COMSOL Project Report

Kaysey Nguyen

May 1, 2024

Physics Problem

The physics problem is on the fluid mechanics of a pipe with its inlet at the bottom and its outlet at the top right end. With the pipe having four bends, I want to compare the velocity and pressure of water entering the pipe at 2 m/s and exiting at the end with 0 Pa when the end of the pipe is cone-shaped and not cone-shaped.

Equations

The relevant equations are from fluid mechanics, the Navier-Stokes equation (1), and the continuity equation (2). The following equations are taken from class notes. These equations will be similar in COMSOL (3) and (4) which is in the Using Comsol section.

$$\rho(\frac{dV}{dt} + V * \nabla V) = - \nabla P + \mu\nabla^2 V + f \tag{1}$$

$$\nabla * V = 0 \tag{2}$$

Details of Geometry

For the physics problem, the details of the geometry of the pipe will have a total height of 9 inches and a total width of 7.5 inches with the bends and shorted pipes connecting them. Throughout the whole pipe, its radius is 0.5 inches. The only time the radius will change will be at the end of the pipe when it is changed to a cone, making the width 8 inches, which will be explained later.

The pipe is stood vertically and begins with a 2-inch cylinder. Above that will be a 90° bend, the first bend, to the right and is 1 inch high. To the right of the bend, there is a 1x1 inch cylindrical pipe to the right that goes into another 90° bend upwards, the second bend. The third bend is the same as the first and the fourth bend is the same as the second and are connected by another 1x1 inch cylindrical pipe. At the end of the fourth bend, which should be vertical, there will be a two-inch pipe that will be the end of the pipe; no cone. For the cone, its dimensions will replace the pipe after the fourth bend, which will have a bottom radius of 0.5 inches, a top radius of 1 inch, and a height of 2 inches.

Boundary Conditions

The pipe will have two boundary conditions, the inlet and the outlet. As described in the Physics Problem section, the inlet is where water will be entering at a speed of 2 m/s while the outlet has 0 Pa. The inlet will be located at the bottom of the 2-inch cylinder at the bottom left while the outlet is at the end of the other 2-inch cylinder or cone. The material of the pipe will have water however it would be interesting to see the results with other materials if the results will be similar, especially for the end with the cone. If one cannot gain a solution for using water as the material, it is best to change the ‘Wall condition’ option from ‘No slip’ to ‘Slip’.

Using COMSOL

I will use COMSOL for building and obtaining my results as plots for both velocity and pressure. I thought it was best to build this as a 3D model and choose the ‘Laminar Flow’ and ‘Stationary’ options to begin. The first thing is to change the units from meters to inches to match the model. Building the geometry according to the Details of Geometry section was difficult at first because when I revolved the second bend, it faced downwards. Luckily I was able to fix it by rotating but later on, I found that I could change the numbers in the ‘Point on the Revolution Axis’ and ‘Direction of Revolution Axis’ in the Revolve settings with the ‘Axis type’ kept at 2D. Below are equations (1) and (2) shown in COMSOL, (3) and (4) respectively, under the Laminar Flow section.

$$\rho(u * \nabla)u = \nabla * [-\, pI + K] + F \tag{3}$$

$$\rho\nabla * u = 0 \tag{4}$$

After running the mesh at normal, any option is fine, and computing the study, I added a ‘Cut plane’ in the dataset, setting the plane to xz and the y-coordinate to 0. In a 2D Plot Group in ‘Results’, I changed the dataset to the cut plane that I added and also added a ‘Surface’. In the ‘Surface’ I kept the velocity that was there on default and plotted. Making a 2D Plot Group for pressure is the same way, but changing the default velocity to pressure instead. Another 2D plot Group with the same cut plane will be made to get the pressure contours and arrow surface of the pipe. The arrow surface is to visualize where the pressure is going. Thus, the number of plots in total for the results will be six, or three for the pipe without cone and with cone.