INTRODUCTION

This system contains a cylindrical pipe with an axisymmetric sudden expansion. The symmetry along the x-axis allows for a computational analysis over half of the pipe that is mirrored on the other half. The inlet pipe has a radius, R1 = 1m and the outlet pipe has a radius , R2 = 2m. The flow has an inlet velocity of V1= 0.277 m/s and outlet velocity to be determined using the model. The outlet pressure is atmospheric at 1atm. The fluid has a density of ρ=1kgm3 and dynamic viscosity of μ=0.01kgm*s so it is similar to water but about 10x more dense. The Reynolds number for the liquid prior to expansion is given by the equation:

$$ReD=(2R)\rho V\mu \qquad (1)$$

Using R=R1 and V=V1, ReD1=55.4 which indicates that the flow is laminar. The radius and length after the sudden expansion is R2=2m and L2=100m respectively. Now with R=R2 and V=V2The new Reynolds number is 27.6 which indicates that the flow is still laminar.

Since the flow is incompressible, the average velocity across the pipe will be inversely proportional to the cross-sectional area. Therefore, since $R_2 > R_1$, the average velocity at the outlet will be lower than V_1. Based on this relationship and the values are given, the average outlet velocity is expected to be approximately 0.069ms

Due to frictional forces across the pipe surface, it is expected that the fluid will have the largest velocity at the furthest point away from the pipe wall (i.e. the centerline), and will move slowly along the pipe surface.

INPUT

Given that the flow is laminar the entrance length is given by,

$$Le=0.06\ ReD$$

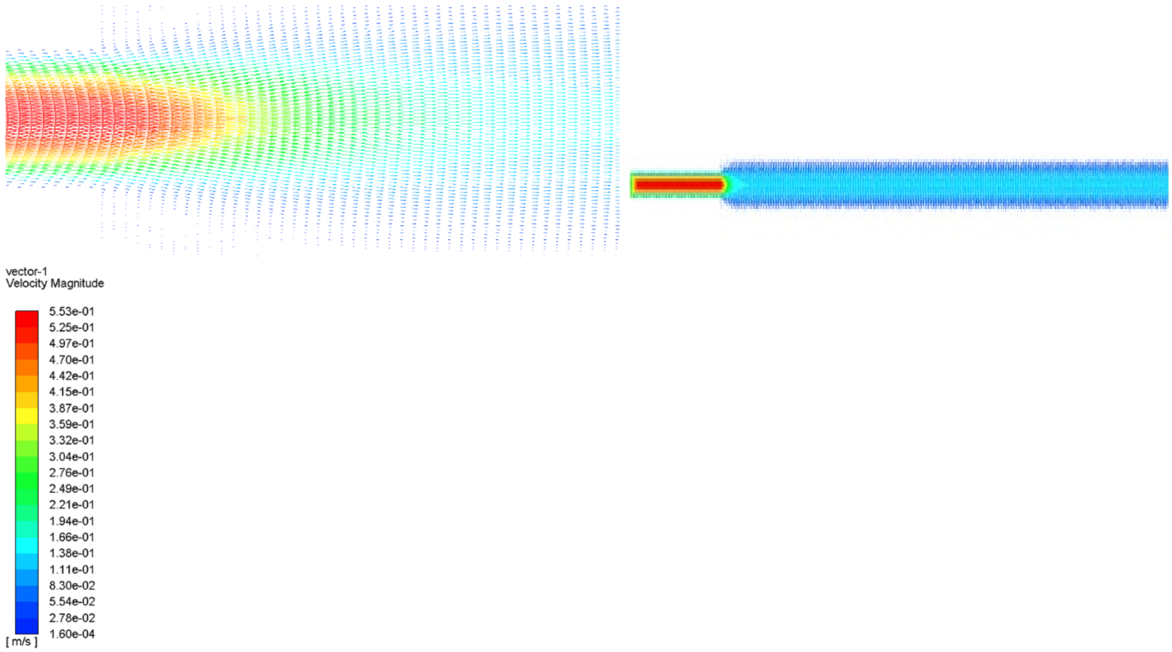
Therefore, to get a fully developed flow at the expansion point, the length of the first part of the pipe must be more than 6.65m.

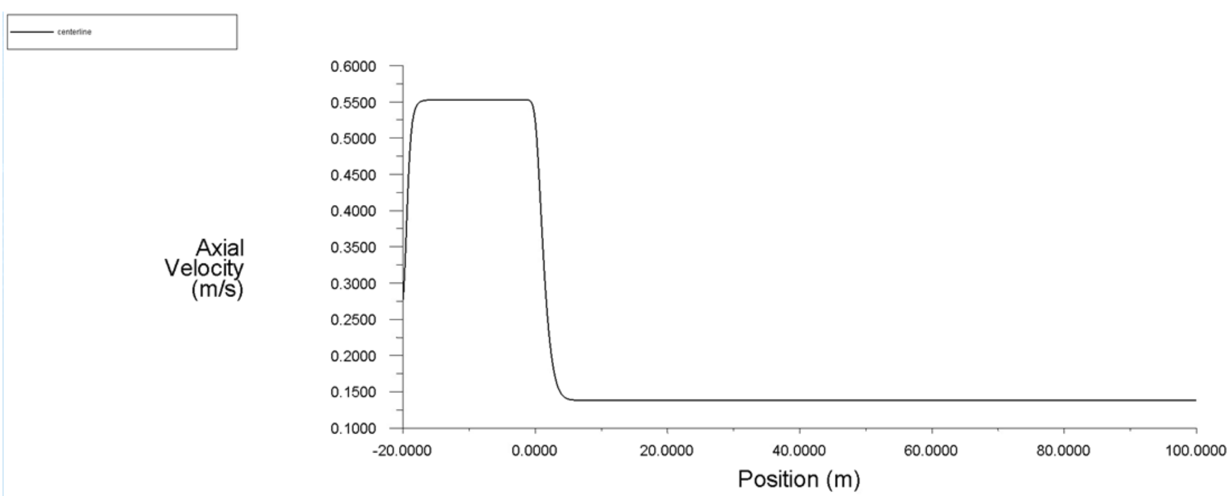
The model for the given system is made in the ANSYS workbench, by joining two rectangles, with dimensions (L1=20m, R1=1m) and (L2=100m, R2=2m), and removing the overlapping segments. This sketch is converted to a solid with a thickness of 0.1m. The mesh used for the system is grid style with a cell for every 2m along the length, and for every 0.05m along the width of the pipes. As shown in the figure below, this meshing results in 22641 nodes and 22000 elements across the system with no tearing.

In ANSYS fluent, the model is specified as viscous and laminar flow. Based on the flow conditions given, the flow is assumed to be incompressible and, therefore, the energy equation is turned off for the model. The material properties and input and output conditions are specified with, the density of ρ=1kgm3, the dynamic viscosity of μ=0.01kgm*s, a constant and uniform inlet velocity, and outlet pressure of 1 atm. And the centerline for the axisymmetric model is specified as the bottom segment.

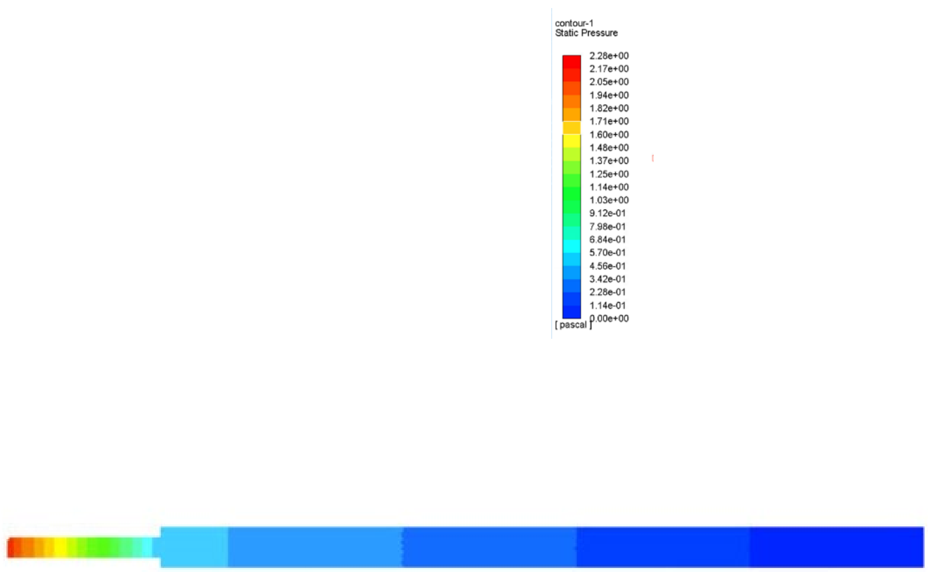
RESULTS

The velocity distribution calculated by the model confirms the initial assumption that the fluid flows faster along the centerline and slower along the edges. The velocity distribution is shown in the figure below to highlight that assumption. As seen in the figure on the left,the outlet velocity is less than the inlet velocity. This is caused because the fluid must fill the sudden extra space within the system.


vector-1
Velocity Magnitude
5.53e-01
5.25e-01
4.97e-01
4.70e-01
4.42e-01
4.15e-01
3.87e-01
3.59e-01
3.32e-01
3.04e-01
2.76e-01
2.49e-01
2.21e-01
1.94e-01
1.66e-01
1.38e-01
1.11e-01
8.30e-02
5.54e-02
2.78e-02
1.60e-04
[m/s]


centerline
Axial Velocity (m/s)
0.6000
0.5500
0.5000
0.4500
0.4000
0.3500
0.3000
0.2500
0.2000
0.1500
0.1000
-20.0000
0.0000
20.0000
40.0000
60.0000
80.0000
100.0000
Position (m)

The velocity distribution of the average velocity across the length of the pipe is shown above. As expected, once the flow is fully developed, the axial velocity in the second part of the pipe is much lower than the first.



This relationship is illustrated by the Navier-Stokes equations and upheld in the CFD solution. Equation 2 highlights when the system goes on for a long duration of time that velocity in the system will drop along with the pressure. The magnitude of pressure will change with the change in energy, and that is dependent on the fluid velocity, geometry of expansion, and the Reynolds number.

$$\rho dVdt=\rho g-\nabla p+\mu\nabla 2V \quad (2)$$

CONCLUSION

After completing the CFD analysis using a mesh size of 22000 elements, it was shown that the velocity reaches a constant value of approximately 0.535 m/s beyond a certain distance from the inlet. However, in a fully-developed laminar pipe flow, the analytical max axial velocity should be equal to two times that of the uniform inlet velocity, which in this case would mean that the max axial velocity should be around 0.554 m/s. Thus, the initial mesh is inadequate. Repeating the solution using a finer mesh, 44000 elements to be exact, the velocity profile showed a max axial velocity of around 0.551 m/s which is very close to the analytical solution and an improvement over the previous mesh. As shown in both velocity profiles, the recirculation region, or where the axial velocity reaches a constant of about 0.145 m/s happens around 5 meters. This aligns very closely with experimental data, showing that the recirculation length for *Re* = 55.4 is approximately 5 times the inlet radius.[2] To further compare the results, a contour of the stream function was created using a band size of 50 levels. The results of the contour coincide very closely with the streamlines obtained via PIV experiment.[2] To ensure that increasing the mesh size again would not lead to even better results, 3 more trials were conducted to further verify and validate the solution. It shows that there was convergence on 1E-6. Thus, it indicates that the model is accurate and our mesh number was adequate to predict the real-world model.

VALIDATION

Meshing is important because it provides the accuracy of the model. It breaks the model up into smaller control volumes. This was determined because the model has no tearing with the boundary conditions. If the nodes and elements are increased the model becomes too complicated to solve analytically and begins to break down. While if there were fewer nodes and elements there would be tearing on the boundary layer and the model would not be valid. Below is a plot showing convergence using a mesh element of 44000. There were four other trials completed and they converged to 1E-6 at 1970, 2786, 2787, and 3914 iterations indicating that the model is valid.

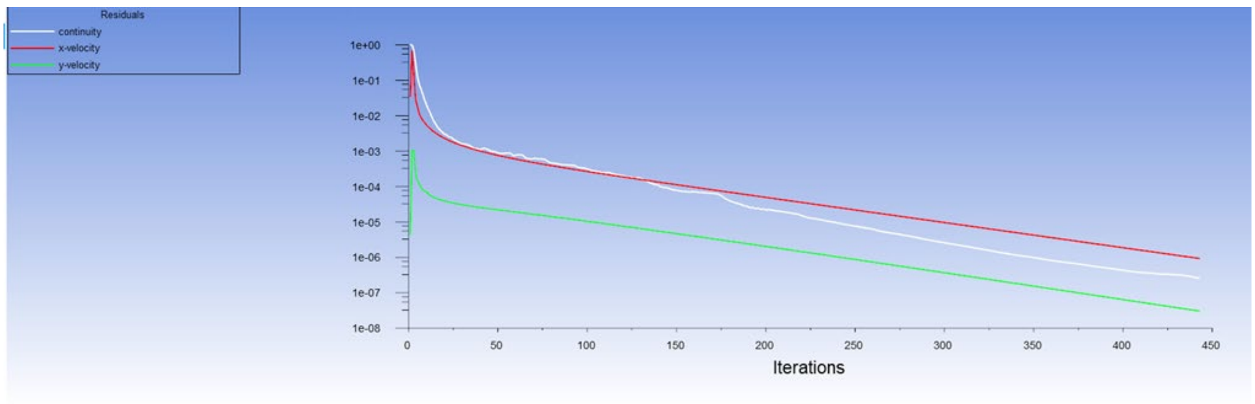
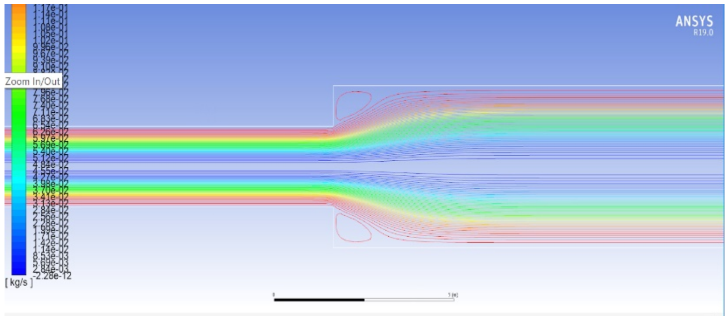


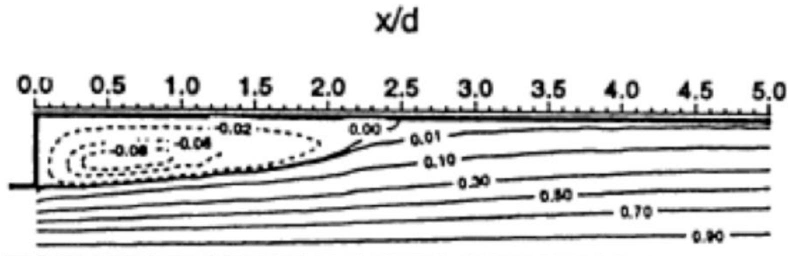
Figure 5 Convergence Plot

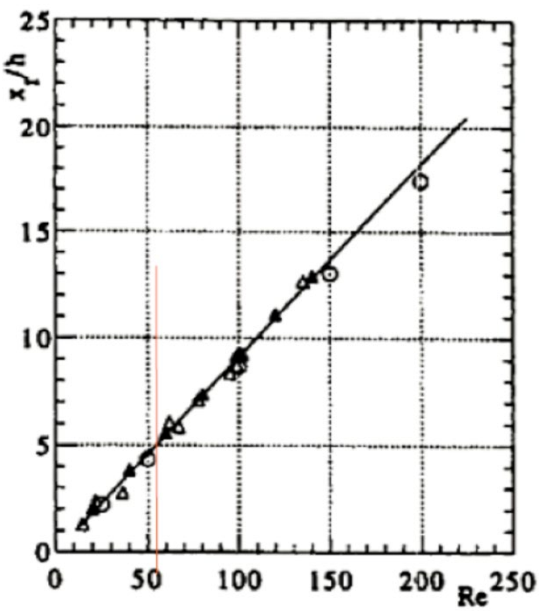
Works Cited

1. White, Frank M. *Fluid Mechanics*. 8th ed., McGraw-Hill Education (India), 2017.

2. Hammad, KJ, Otugun, MV and Arik, EB “A PIV study of the laminar axisymmetric sudden expansion flow”, **Experiments in Fluids** 26 (1999) pp. 266-272.







Weston Dastrup	Modeling, Writing, Editing
James Paloukos	Modeling, Writing, Editing
Joshua Whitehead	Modeling, Writing, Editing
Sai Pandit	Writing, Editing
Tyler Riddle	MIA