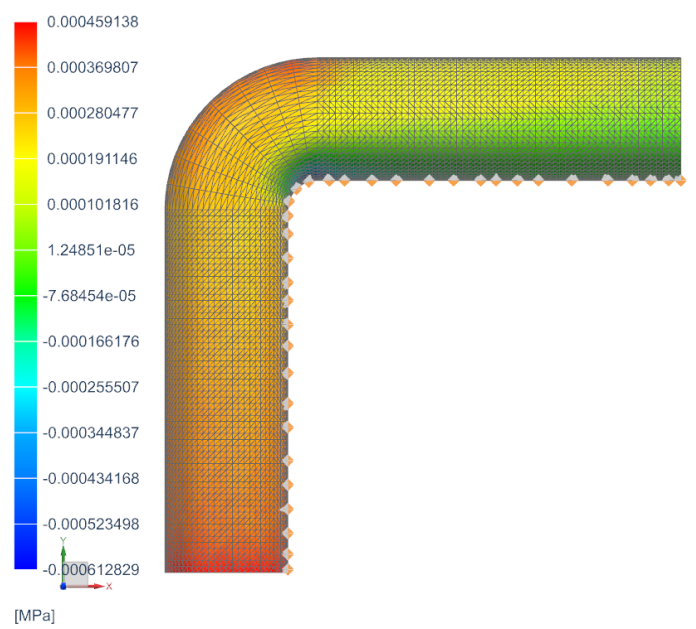


Figure 16: Location of Data points for Inner and Outer Pressure Drop Curves

The resulting plot of the pressure along these curves is seen below:

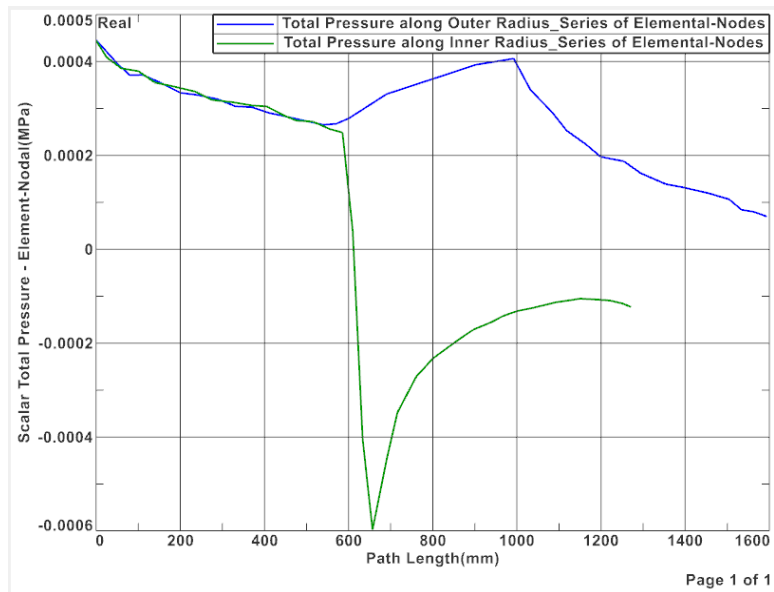


Figure 17: Plots of Pressure Drops along minimum and maximum curve radii

Discussion

The max velocity appears at the inner corner of the pipe since there is a pressure drop caused by the centripetal force of the fluid turning the corner. Initially, the fluid is moving in a straight line and has linear inertia. As the fluid reaches the corner it wants to keep going straight, resulting in a majority of the water molecules pooling at the far edge of the pipe, thus increasing the pressure there and lowering the pressure at the inner edge. This result can be seen in both the colour gradient images (figures 10 and 11) and the pressure drop plots (figure 17). In these plots, we can see the pressure along the pipe's inner and outer sides starting constant and then diverging at the corner. The maximum pressure observed along this edge was 0.000459MPa.

Due to Bernoulli's principle, a lower pressure results in a higher velocity at the inner edge. This result is seen in Figures 12 and 13 with a max velocity of 1394.14mm/s. The minimum velocity is caused by the friction of the fluid over the pipe as there is an increase in contact after the bend as the corner creates a more turbulent flow and alters the direction of the fluid. The minimum velocity was found to be 231.87mm/s.

A corresponding temperature increase of 0.003 C is seen in the inner corner. This is mostly insignificant but can be related to the increased kinetic energy of the water as it is accelerated at the bend. Assuming water as an incompressible liquid, water's caloric equation of state can be assumed to be independent of pressure, and simply a relationship between internal energy and temperature.

Summary

For the specified bent steel pipe with a water flow rate of 100 m³/hr, it was discovered using CFD analysis that the liquid reached a max velocity of 1394mm/s at the inner corner of the pipe bend. The minimum velocity appeared at the inner edge of the pipe just after the bend and had a value of 232 mm/s. The inverse is seen for static pressure, where the max static pressure appears at the outer edge of the bend, and the minimum pressure appears at the inner edge of the bend. The values for maximum and minimum static pressure are 2.02×10^{-4} MPa and -1.25×10^{-3} MPa respectively. At cross section C, it appears in Figure 15 that the fluid velocity is mostly consistent across the section with only the bottom edge of the pipe showing a slightly lower velocity due to friction.

Overall, this was another successful lab. CFD is a developing tool that will be extremely useful for new engineers to understand as they enter the workforce. So by completing this lab, students will have an advantage as they will now possess highly sought-after skills. It was interesting to observe the fluid velocity and pressure as a gradient of colours to visualize the working principles that govern fluid flow. Having seen images of CFD analysis' before it was exciting to build a CFD model of our own and gain confidence to be able to create new CFD analyses of our choice in the future.

Similar to other labs, the only issue we saw with this lab was the communication of expectations. Where the lab outline says one thing but opposing information was given in the lab session. While in the lab session, it was mentioned that we were to use McMaster-Carr to find a pipe of similar size to that given in the lab outline drawing. This was never discussed in the lab outline, so it is very easy to get confused about the expectations for the final report if some items are not given in writing. The same can be said for the gasket and flange deliverables that were reportedly mentioned in class but there is no further information on those aspects anywhere else. If all expectations were delivered in writing there would be no discrepancies in what the TAs expect and what they are submitted.

References

[1] “Standard-Wall Unthreaded Welded Pipe,” McMaster-Carr, <https://www.mcmaster.com/44635K111/> (Accessed: Oct. 12, 2023).

[2] University of Victoria, ‘MECH 410 Laboratory 3 Manual’, Zuomin Dong, Siyang Steven Liu, Chon Him Lawrence Wong