Anthony Gisolfi
EMCH 360
12/7/2022

Analysis of Pressure Drop in Different Pipes

Abstract

Pressure drop is a powerful tool which is united by engineers of the healthcare and oil industries to diagnose artery disease and make efficient oil pipe systems. In this study, the ANSYS fluent software was used to model pressure drop in 16 different simulations. These simulations were run in inviscid and viscid conditions with altering velocity of the flow (turbulent/laminar), diameter of the pipe, type of liquid, and shape of object obstructing the pipe. The efficiency and accuracy of the models were analyzed through the pressure and velocity contours and the calculated required power to obtain the flow. The simulations with triangular obstructions were found to show more accurate fluid models in inviscid conditions while circular obstructions thrived under viscid conditions. Trials with turbulent velocities required larger amounts of power and had smaller friction factors due to their high velocity compared to those with laminar velocities. The laminar pipes models tended to be more efficient than the turbulent ones due to low required power. The most efficient model was the 2m circular obstruction with laminar octane flow.

Introduction

Pressure drop can be defined as the difference in pressure between two points, especially in a fluid network. In the world of fluid mechanics, this concept is highly valuable to engineers as it provides valuable information of how a fluid network will react in a given situation. There are many variables that influence pressure drop which include but are not limited to size of the pipe, velocity of the fluid within the pipe, the type of fluid present, and the viscosity of the fluid. Pressure drop is often calculated though the use of the extended Bernoulli's equation:

$$z_1 + \frac{v_1^2}{2g} + P_1\mu_1\frac{g_c}{g} + H_p = z_2 + \frac{v_2^2}{2g} + P_2\mu_2\frac{g_c}{g} + H_L \qquad [1]$$

Anthony Gisolfi
EMCH 360
12/7/2022

Pressure drop is key concept in many industries such as healthcare and oil manufacturing. Healthcare professionals require pressure drop to fully understand how arteries pump blood in the human body. Research of pressure drop in this field can help to diagnose dangerous diseases that involve blood pressure like atherosclerosis. A recent study measured the pressure drop in the coronary arteries in an effort to fight coronary artery disease. This study aided in formulating algebraic methods of calculating pressure drop in common scenarios of arteries. These advancements helped to create more accessible artery disease diagnosis since an algebraic method is much more applicable to any healthcare provider opposed to expensive 3D artery simulations (Mirramezani, 2019).

The oil and gas industries rely immensely on pressure due to the nature of obtaining and transporting these materials through pipes. It is important for the oil and gas industry to properly understand how the pressure acts in these pipes to prevent leaks. In pipe systems with multiple branches, pressure drop can be used as a method to determine the efficiency of the system. In a 2020 study, a theoretical approach was created using pressure drop as an analysis tool for multibranch horizontal pipe system. In this study, various models of the pipe system also studied how acceleration, friction, mass transfer, mixing, confluency, and gravity influenced the efficiency of the system. Results of these parameters and the simulations allowed for the creation of a model to determine the most optimal pipe system for a given scenario (Yue, 2020).

In this study, the fluid analysis software ANSYS was used to run 16 simulations which showed the pressure and velocity contours of a pipe with 2 different obstacles, 2 pipe diameters, laminar/turbulent velocity patterns, and 2 different fluids. To observe fluid flow blocked by an object, geometry of the pipe was set with either a circle or triangle obstacle imbedded in the pipe. In each of the 16 cases, simulations were run in both laminar and inviscid scenarios to determine which flow scenario provided the best model. Laminar cases were suspected to be most efficient models due to their small velocities.

In the following section, titled modeling, the geometry of the simulations is outlined in detail. Results and analysis of all simulations are listed in the results and discussion section below. Correlations and relationships determined from the results section are located in the conclusion section.

Modeling

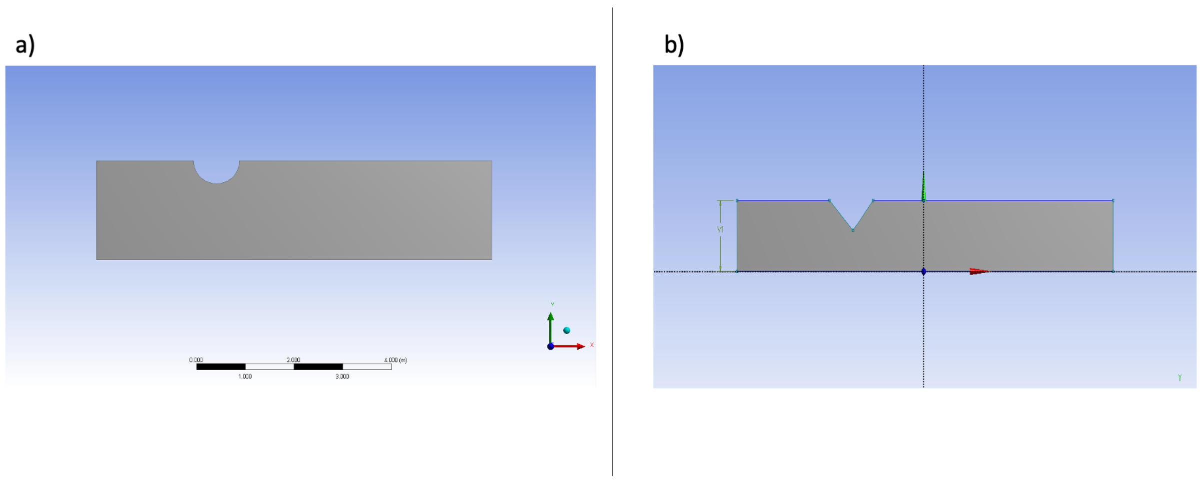


Figure 1. Displays the geometry of the simulation. (A) Circular obstruction geometry with a 2m inlet. (B) Triangular obstruction with a 1m inlet.

The geometry of the simulations, as seen in **Figure 1**, was an 8 m long pipe with either a circle or triangle object in the pipe made in the ANSYS Design Modeler software. Diameter of the pipe was set to be either 2 m or 1 m. The velocity of the flow was set to be either 0.0001 m/s to simulate laminar flow or 0.004 m/s to simulate turbulent flow. The type of fluid in the simulation was either liquid water or octane.

To analyze the pipe flow, pressure and velocity contours of each simulation were observed. The power required to obtain the flow was also analyzed. Reynolds number was obtained using **Equation 2**.

$$Re = \frac{\rho V L}{\mu} \quad [2]$$

Using a surface roughness of drawn tubing, 2.5 μm, the relative pipe roughness was calculated. Friction factor was obtained using the Moody chart for turbulent cases and the Darcy friction factor equation for laminar cases. (**Equation 3**)

$$f = 64/Re \quad [3]$$

With the use of friction factor, head loss was obtained using **Equation 4**.

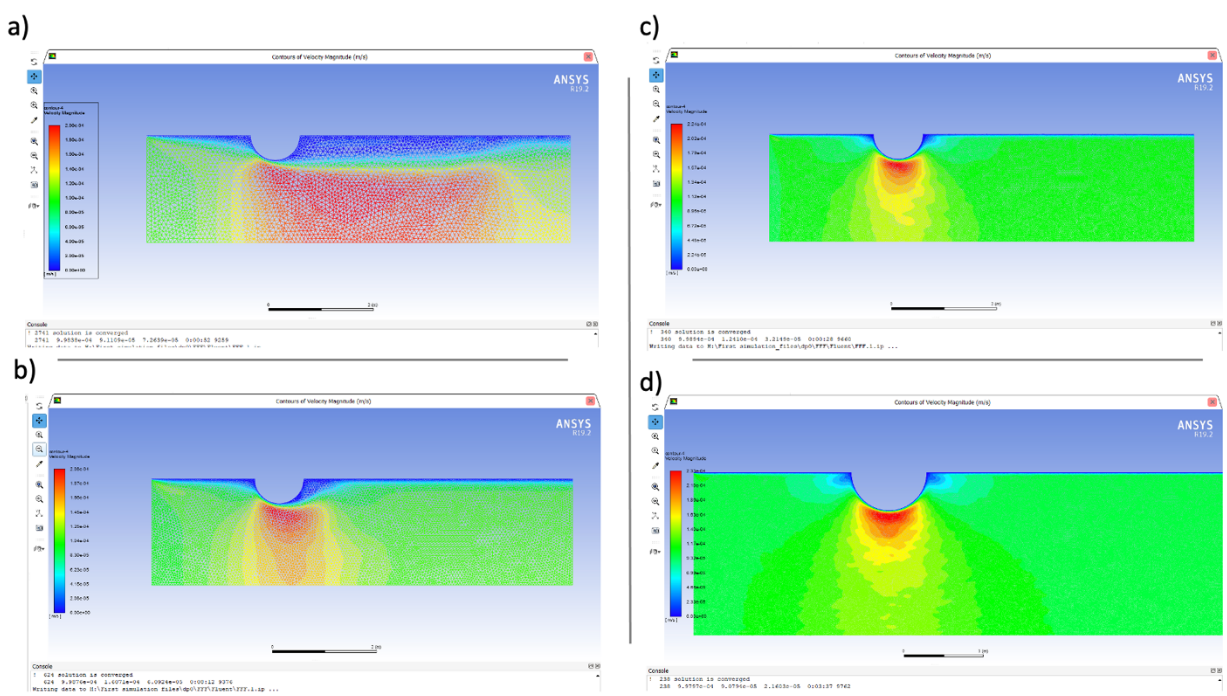
$$h_L = f\frac{LV^2}{D2g} \quad [4]$$

Using head loss, which is derived from the extended Bernoulli's Equation (**Equation 1)**, pressure drop was calculated using **Equation 5**.

$$\Delta p = \rho g h_L \quad [5]$$

Finally, power required to obtain the flow of the liquid was calculated using **Equation 6**.

$$Power = \Delta p A V \quad [6]$$



Anthony Gisolfi
EMCH 360
12/7/2022

Figure 2. Displays the velocity contours for the respective mesh sizes of the grid independency study. (A) Body mesh size of 0.075 m and an edge size of 0.0375 m. (B) Body mesh size of 0.05 m and an edge size of 0.025 m. (C) Body mesh size of 0.025 m and an edge size of 0.0125 m. (D) Body mesh size of 0.0125 m and an edge size of 0.00625 m.

To ensure no mesh size bias existed, a grid independency study was completed. In this grid independency study, 4 simulations were run at different mesh sizes to determine which mesh size is appropriate to use. **Figure 2d** shows the mesh size chosen for all simulations. It is noteworthy that this mesh size shows an almost identical velocity contour to **Figure 2c** which proves grid independence was achieved.

Results and Discussion

Data was obtained in all sixteen trials in viscid and inviscid conditions in the ANSYS fluid software. Pressure drop, head loss, and power were calculated using Microsoft Excel.

Table 1. Calculations performed for pressure drop, head loss, and power for all sixteen trials.

Trial	length (m)	Obstacle shape	Diameter (m	Fluid type	velocity (m/s)	Renyolds number	Density (kg/m^3)	relative pipe roughness	Friction facto	Head loss	Pressure Drop	Area (m^2)	Power (W)	
1	8	Circle	1	Water	0.0001	611	997	2.50E-06	0.10467779	4.26821E-10	4.17455E-06	7.60730092	3.17571E-09	gravity
2	8	Circle	1	octane	0.0001	741	703	2.50E-06	0.08636977	3.5217E-10	2.42872E-06	7.60730092	1.8476E-09	9.81
3	8	Circle	1	Water	0.004	24457	997	2.50E-06	0.026	1.69623E-07	0.001659008	7.60730092	5.04823E-05	surface roughness (m)
4	8	Circle	1	octane	0.004	29640	703	2.50E-06	0.0245	1.59837E-07	0.001102304	7.60730092	3.35422E-05	2.50E-06
5	8	Circle	2	Water	0.0001	611.4	997	1.25E-06	0.10467779	2.1341E-10	2.08728E-06	14.4292037	3.01177E-09	
6	8	Circle	2	octane	0.0001	741	703	1.25E-06	0.08636977	1.76085E-10	1.21436E-06	14.4292037	1.75222E-09	
7	8	Circle	2	Water	0.004	24457	997	1.25E-06	0.026	8.48114E-08	0.000829504	14.4292037	4.78763E-05	
8	8	Circle	2	octane	0.004	29640	703	1.25E-06	0.0245	7.99185E-08	0.000551152	14.4292037	3.18107E-05	
9	8	Triangle	1	Water	0.0001	611.4	997	2.50E-06	0.10467779	4.26821E-10	4.17455E-06	7.75	3.23528E-09	
10	8	Triangle	1	octane	0.0001	741	703	2.50E-06	0.08636977	3.5217E-10	2.42872E-06	7.75	1.88226E-09	
11	8	Triangle	1	Water	0.004	24457	997	2.50E-06	0.026	1.69623E-07	0.001659008	7.75	5.14292E-05	
12	8	Triangle	1	octane	0.004	29640	703	2.50E-06	0.0245	1.59837E-07	0.001102304	7.75	3.41714E-05	
13	8	Triangle	2	Water	0.0001	611.4	997	1.25E-06	0.10467779	2.1341E-10	2.08728E-06	15.75	3.28746E-09	
14	8	Triangle	2	octane	0.0001	741	703	1.25E-06	0.08636977	1.76085E-10	1.21436E-06	15.75	1.91262E-09	
15	8	Triangle	2	Water	0.004	24457	997	1.25E-06	0.026	8.48114E-08	0.000829504	15.75	5.22588E-05	
16	8	Triangle	2	octane	0.004	29640	703	1.25E-06	0.0245	7.99185E-08	0.000551152	15.75	3.47226E-05	

Anthony Gisolfi
EMCH 360
12/7/2022

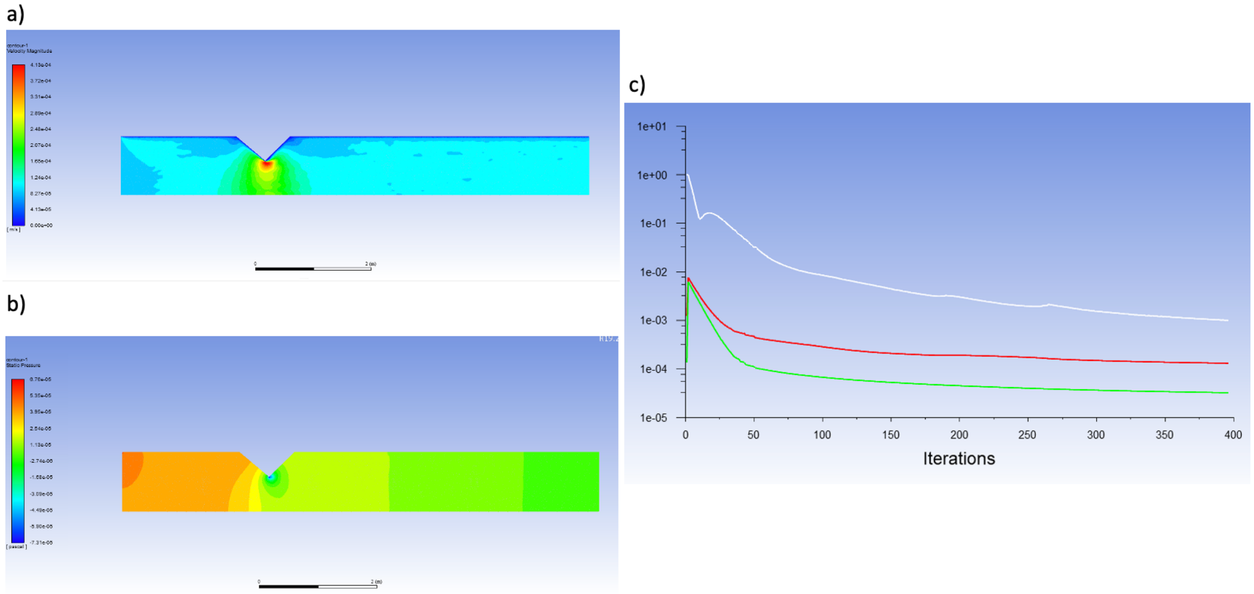


Figure 3. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

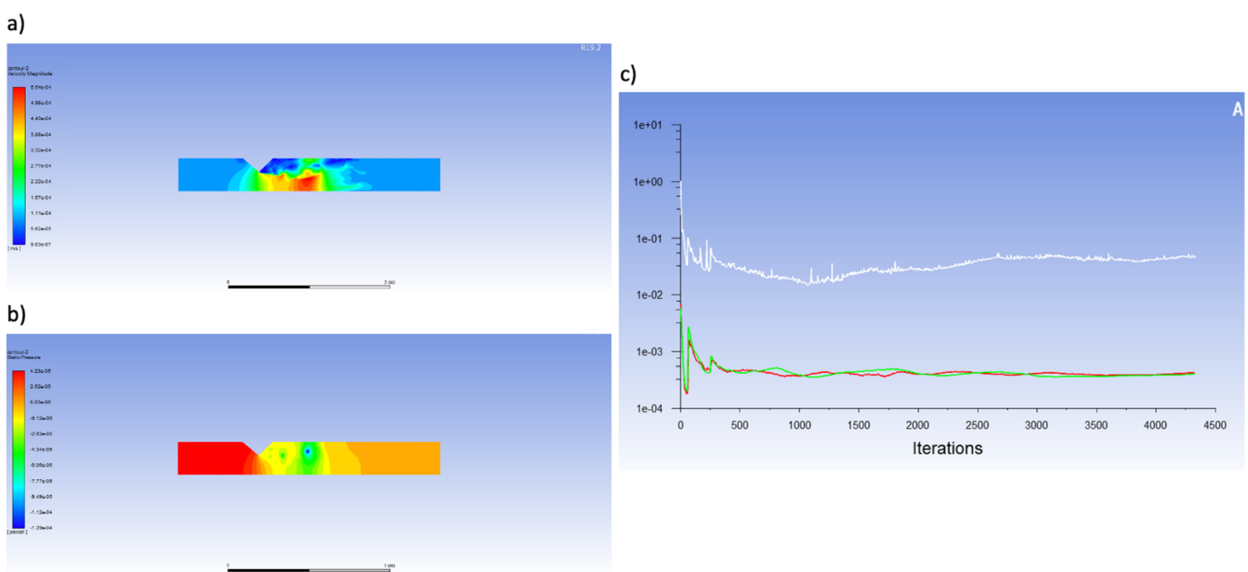


Figure 4. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s.

(B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the first simulation, represented by **Figure 3 and Figure 4**, a 1m long triangle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime is 1.88226 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 4** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the octane after the obstacle than the viscid model. The Reynolds number of this flow was found to be 741 and the Darcy friction factor was found to be 0.0864. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

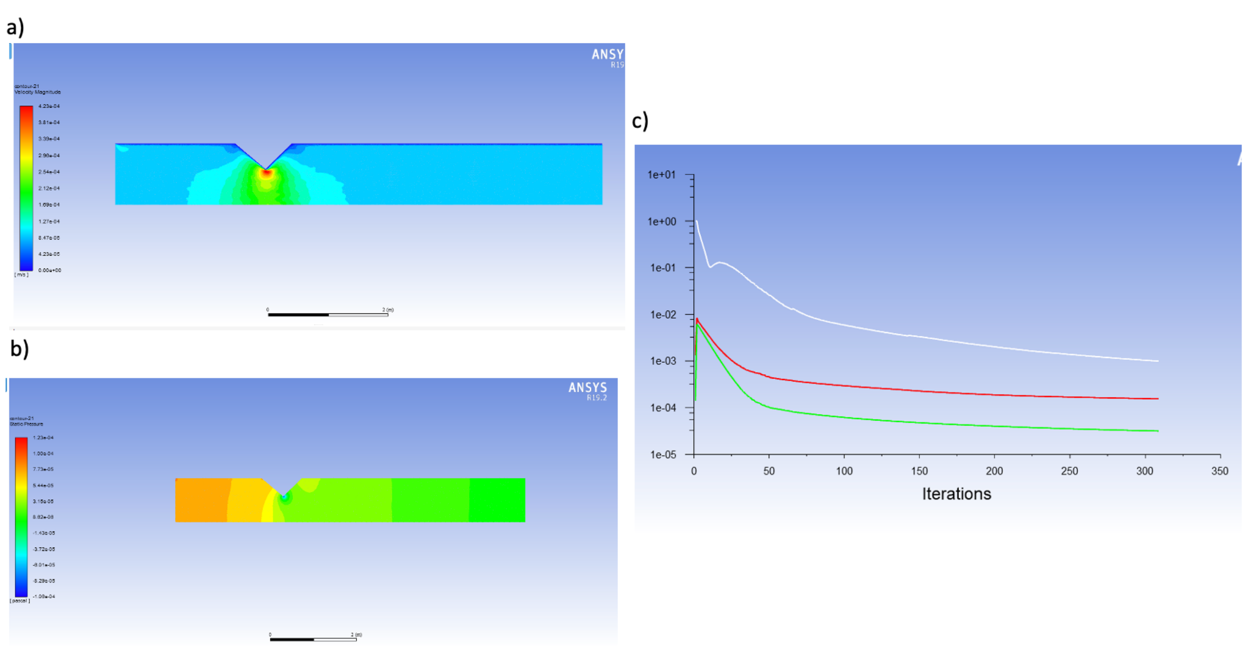


Figure 5. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid water flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the

pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

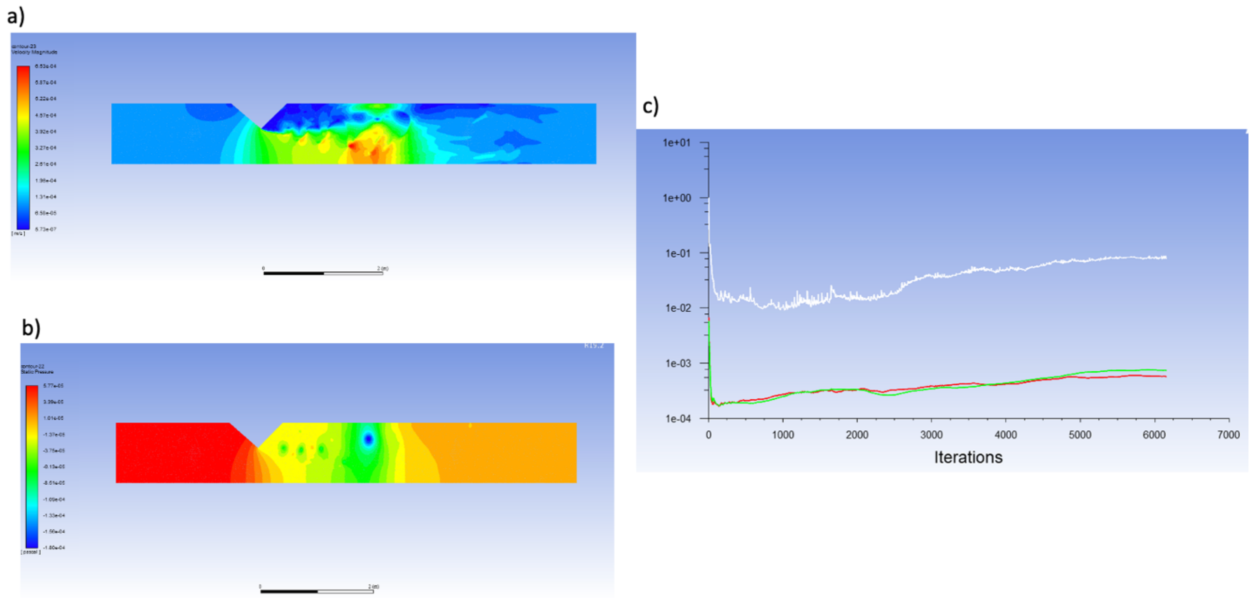


Figure 6. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid water flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the second simulation, represented by **Figure 5 and Figure 6**, a 1m long triangle obstacle was placed in a pipe with liquid water. The water flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime is 3.23528 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 6** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 611.4 and the Darcy friction factor was found to be 0.104677789. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

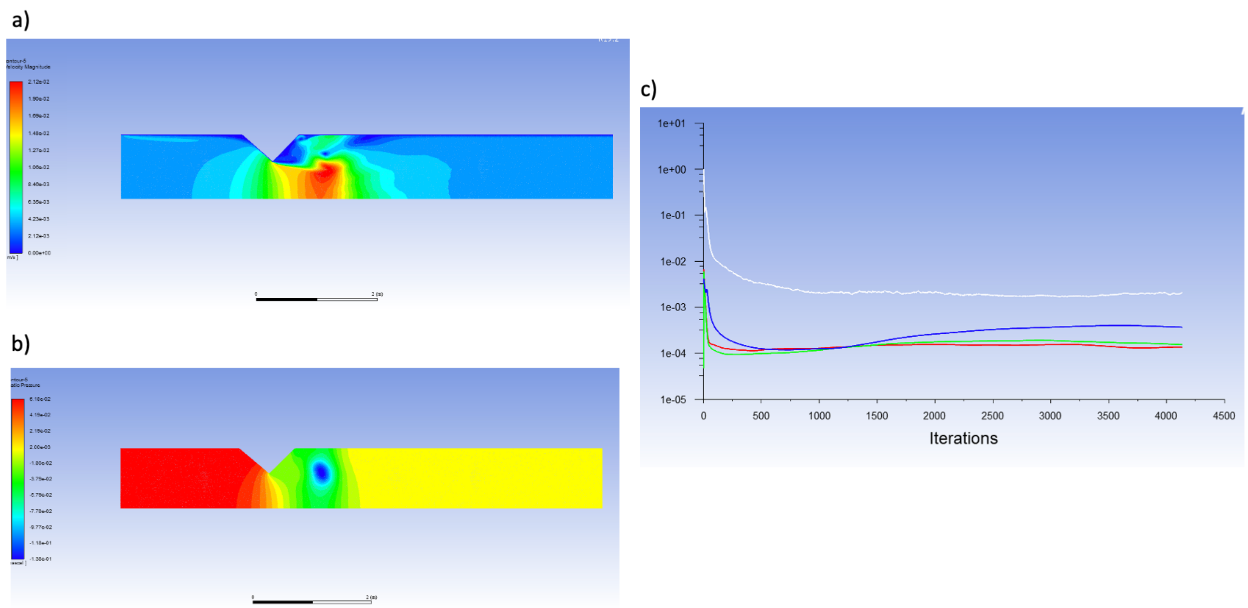


Figure 7. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid octane flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

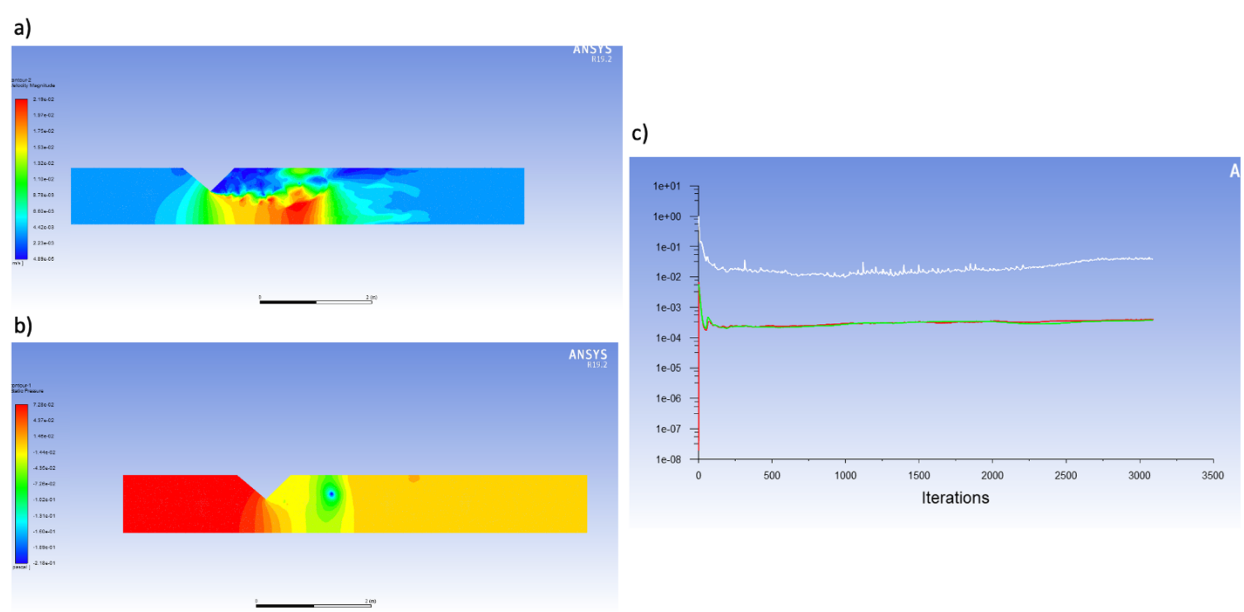


Figure 8. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid octane flows through turbulently at 0.004 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the third simulation, represented by **Figure 7 and Figure 8**, a 1m long triangle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0341714 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 8** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 29640 and the Darcy friction factor was found to be 0.0245. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

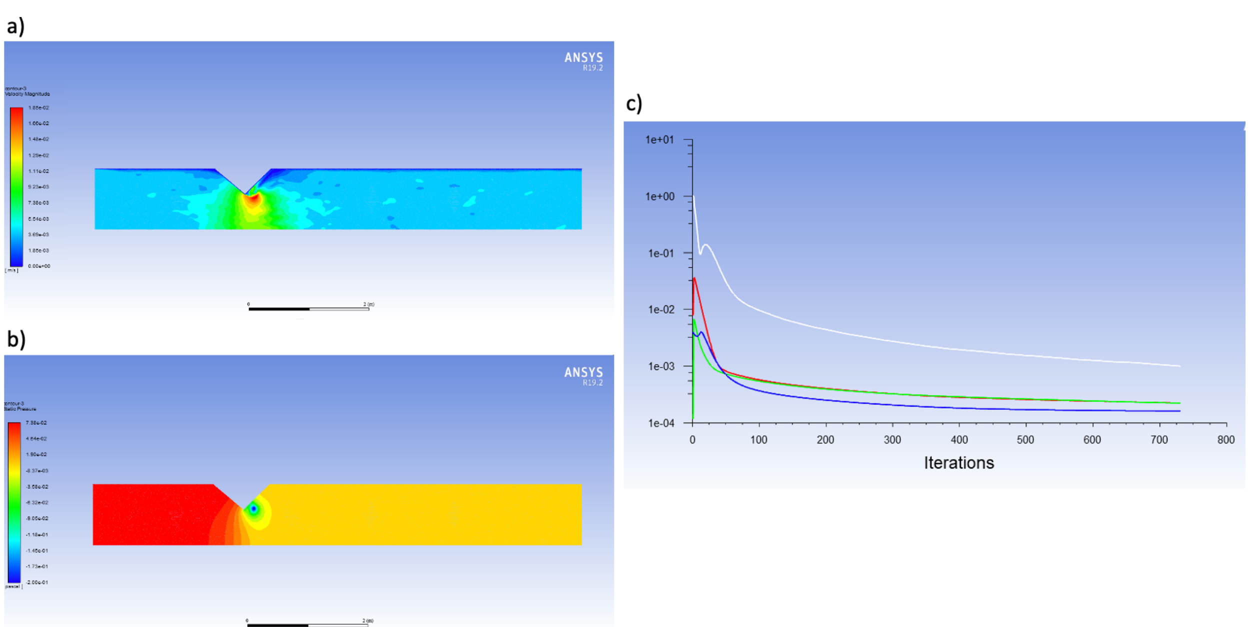


Figure 9. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid water flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (c) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

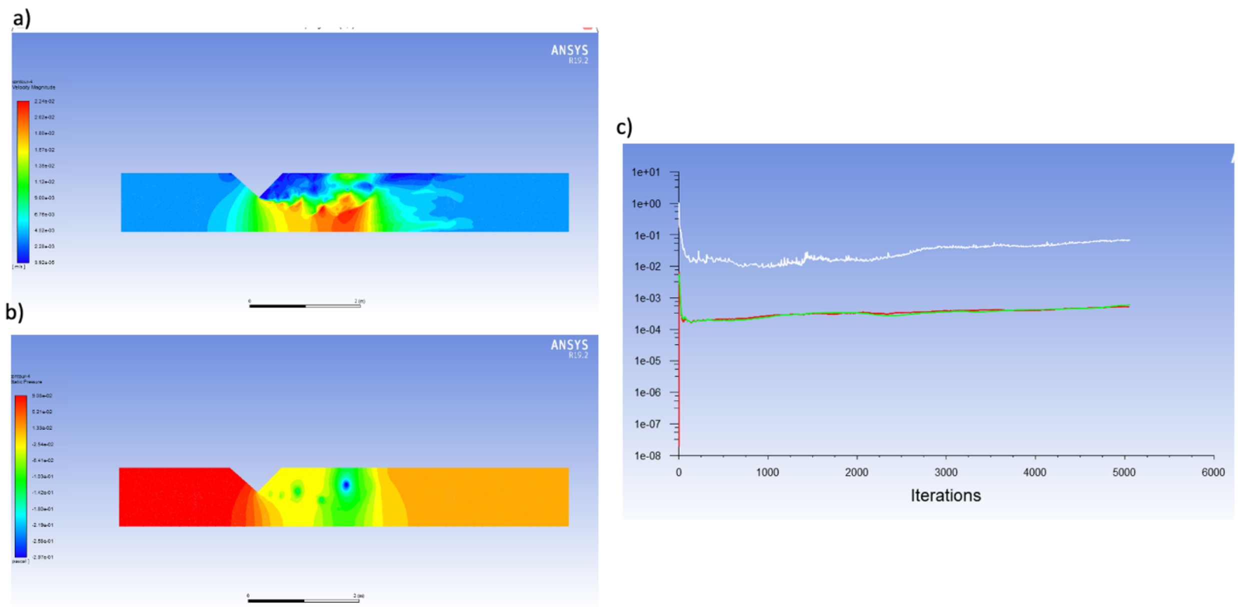


Figure 10. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid water flows through turbulently at 0.004 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the fourth simulation, represented by **Figure 9 and Figure 10**, a 1m long triangle obstacle was placed in a pipe with liquid water. The liquid water flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0514292 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent **Figure 10** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 24457 and the Darcy friction factor was found to be 0.026. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

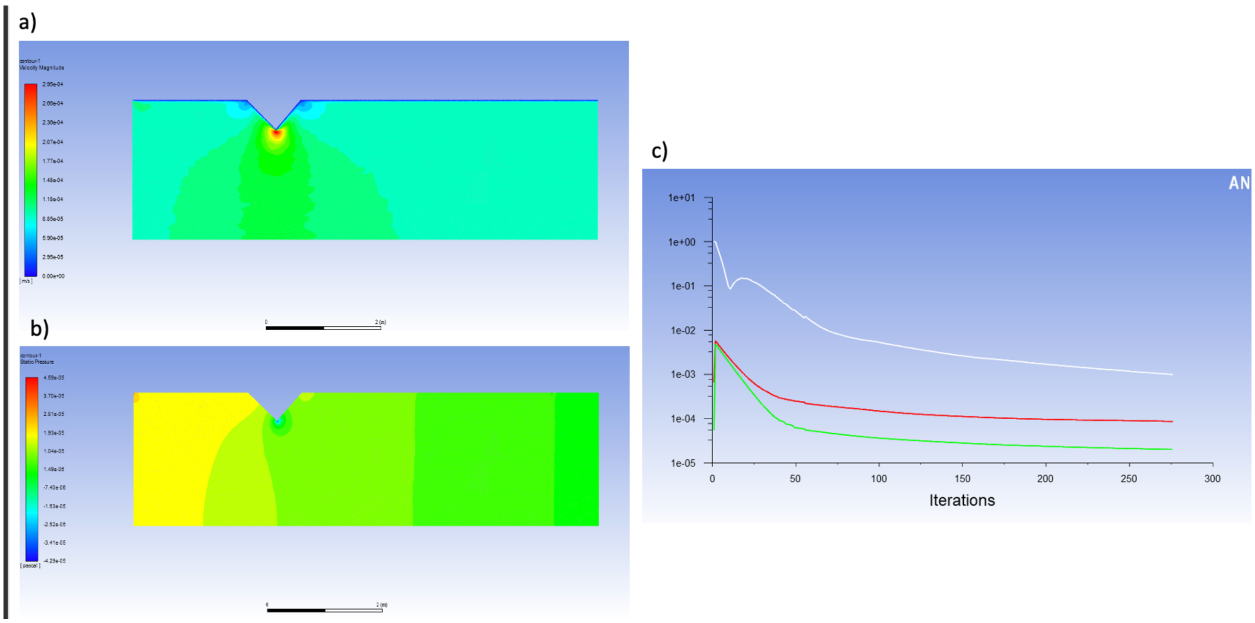


Figure 11. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

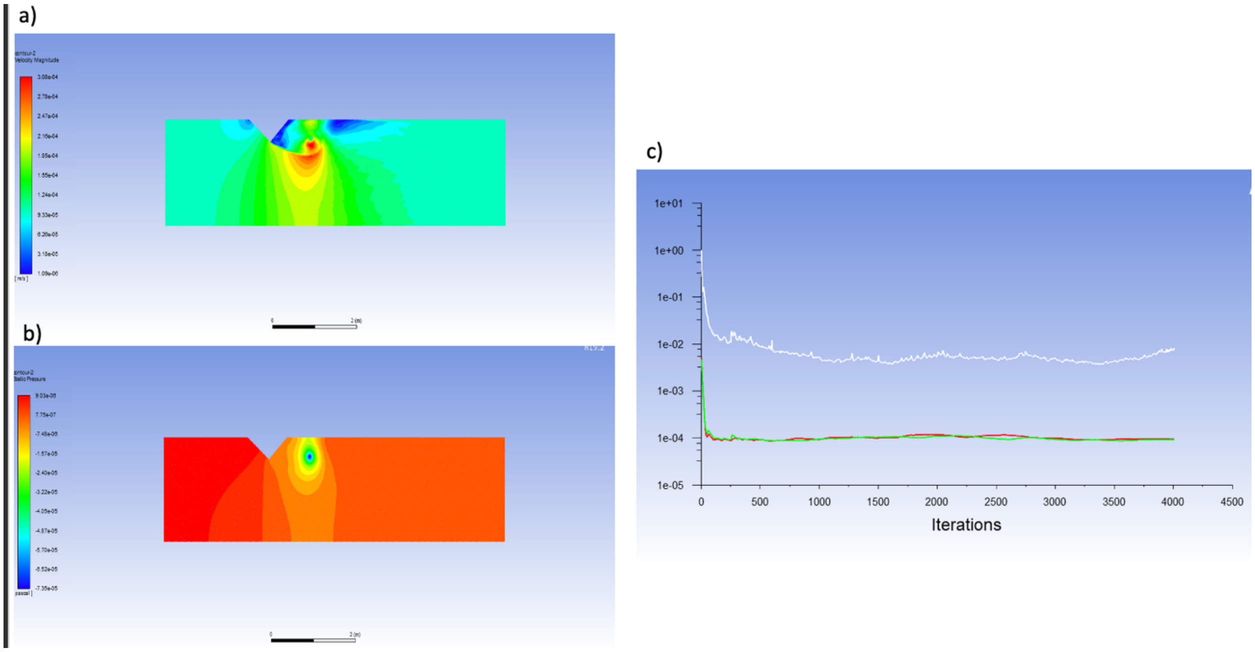


Figure 12. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid octane flows through turbulently at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the fifth simulation, represented by **Figure 11 and Figure 12**, a 2m long triangle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime 1.91262 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 12** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 741 and the Darcy friction factor was found to be 0.086369771. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

Anthony Gisolfi
EMCH 360
12/7/2022

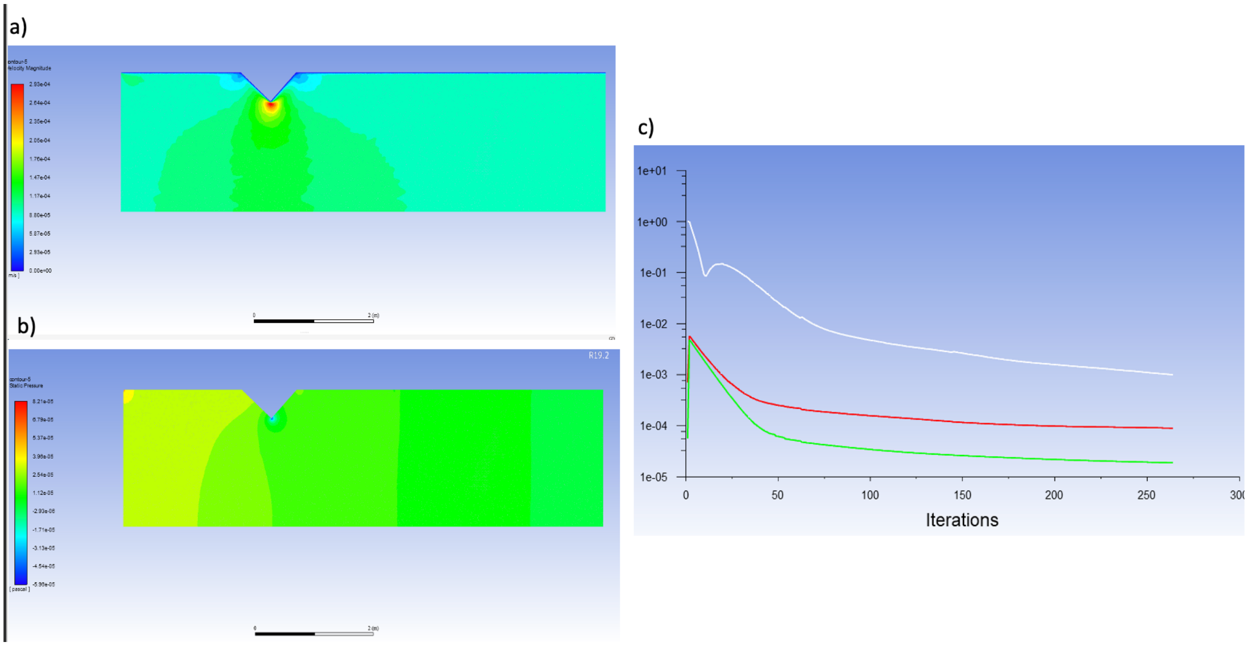


Figure 13. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid water flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

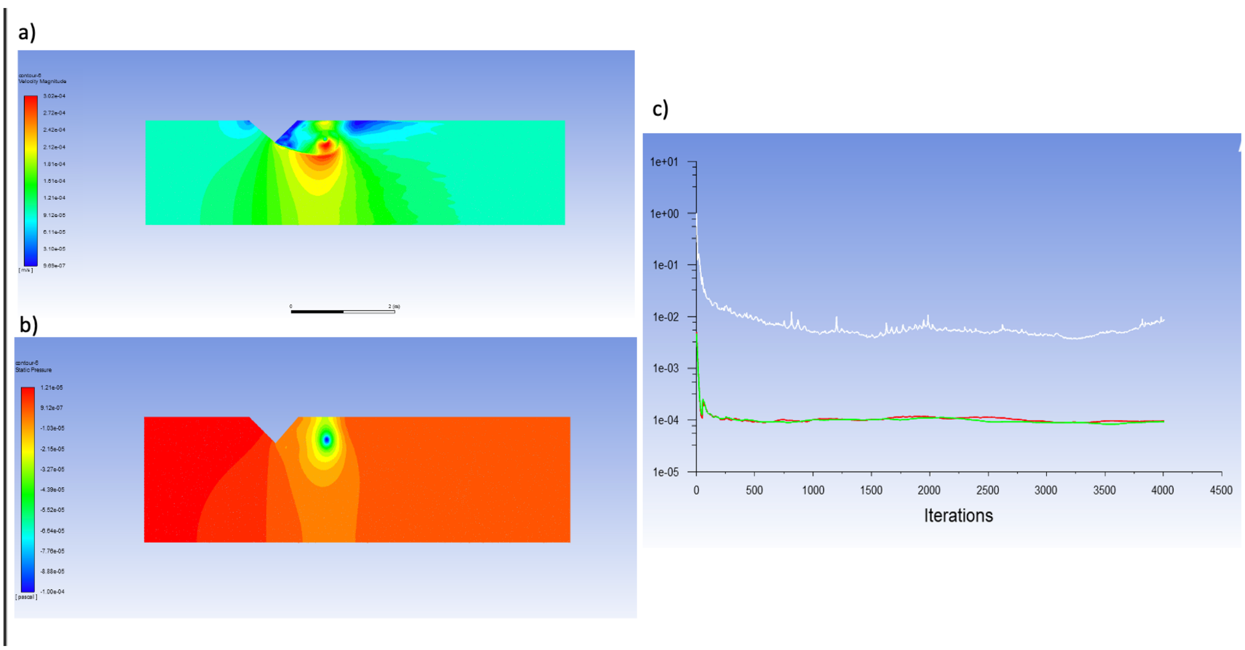


Figure 14. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid water flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s.

(B) Shows the pressure contour of the flow in pascals. (c) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the sixth simulation, represented by **Figure 13 and Figure 14**, a 2m long triangle obstacle was placed in a pipe with liquid water. The water flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime 3.28746 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 14** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 611.4 and the Darcy friction factor was found to be 0.104677789. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

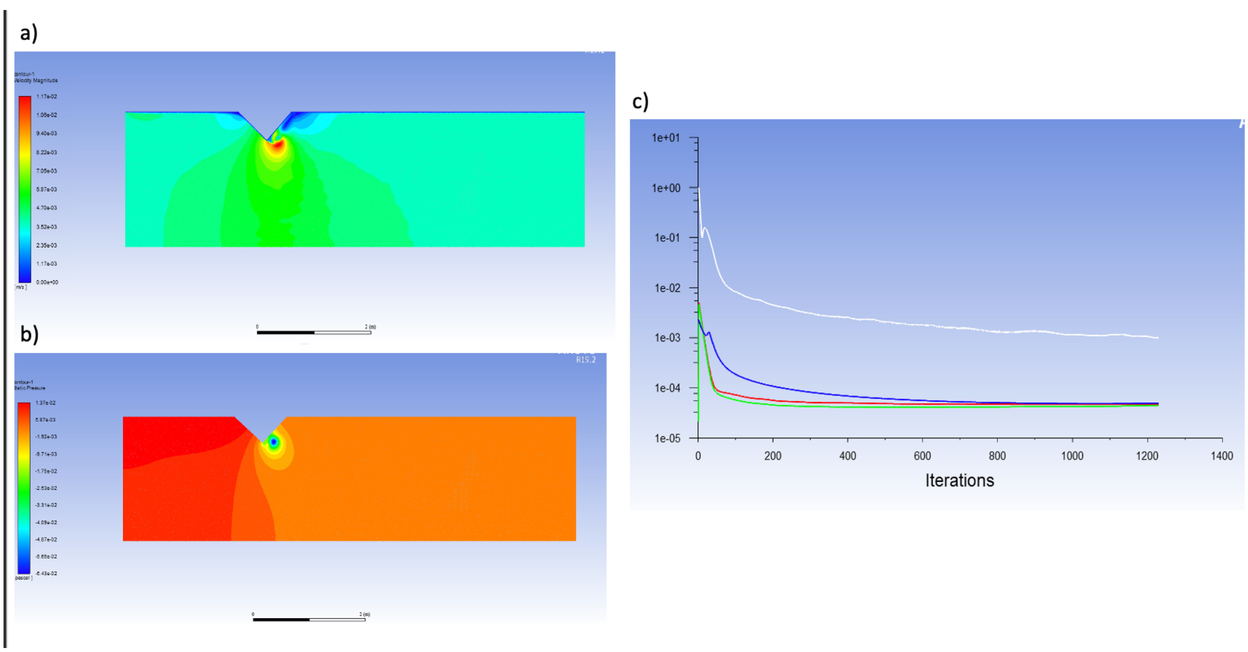


Figure 15. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid octane flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the

pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

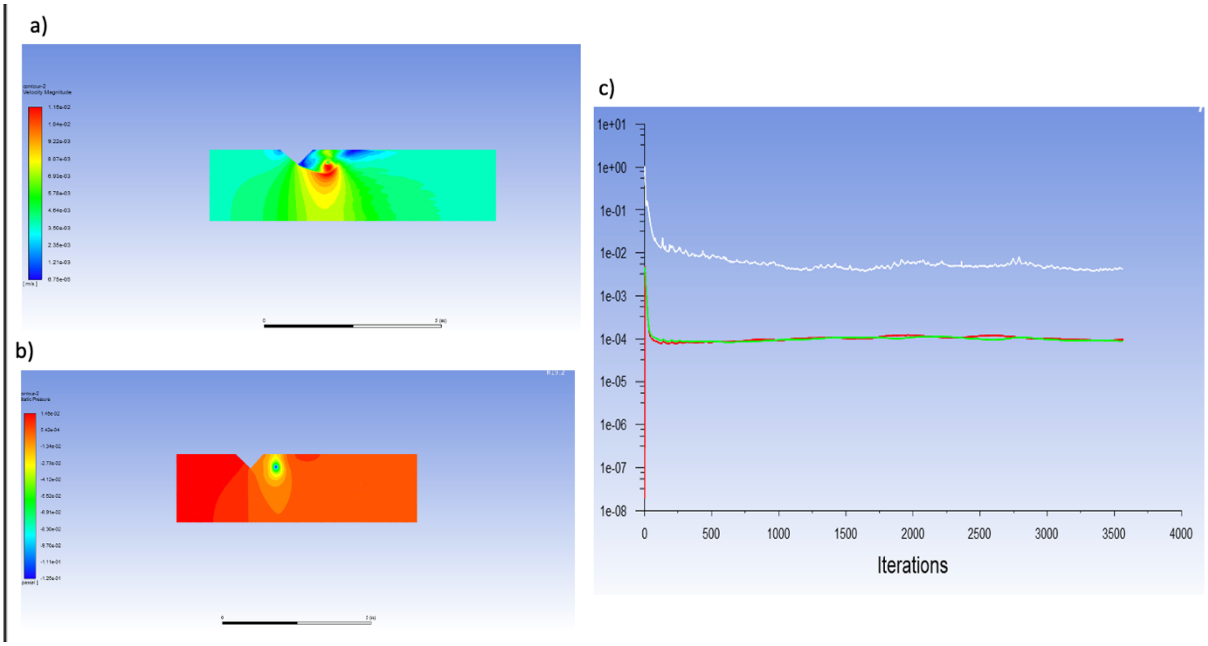


Figure 16. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid octane flows through turbulently at 0.004 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the seventh simulation, represented by **Figure 15 and Figure 16**, a 2m long triangle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0347226 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent**. Figure 16** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 29640 and the Darcy friction factor was found to be 0.0245. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

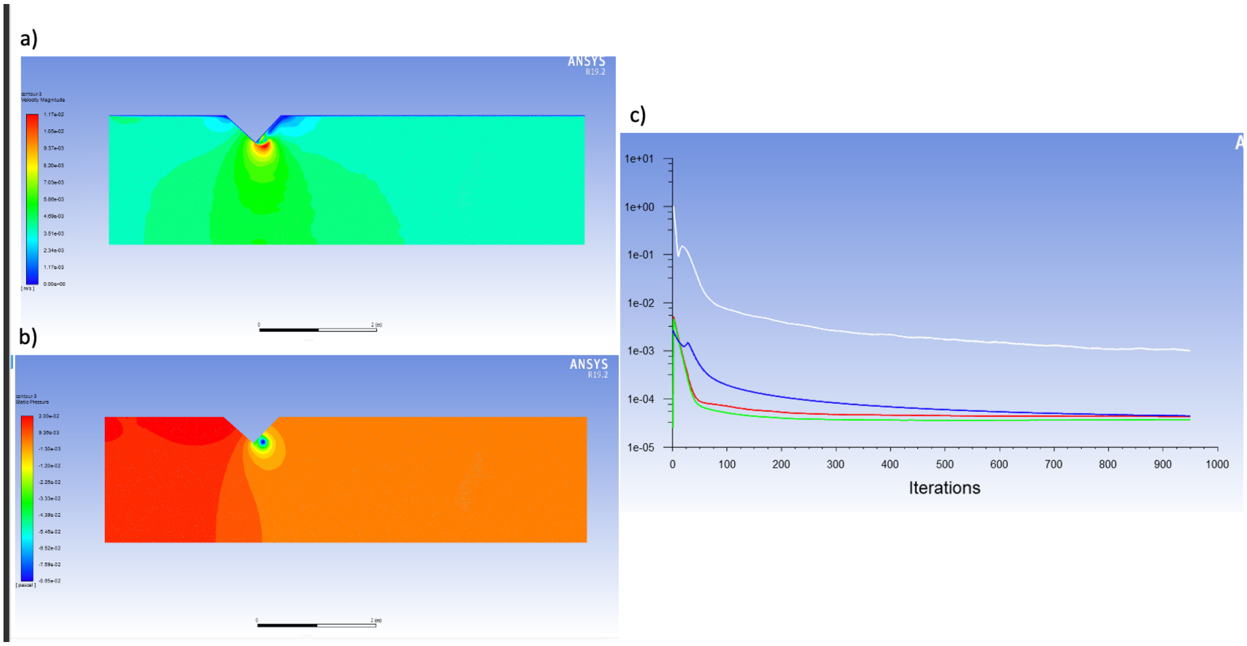


Figure 17. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid water flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

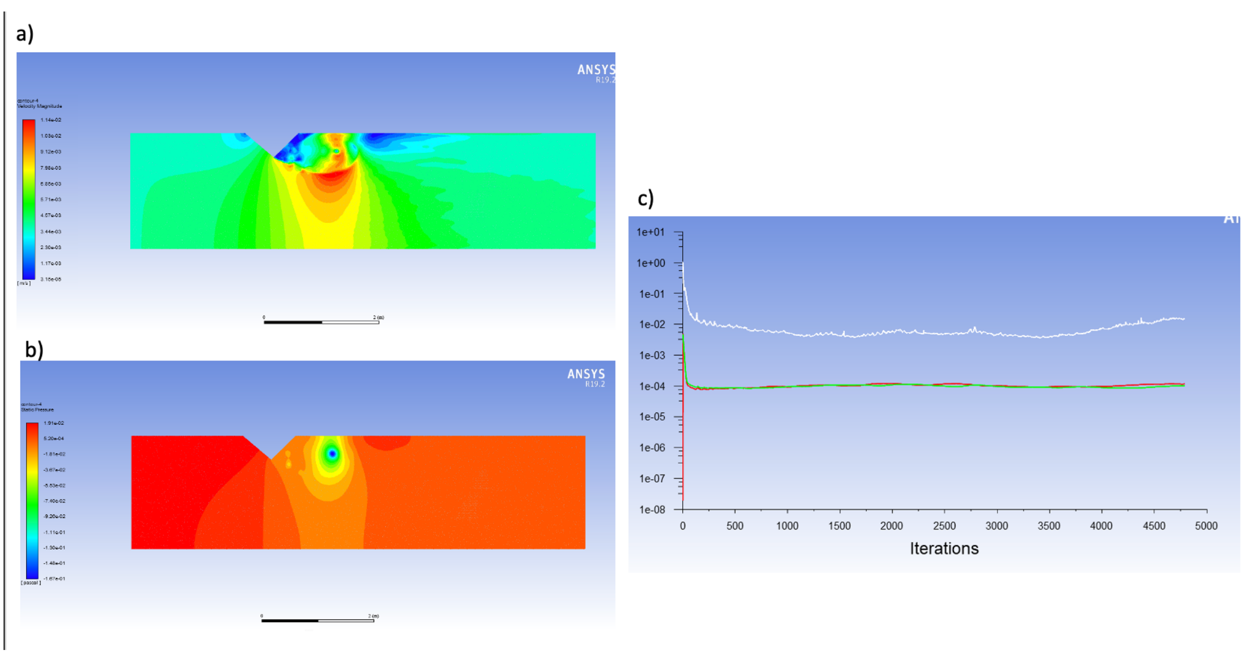


Figure 18. This pipe simulation involves a 2 meter diameter pipe with a triangular obstacle. Liquid water flows through turbulently at 0.004 (m/s) with inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the eighth simulation, represented by **Figure 17 and Figure 18**, a 2m long triangle obstacle was placed in a pipe with liquid water. The liquid water flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0522588 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 18** represents the simulation modelled at inviscid conditions, which more accurately depicts the velocity and pressure of the water after the obstacle than the viscid model. The Reynolds number of this flow was found to be 24457 and the Darcy friction factor was found to be 0.026. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

Anthony Gisolfi
EMCH 360
12/7/2022

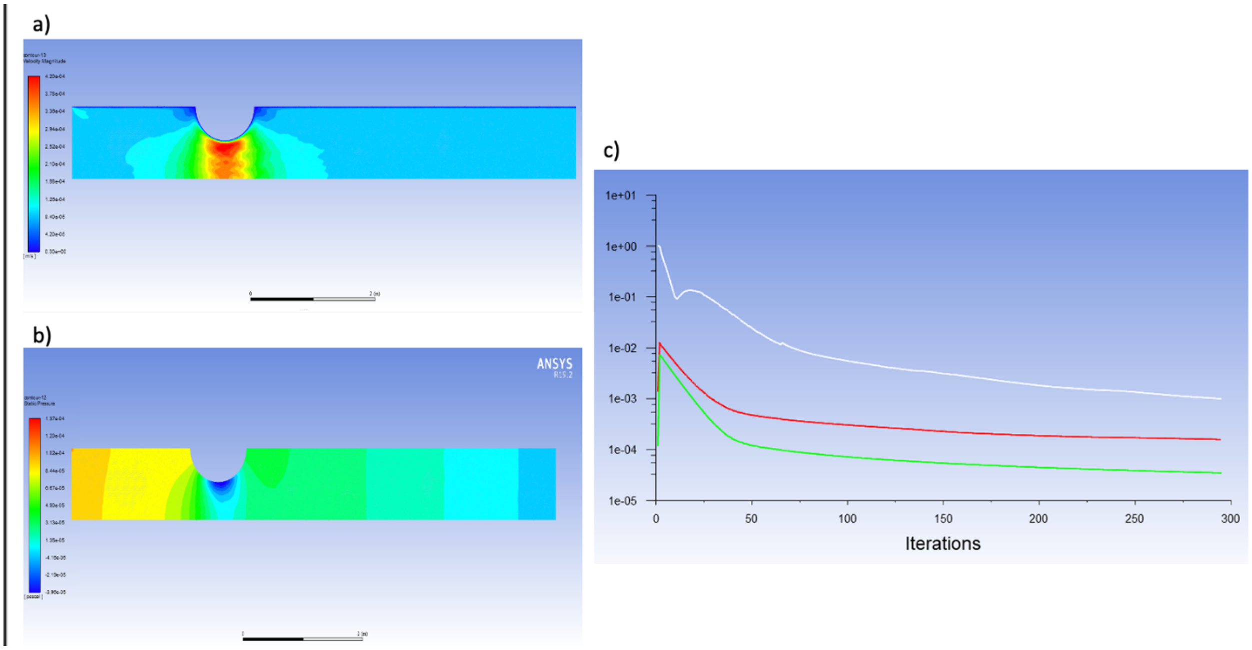
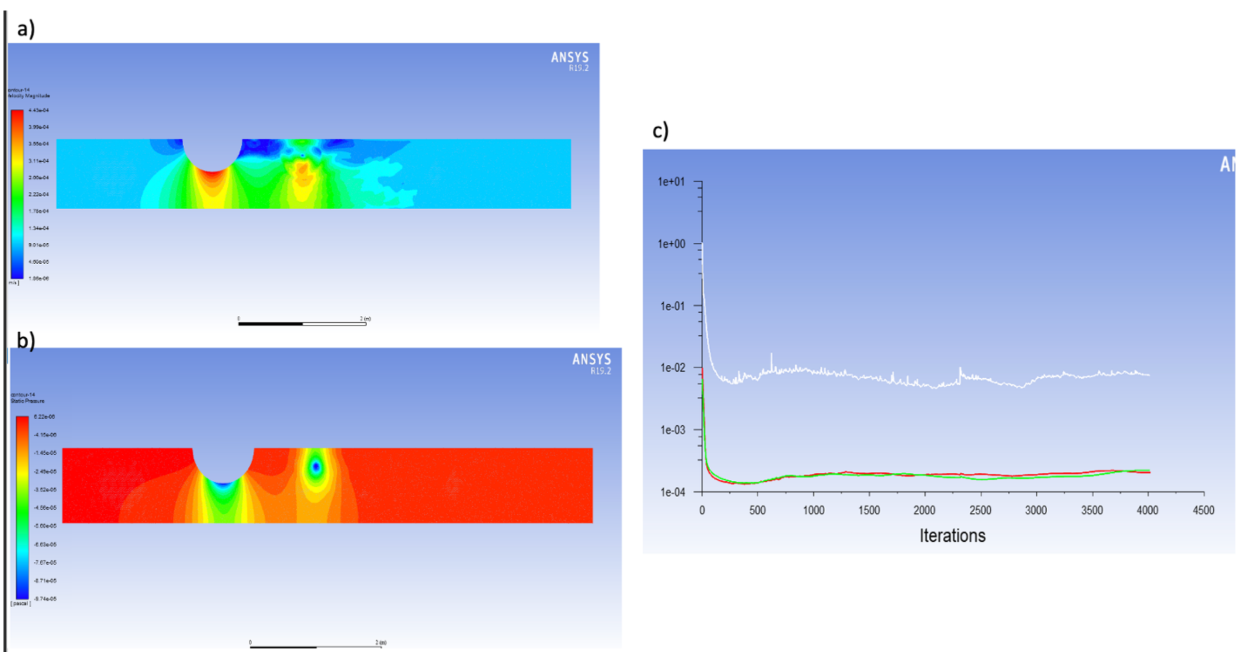


Figure 19. This pipe simulation involves a 1 meter diameter pipe with a triangular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.



Anthony Gisolfi
EMCH 360
12/7/2022

Figure 20. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the ninth simulation, represented by **Figure 19 and Figure 20**, a 1m diameter circle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime 1.8476 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 20** represents the simulation modelled at inviscid conditions, which more accurately depicts the pressure of the water after the obstacle than the viscid model. The velocity contours show little to no difference in the viscid and inviscid models. The Reynolds number of this flow was found to be 741 and the Darcy friction factor was found to be 0.086369771. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

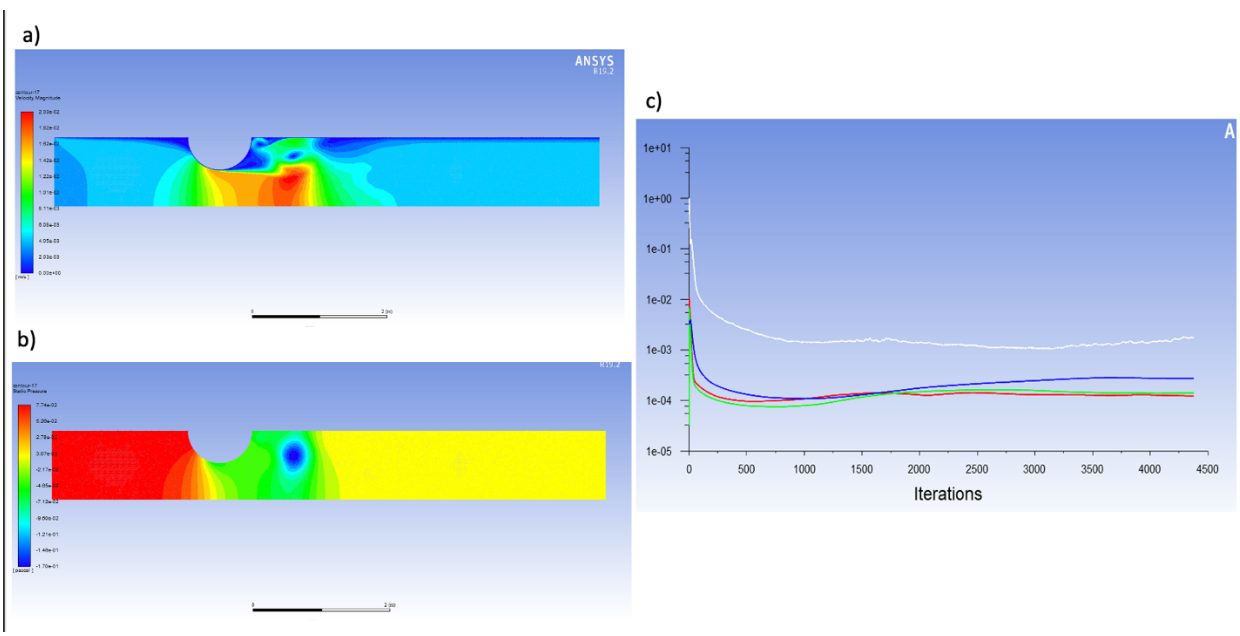


Figure 21. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid octane flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

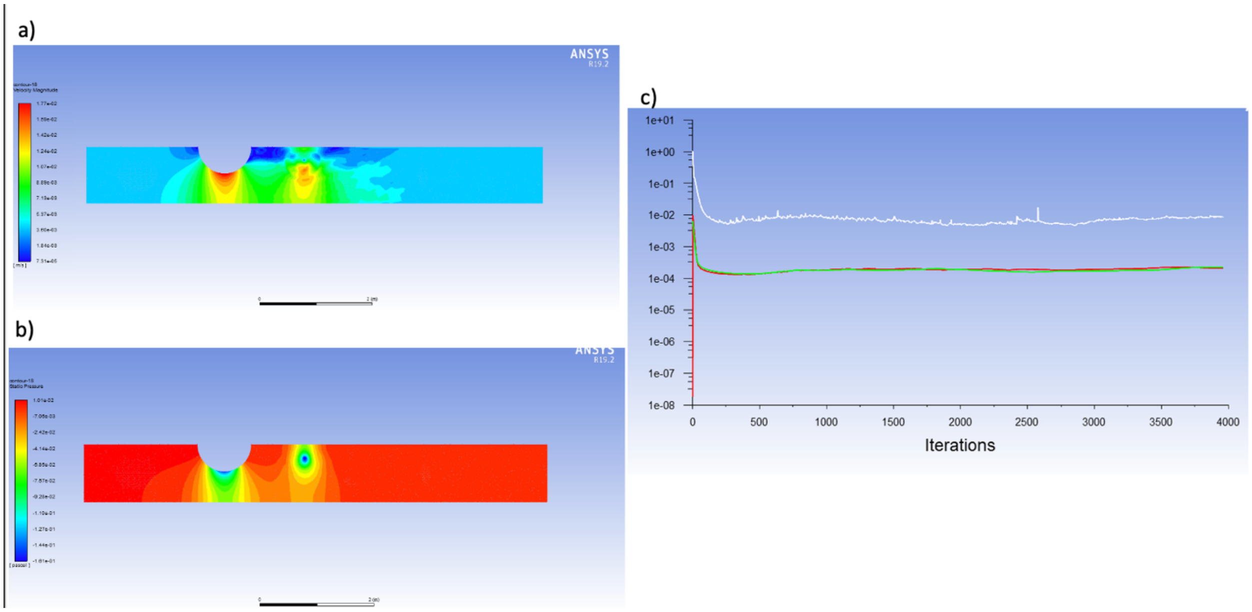


Figure 22. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid octane flows through turbulently at 0.004 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the tenth simulation, represented by **Figure 21 and Figure 22**, a 1m diameter circle obstacle was placed in a pipe with liquid octane. The octane flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0335422 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 22** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 29640 and the Darcy friction factor was found to be 0.0245. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

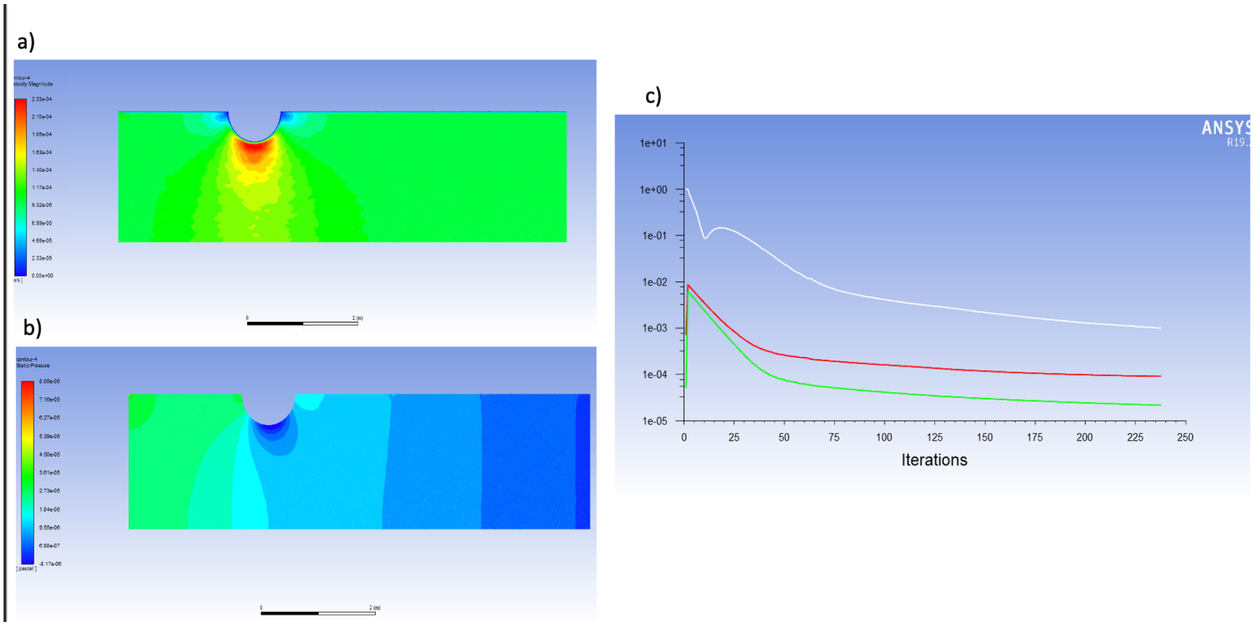


Figure 23. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

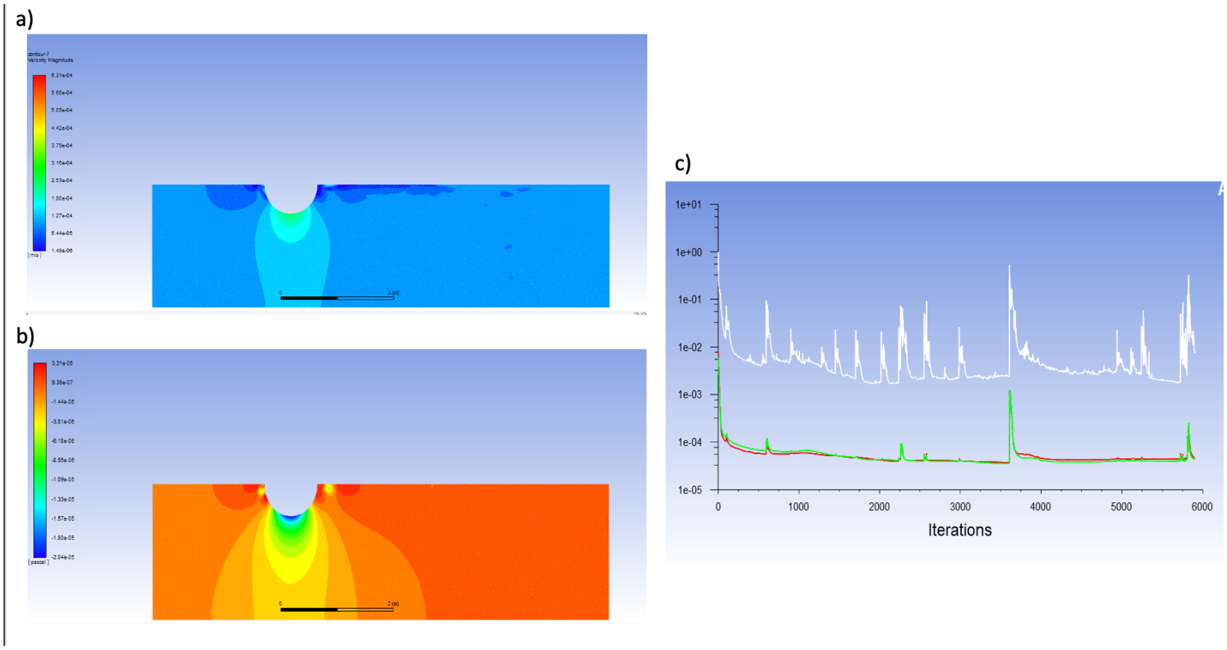


Figure 24. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid octane flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the eleventh simulation, represented by **Figure 23 and Figure 24**, a 2m diameter circle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime 1.75222 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 24** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 741 and the Darcy friction factor was found to be 0.086369771. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

Anthony Gisolfi
EMCH 360
12/7/2022

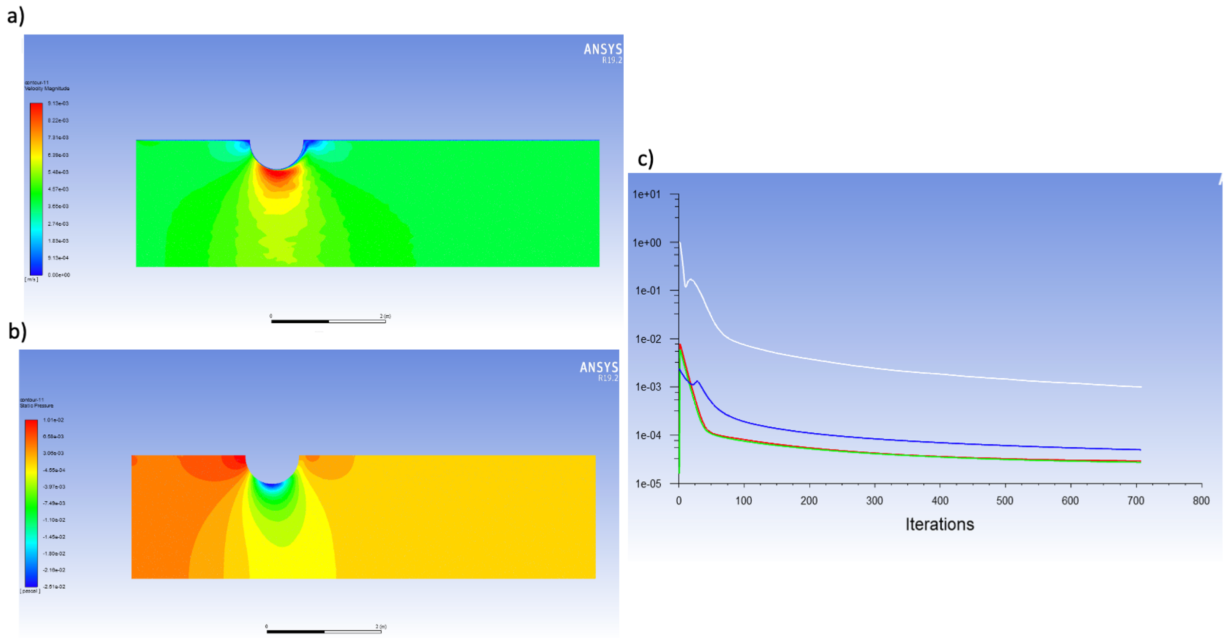


Figure 25. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid octane flows through turbulently at 0.0004(m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (c) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

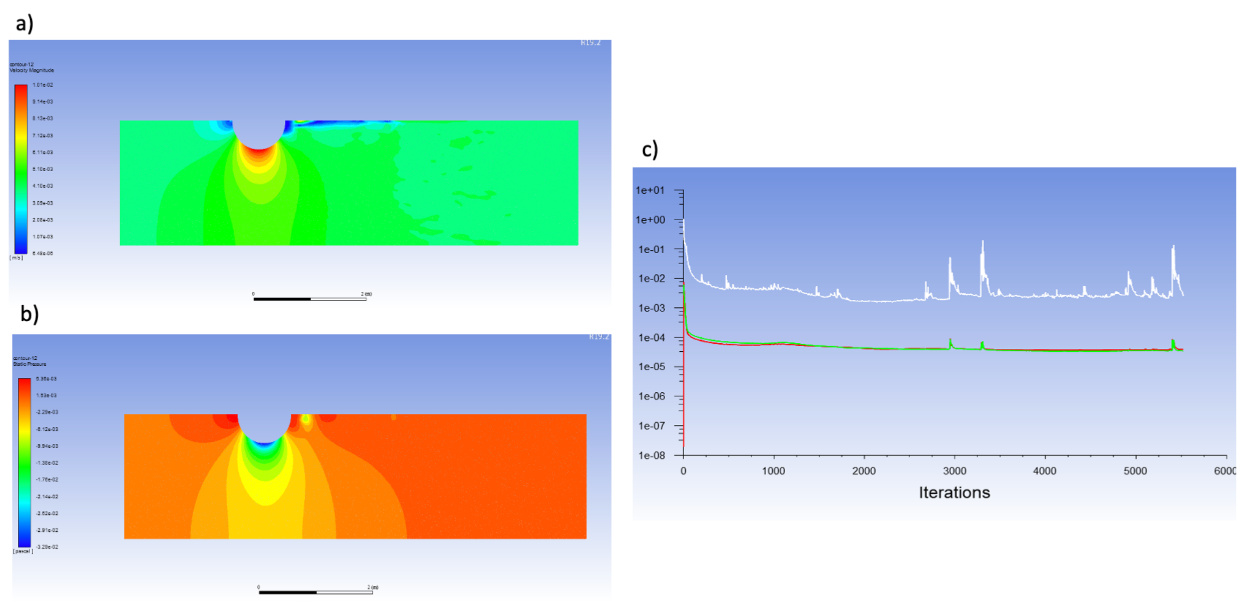


Figure 26. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid octane flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the twelfth simulation, represented by **Figure 25 and Figure 26**, a 2m diameter circle obstacle was placed in a pipe with liquid octane. The liquid octane flowed at a turbulent speed of 0.004 m/s. The power required for this flow regime 0.0318107 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 26** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 29640 and the Darcy friction factor was found to be 0.0245. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

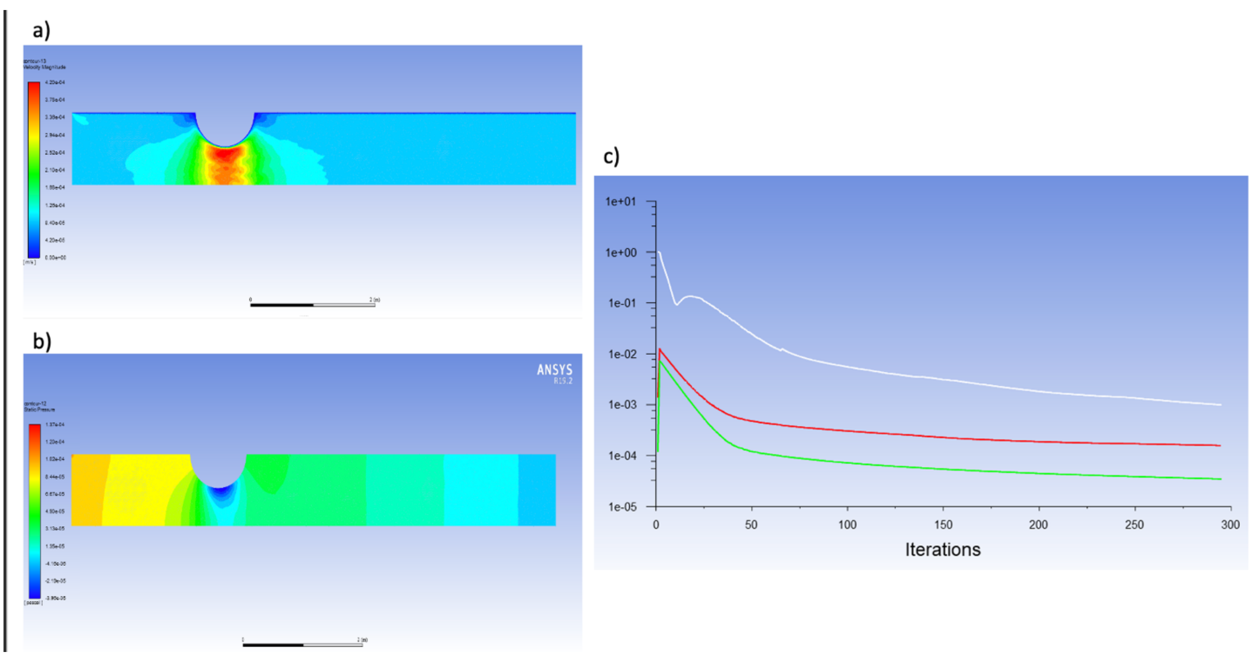


Figure 27. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid water flows through laminarly at 0.0001 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

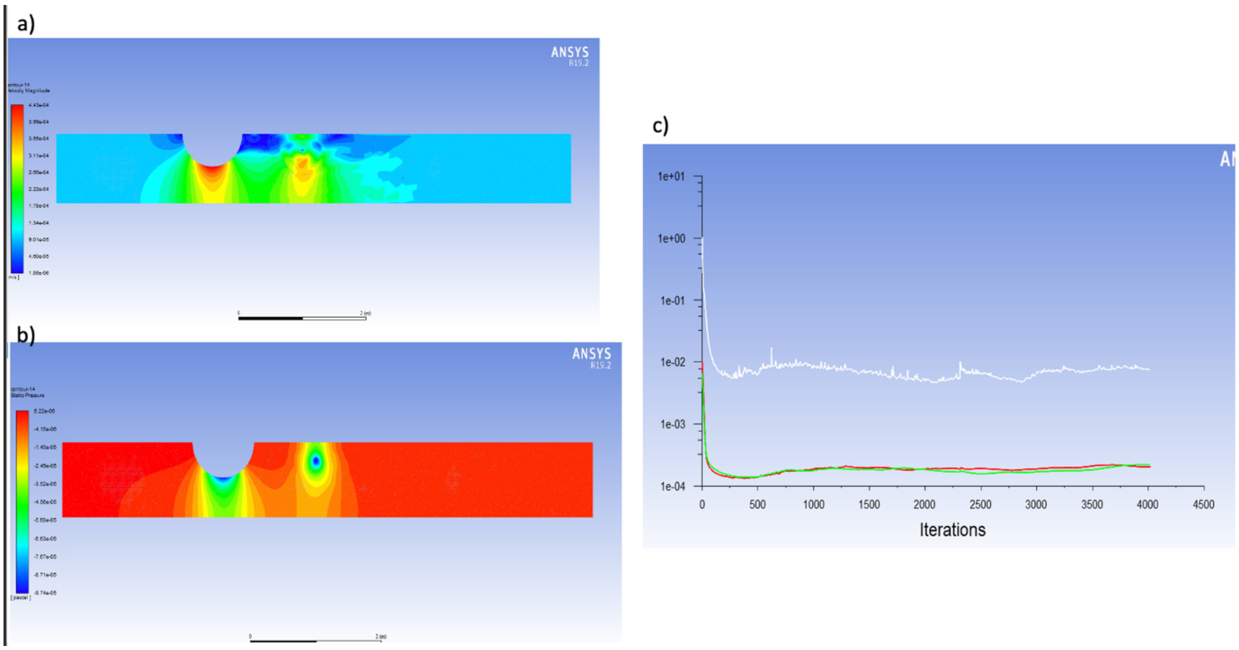


Figure 28. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid water flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the thirteenth simulation, represented by **Figure 27 and Figure 28**, a 1m diameter circle obstacle was placed in a pipe with liquid water. The water flowed at a laminarly speed of 0.0001 m/s. The power required for this flow regime 3.17571 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 28** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 611 and the Darcy friction factor was found to be 0.104677789. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

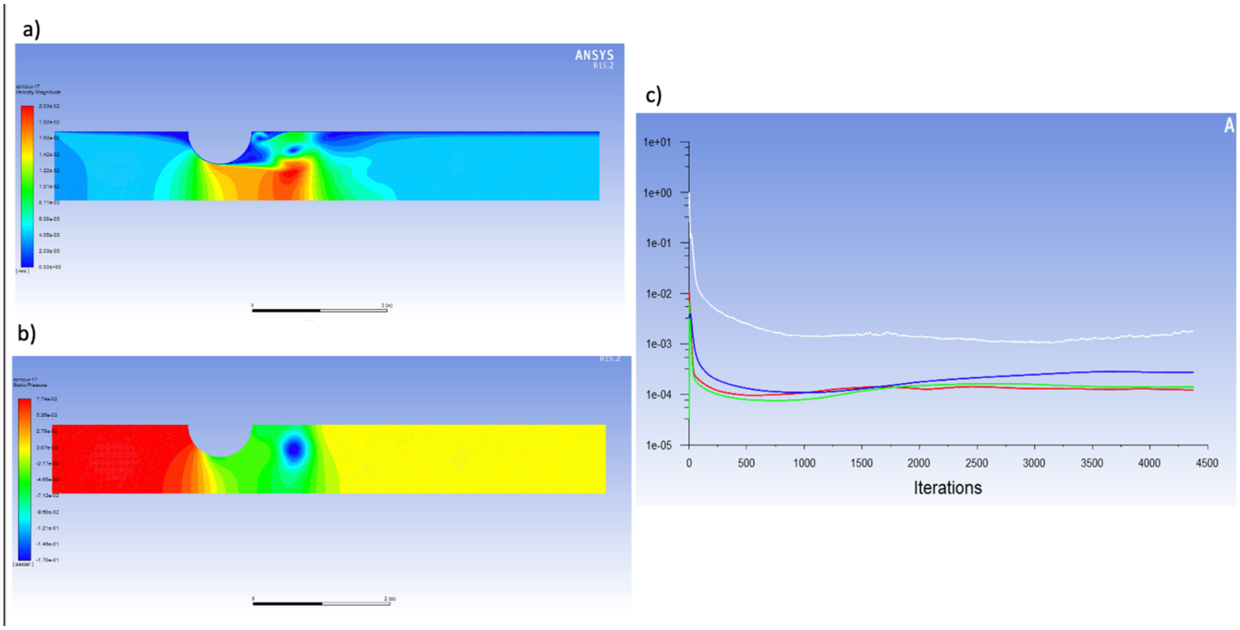


Figure 29. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid water flows through turbulently at 0.004 (m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines, and blue (nut) over an amount of iterations.

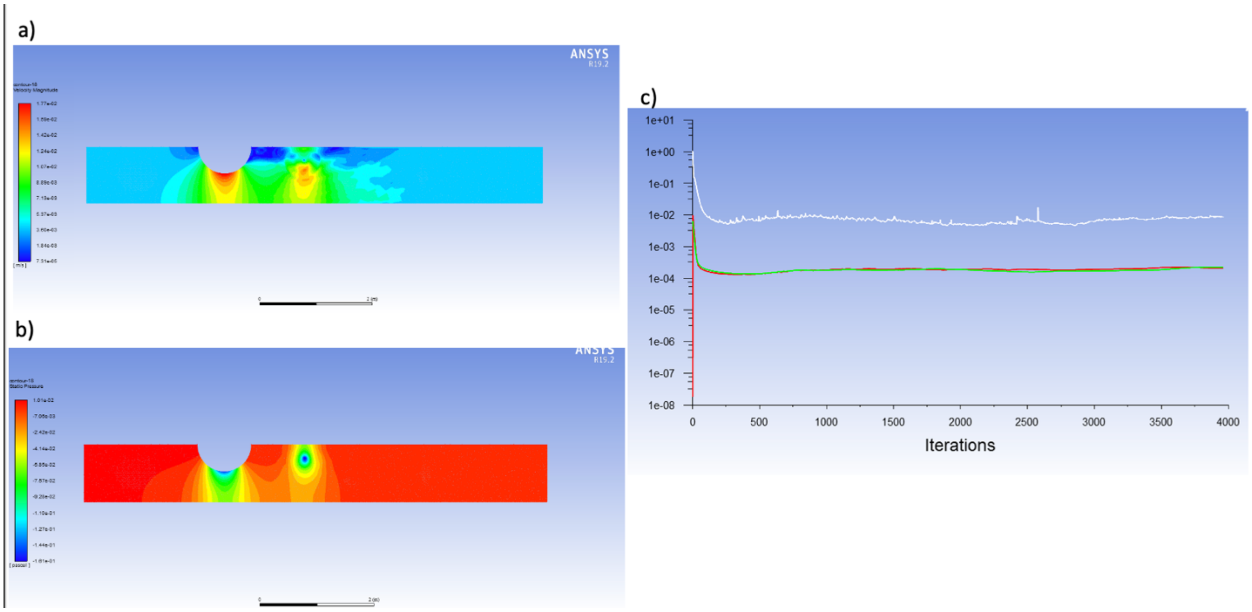


Figure 30. This pipe simulation involves a 1 meter diameter pipe with a circular obstacle. Liquid water flows through turbulently at 0.004 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the fourteenth simulation, represented by **Figure 29 and Figure 30**, a 1m diameter circle obstacle was placed in a pipe with liquid water. The water flowed at a turbulently speed of 0.004 m/s. The power required for this flow regime 0.0335422 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 30** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 24457 and the Darcy friction factor was found to be 0.026. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

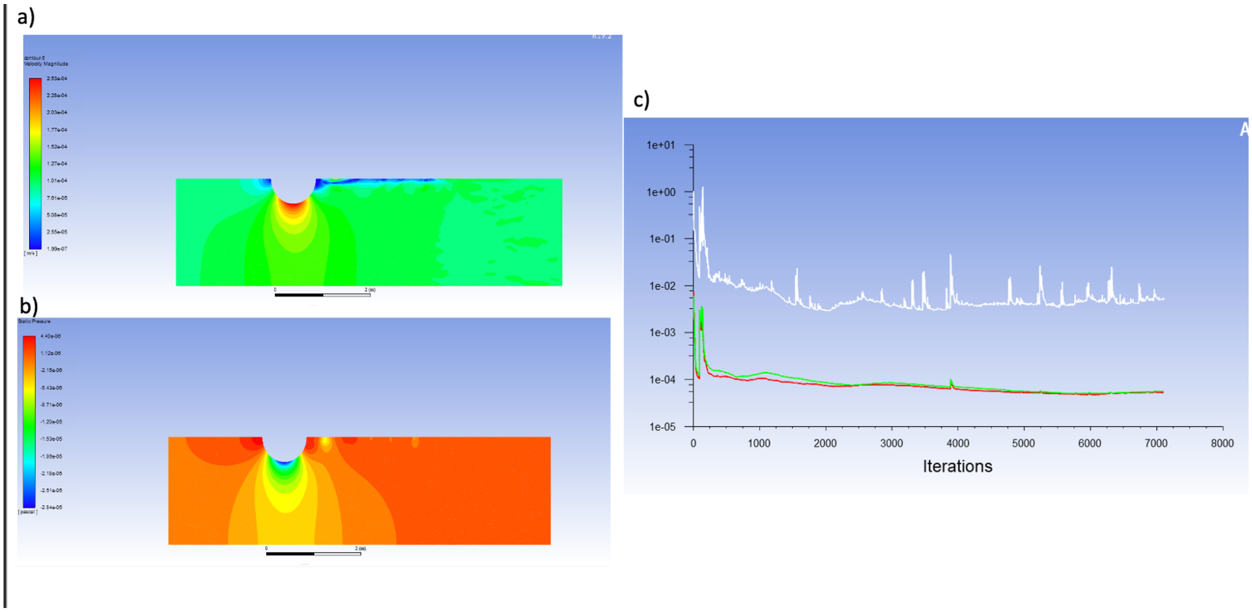


Figure 31. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid water flows through laminarly at 0.0001 (m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

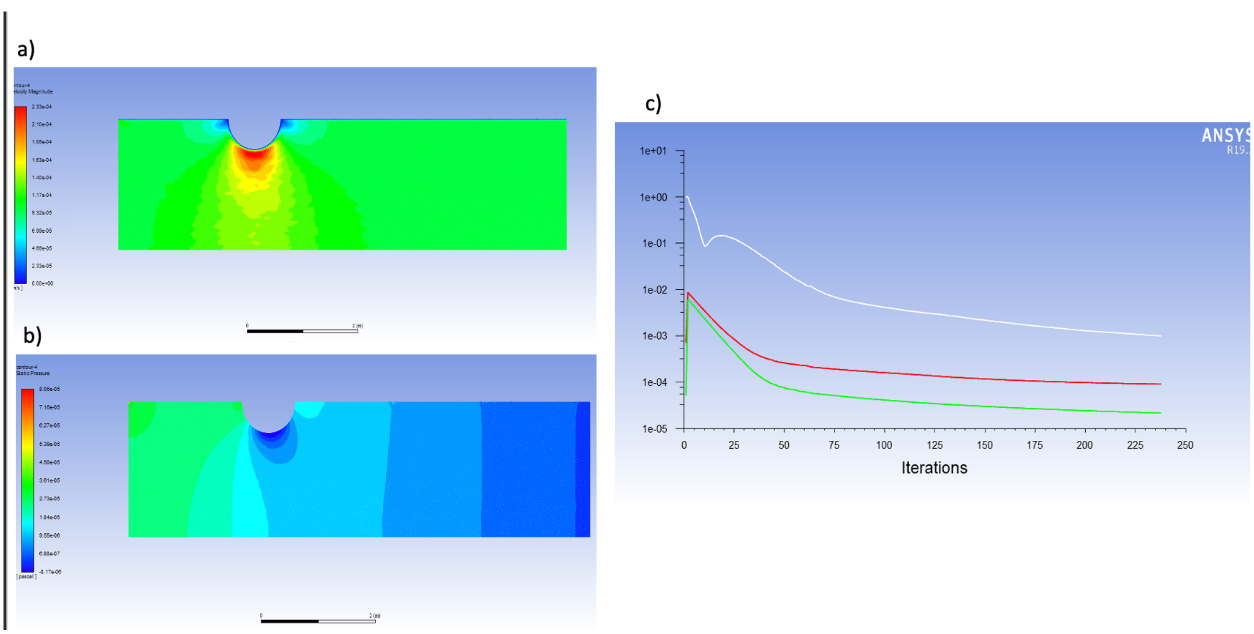


Figure 32. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid water flows through laminarly at 0.0001(m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour

of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

In the fifteenth simulation, represented by **Figure 31 and Figure 32**, a 2m diameter circle obstacle was placed in a pipe with liquid water. The liquid water flowed at a laminar speed of 0.0001 m/s. The power required for this flow regime 3.01177 nW, as seen in **Table 1**. This model should be treated as a steady state flow regime since the flow is not time dependent. **Figure 32** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 611.4 and the Darcy friction factor was found to be 0.10467779. In this scenario, there is a relatively high friction factor compared to the turbulent cases and consequently there is more pressure loss created due to friction. This pipe is more efficient than the turbulent scenario due to its low required power.

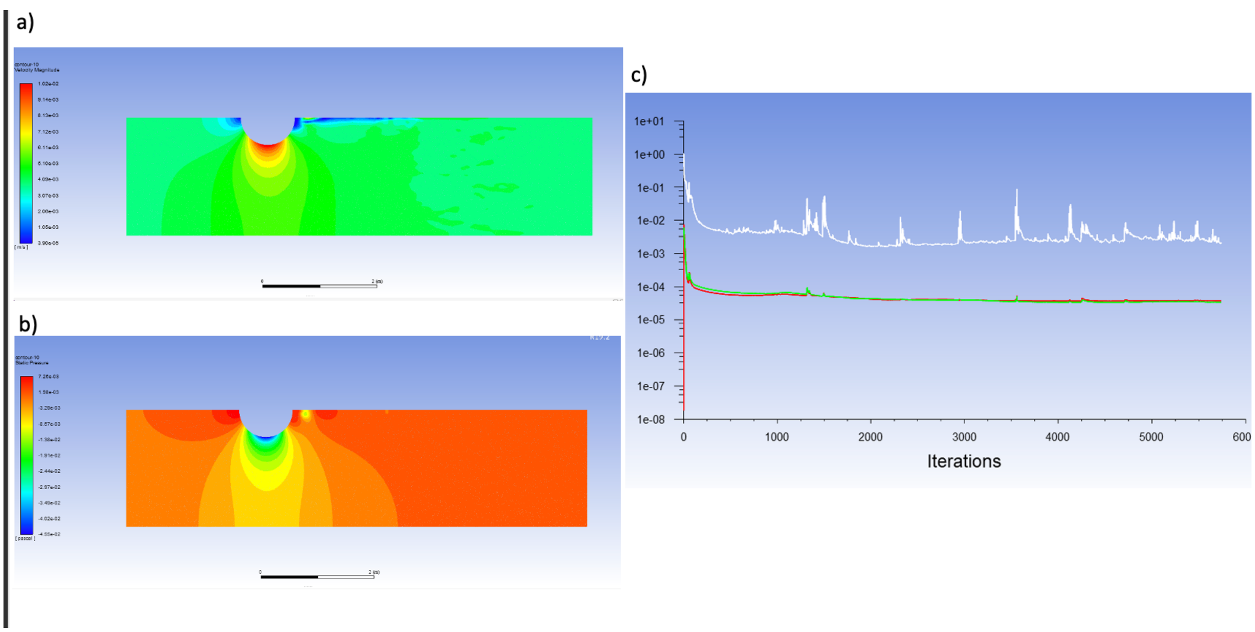


Figure 33. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid water flows through turbulently at 0.004(m/s) at inviscid conditions. (A) Shows the velocity magnitude contour of the flow in m/s. (B)

Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity) lines over an amount of iterations.

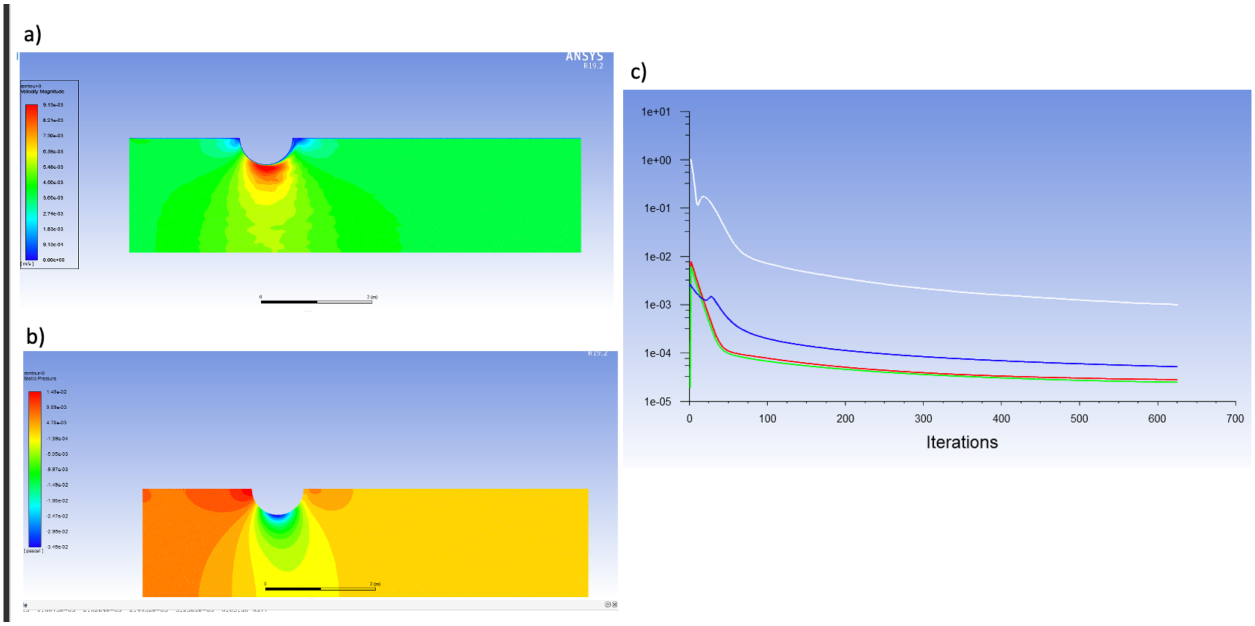


Figure 34. This pipe simulation involves a 2 meter diameter pipe with a circular obstacle. Liquid water flows through turbulently at 0.004(m/s). (A) Shows the velocity magnitude contour of the flow in m/s. (B) Shows the pressure contour of the flow in pascals. (C) Calculations of the residuals shown by the white (continuity), red (x-velocity), green (y-velocity), and blue (nut) lines over an amount of iterations.

In the sixteenth simulation, represented by **Figure 33 and Figure 34**, a 2m diameter circle obstacle was placed in a pipe with liquid water. The water flowed at a turbulently speed of 0.004 m/s. The power required for this flow regime 0.0478763 mW, as seen in **Table 1**. For simplicity, this model was treated as a steady state flow regime, but it should be modelled as transient for more accuracy, since the flow is time dependent. **Figure 34** represents the simulation modelled at inviscid conditions, which does not accurately depict the pressure and velocity of the water after the obstacle compared to the viscid model. The Reynolds number of this flow was found to be 24457 and the Darcy friction factor was found to be 0.026. In this scenario, there is a relatively low friction factor compared to the laminar cases and consequently there is less pressure loss created due to friction. This pipe is less efficient than the laminar scenario due to its low required power.

Conclusion

Analysis of pressure drop has many powerful applications such as aiding in the diagnoses of artery diseases and designing oil pipes. Using the ANSYS fluid software, pressure drop of obstructed flow regimes were analyzed with altering pipe diameter, fluid velocity, type of fluid, and shape of obstruction in both inviscid and viscid conditions. Turbulent trials had required higher power and had lower friction factors due to their high velocity compared to the laminar trials. Since they required more power, the turbulent pipes tended to be less efficient than the laminar pipes as expected. The pipes that had triangular obstructions showed more accurate flow models under inviscid conditions while the circular obstructions tended to show more accurate models under viscid conditions. The most efficient of the 16 trials performed was the octane 2m circular obstruction with a 0.0001 m/s velocity due to its low required power and pressure drop. In an effort to obtain more realistic data, future studies can implicate transient models for turbulent simulations may produce more realistic data.

References

Mirramezani, M., Diamond, S. L., Litt, H. I., and Shadden, S. C. (January 18, 2019). "Reduced Order Models for Transstenotic Pressure Drop in the Coronary Arteries." ASME. J Biomech Eng. March 2019; 141(3): 031005. https://doi.org/10.1115/1.4042184

Yue, P., Yang, H., He, C., Yu, G. M., Sheng, J. J., Guo, Z. L., Guo, C. Q., & Chen, X. F. (2020). Theoretical Approach for the Calculation of the Pressure Drop in a Multibranch

Anthony Gisolfi
EMCH 360
12/7/2022
Horizontal Well with Variable Mass Transfer. *ACS omega*, *5*(45), 29209–29221.

https://doi.org/10.1021/acsomega.0c03971