Simulation and Analysis of Flow within a Two Dimensional Channel

Group Name: Phoenix Dynamics.

Group Members:
Inzamamdeen Kassim
Josue Criollo
Basel Abdelhalim
Christopher Triana

Date: 04/15/2019

The City College of New York
138th Street & Convent Avenue
New York, New York 10031

Acknowledgement

This project is sponsored by the City College of New York-Grove School of Engineering. The success and final outcome of this assignment required a lot of guidance and assistance from many people including classmates. We are extremely fortunate to have achieved this all along the competition of our assignment work. We thank Professors Balazs Fekete and Hansong Tang along with his teaching assistant W. Dong(Victor) from our Fluid Mechanics course for being extremely helpful.

Table of Contents

List of Figures

List of Tables

Abstract

In fluid mechanics, we learned how a fluid behaves when they are acting on a particular system. For this project, we will be working with a system which contains water-liquid, a channel whose dimensions are 15m in total length, 1m in width for the small pipe and 2m in width for largest portion of the channel. We set the center of the pipe to have xy coordinates of (0,0) between the small and large sections of the channel. For our project, we were told to use two different velocities. The first velocity that we used was 0.2 m/s (turbulent) and the second velocity is 0.002 m/s (laminar). The goal of this report was to analyze these two flows and compare a variety of their fluid properties such as the streamlines of the channel from start to exit, comparing the different pressures occurring in the flow through contour lines, vorticity, and shear stress. Included in this report are the XY plot of the velocity vs the positions for each respective flow. All of this was possible through a program known as FLUENT. With this program, the group was able to provide excellent visualization of the different properties that are occurring within each flow. We also found that the maximum pressure at the bottom of the channel using these two velocities. In addition, with the use of fluent we were able to calculate the shear stress on the cavity, which matched values in the figures of velocity and pathlines of the flow. Last, we compared between the relationship of velocity and position of the flow before and after the expansion of channel.

The ability of calculating this complicated behavior in the flow field with relative ease and a large degree of accuracy shows how important and utilization computer software, namely FLUENT, is in any field of engineering.

I. Introduction

The main idea of this report is to investigate the behavior of a liquid(water) acting on a channel that our lab instructor provided us. We want to increase our ability to apply fundamental principles that our instructor has been teaching us in the classroom to analyze all the data required for the flow of the liquid effectively. In order to study the system accurately, we applied the program known as FLUENT. This program in particular is a very powerful and flexible general-purpose computational fluid dynamics software used to model and manage fluid flow, heat transfer, or turbulence in a system. Based on the results from FLUENT, we will plot velocity and pressure along the given channel length and we will also calculate the flow rate at different cross sections.

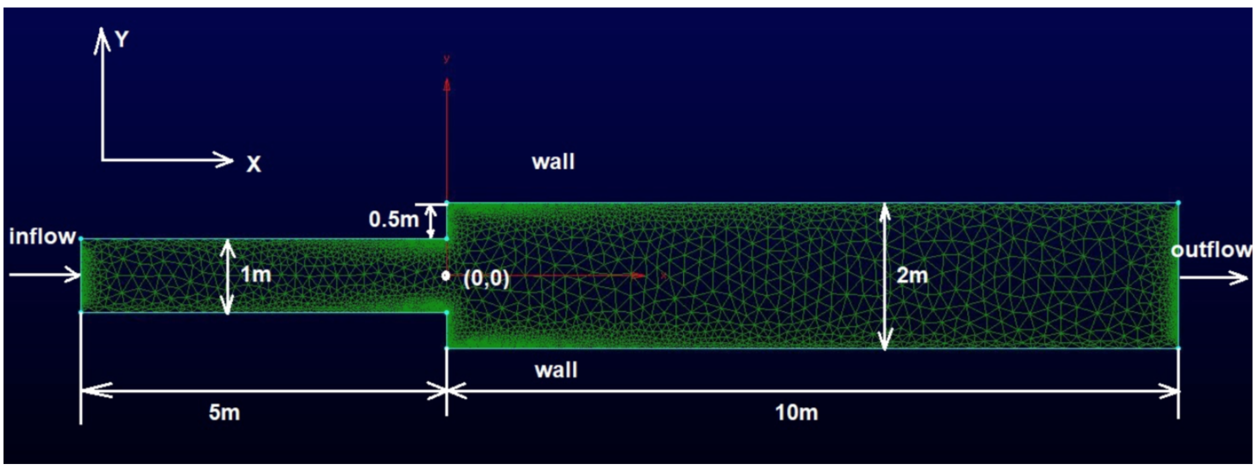

***Figure 1**:Given channel to analyze Flow rate*

Figure 1 above represents a simple two dimensional channel with its own dimensions. We can describe this channel by mentioning that the inflow of the liquid is from the left-hand side of the channel when x=-5m being a positive inflow in the x-axis, and the outflow which is located at the right-hand side of the channel at x=10m.

II. Flow Simulation Using FLUENT

Today, the use of simulation technologies is dominant across the sciences. From the interactions of subatomic particles to orbital mechanics, computer recreation spreads the reach of clean methodical representations, decreases computation time, and provides verification to experimental results. With the help of ANSYS FLUENT software, the verification of these results can be done even faster, more reliable and more available to compute resources from different fluid mechanics methods.

II-1 Initializing FLUENT

A flow field with possessions already outlined incorporating the geometry of the structure, values for the water and other relevant factors were mentioned during the lecture. The requirements for the fluid is that it must be defined as pure water in the fluid region and the only variable that is different during the program is the magnitude of the velocity normal to the inlet. We can see more details in Table 1 below.

Table 1:Values and System Properties used in FLUENT.

Water Properties	Values	Used System	
Density(kg/m3)	998.2	**Operating Pressure(pascal)**	101325
Viscosity(kg/m-s)	0.001003	**Viscous Method**	Spalart Allmaras
Temperature(k)	293.15	**Spatial Dimension**	2D

The flow will be analyzed by initializing velocities 0.2 m/s and 0.002 m/s normal to the inlet of the control volume as shown in Figure 2 below. In order to start, we first need to turn on the Energy equation because simulation will not start temperature meaning that we will not be able to choose thermal in the boundary condition. We then need to get rid of all the air in the channel and only have water inside. Under monitors and residuals, check that we have values of 0.0001 for “monitor check convergence Absolute Criteria”.

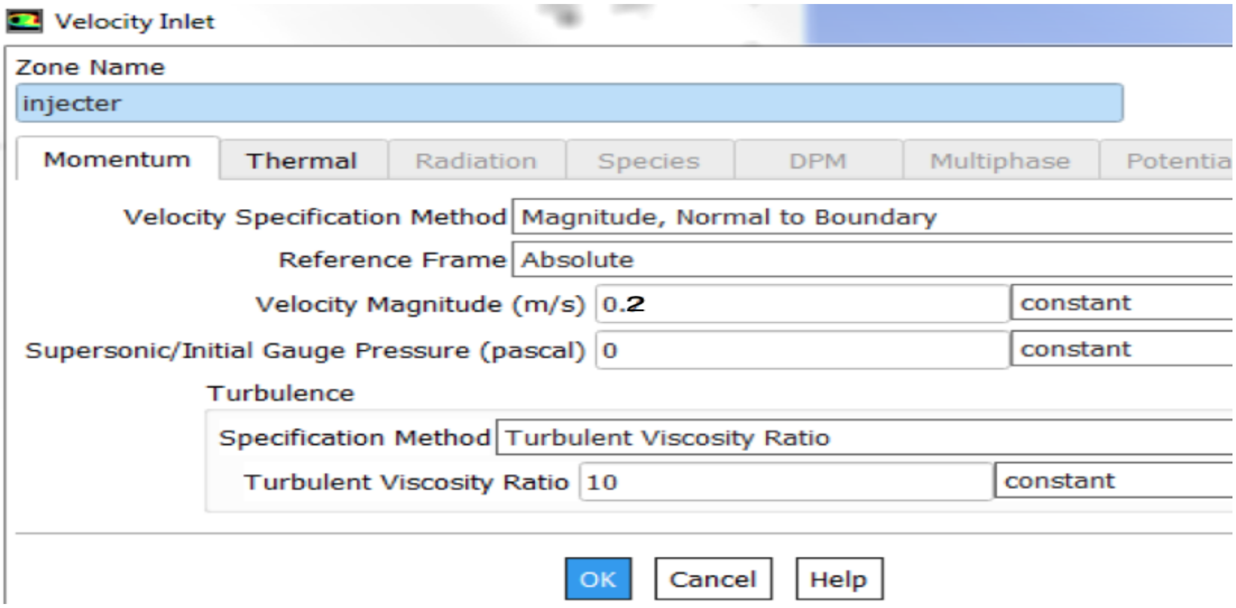

Figure 2:*changing velocities Magnitudes as 0.2 and 0.002*

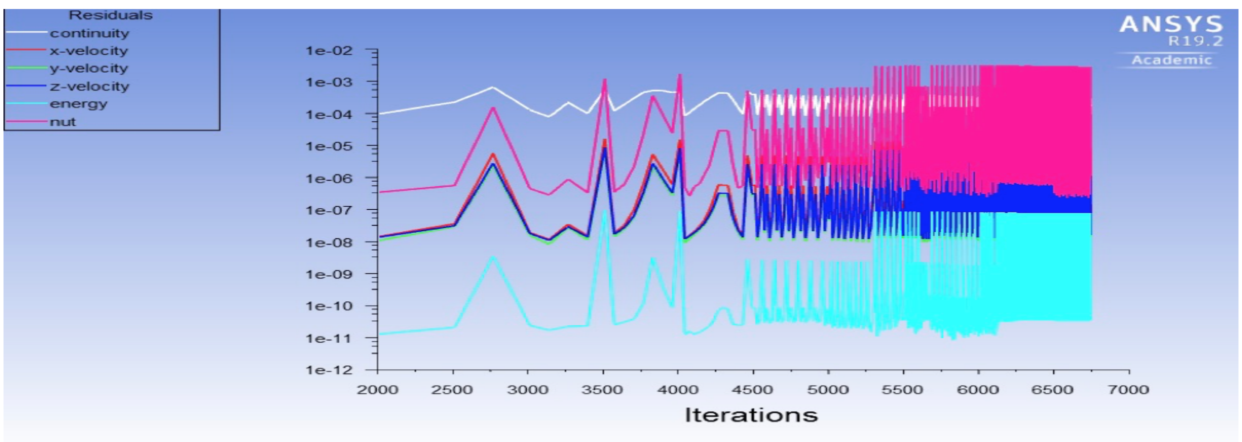

Figure 3*: Scale Residuals.*

Fluent iterates functions over this mesh until they join to constraints recognizing that we they must be held due to the property of continuity. The closer the residuals approach to the continuous nature of the field as shown in Figure 3, the more accurate the model will get; although it might take even longer to get better and more precise results.

II-2 Analyzing Fluid Mechanics Equations

Vorticity Equation

To understand the processes that produce changes in vorticity, we would like to derive the expression that includes the time derivative of vorticity as ω=2Ω which is also equivalent to: $\frac{d}{dt}(\frac{v}{\partial \mathrm{x}} - \frac{\partial u}{\partial y})\ldots = (\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z})i + (\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x})j + (\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x})k$ for more detailed derivation, follow Appendix A.

Shearing Stress Equation

Like in our Mechanics of materials course, he have learned a lot about stress; particularly shearing stress. Both, materials and fluid mechanics share the same concepts in terms of shearing stress. The shear stress on a smooth plane surface is variable over the surface. Hence the total shear force in a given direction is obtained by integrating the component of shear stress in that direction over the total area of the surface. The shear stress on a smooth plane is a direct function of the velocity gradient next to the plane. $\tau = \mu \frac{dV}{dy}$ where $\tau$is the shear stress, μ is the dynamic viscosity, and $\frac{dV}{dy}$is the time rate of strain, which is also the velocity gradient normal to the wall.

Any problem involving shear stress also involves the flow pattern in the vicinity of the surface. The layer of the fluid near the surface that has undergone a change in velocity because of the shear stress at the surface called the boundary layer, and the general area of study of the flow pattern in the boundary layer, as well as of the associated shear stress at the boundary is called boundary layer theory[1]. A better derivation for the shearing stress equation can be found in Appendix B.

Using the equations that are more in detail from Appendix A and B, we were able to determine our shear stress equation and Vorticity equation. Since sometimes our manual calculations can be difficult to get, FLUENT is able to separate analysis over a system to approach behavior. This advantage is particularly significant in many engineering purposes and the ability to model such complicated problems is relevant in any field of engineering. The principal purpose of this analysis will test the contiguity of solutions provided by FLUENT to the fundamental theories of fluid mechanics that have been introduced through the beginning of the semester.

III. Analysis of the Flow

With constraints established and a moderately small error in the values across the channel, properties of the flow can be observed. As mentioned before, FLUENT has many advantages such as supplying us enough tools and functions that allow us as the user to study features of the flow. These characteristics at a specific region, such as velocity and pressure, can be exhibited as vectors or contours. All these utilities are available under the menu entitled "Post-processing" and then in graphics. Once we get the adjustments correctly, we should be able to get our information we desire well-defined and in a brief manner.

III-A Streamlines of the Channel from Entrance to Exit.

Laminar Flow (Velocity = 0.002 m/s)

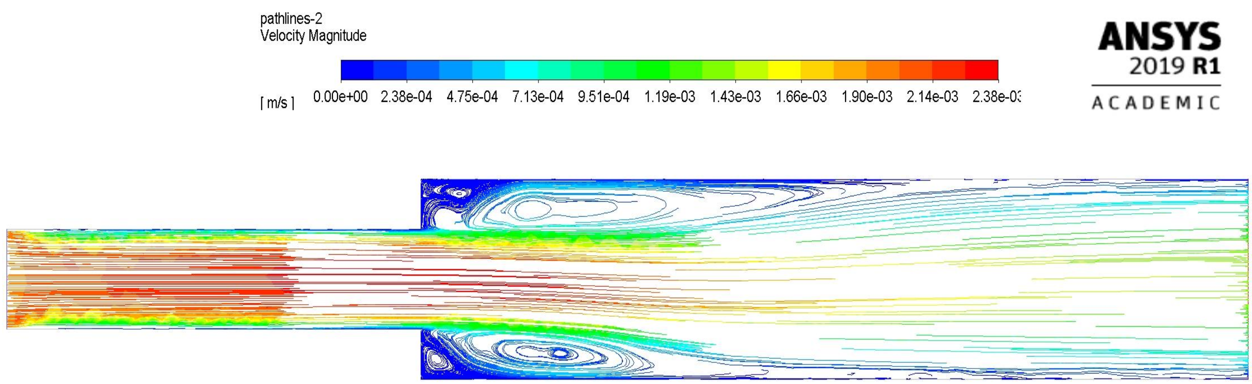

Figure 4:*This image shows the streamlines as the fluid flows in parallel layers, with no disruption between the layers with a velocity of 0.002 m/s.*

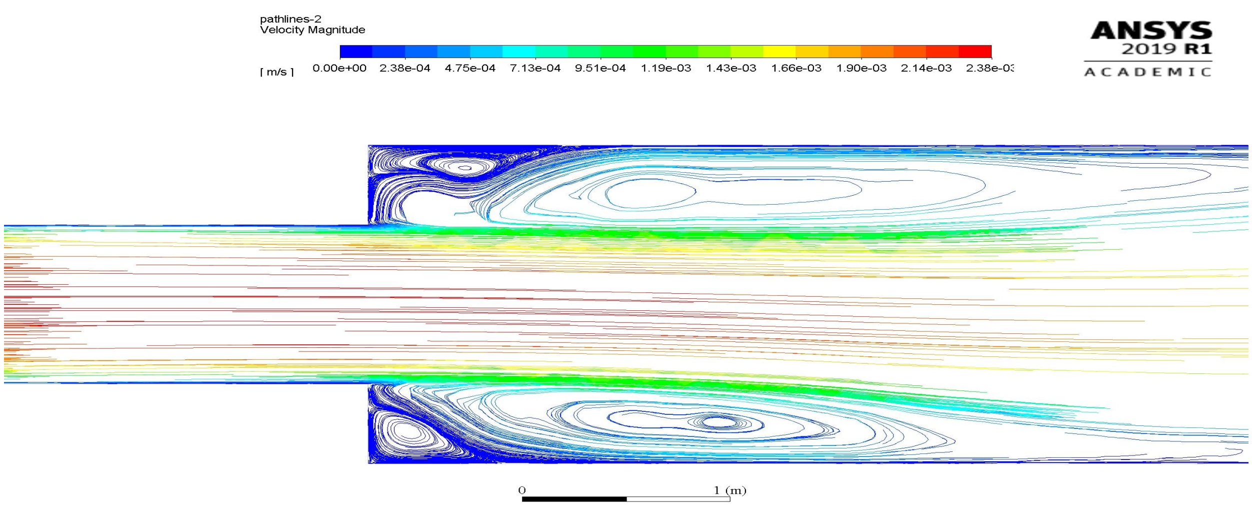

Figure 5:*This image shows the vortices in greater detail while the flow with 0.002 m/s velocity enters the channel.*

Based on the details in figure 4, the image varies in color symbolizing the different velocities that occur in various areas. The red area in the image indicates the maximum velocity, the green area in the channel represents medium velocity and the blue area has almost zero velocity. At such low velocity, the fluid seems to flow without lateral mixing, and adjacent layers slide past one another. Due to the changes of the cross-sectional area and low velocity, the fluid flow creates circular motions "vortex rings" at the corners of the channel as indicated in the blue area. Thus, the top blue area has a counterclockwise vortex rings, while the bottom area has vortex rings flowing clockwise. Figure 5 shows a zoomed in image of figure 4 for better visualization of what is occurring in the channel where it expands.

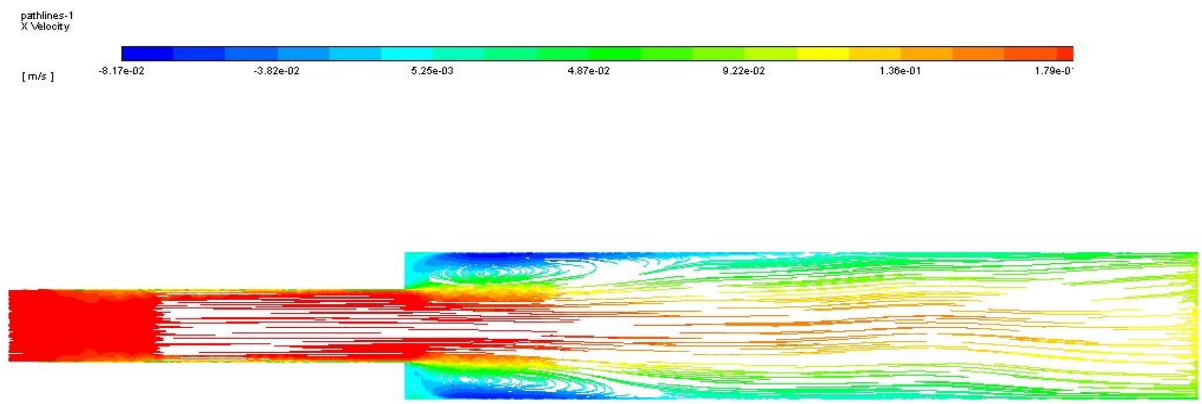

Figure 6*:This figure shows the streamlines as the flow with a velocity of 0.2 m/s enters from the left side of the channel.*

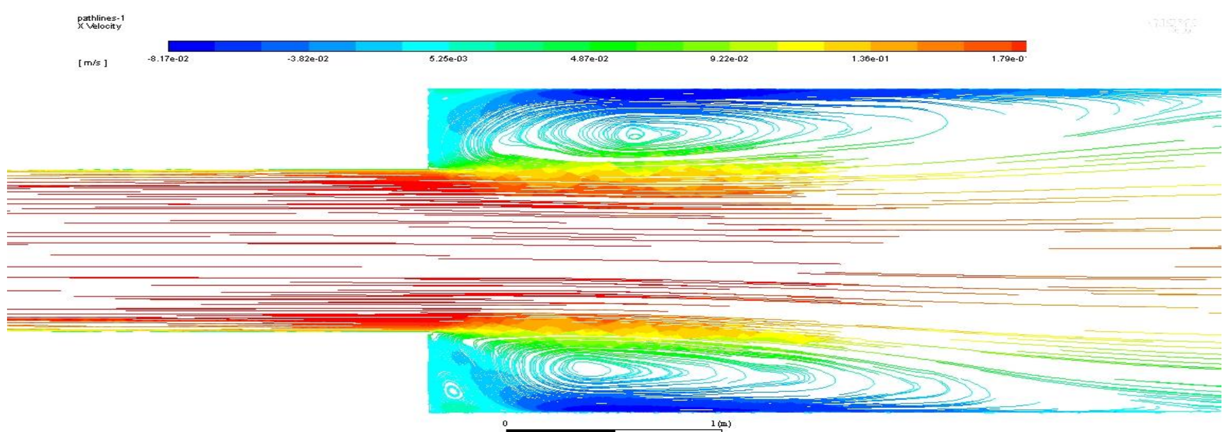

Figure 7*:This image shows the vortices in greater detail as the flow with 0.2 m/s velocity enters the channel.*

Turbulent flow is any pattern of fluid motion characterized by chaotic changes in pressure and flow velocity. Turbulence is caused by excessive kinetic energy in parts of a fluid flow, which overcomes the damping effect of the fluid's viscosity. As shown in figure 6, the image varies in color symbolizing the different velocities that occur in various areas. The red area in the image indicates the maximum velocity, the green area in the channel represent medium velocity and the blue area has almost zero velocity. At such high velocity, the fluid does not flow in parallel layers, the lateral mixing is very high, and there is a disruption between the layers. Due to the changes of the cross-sectional area and high velocity, the fluid flow creates unsteady circular motions “vortex rings” of many sizes which interacts with each other at the corners of the channel as indicated in the blue area. Thus, the top blue area has a counterclockwise vortex rings, while the bottom area has vortex rings flowing clockwise. Figure 7 shows a zoomed in image of figure 6 for better visualization of what is occurring in the channel where it expands.

III-B Contour Lines of Pressure for the Flow.

Laminar Flow (Velocity = 0.002 m/s)

Based on the figure below, it can be seen that the total pressure within the channel varies as indicated by the color spectrum, where red is the largest total pressure while the dark blue areas represent the lowest total pressure. From the filled contour line image, the group noticed that the total pressure is greatest during the 1-inch entrance, however, once it passes this point and there is a change in cross-sectional area, the pressure gradually decreases from left to right until eventually, there is little to no red area. On the top and bottom left portion of the larger area, the pressure is at its lowest, as seen with a very dark blue color. This makes sense since the pressure is coming in from the center and pressure that is diffused into this area is lower as the distance from the center increases. On the right of the channel, it can be seen that there is moderate pressure as indicated by the yellow-green color. This gradual decrease in total pressure can also be seen in the unfilled contour image.

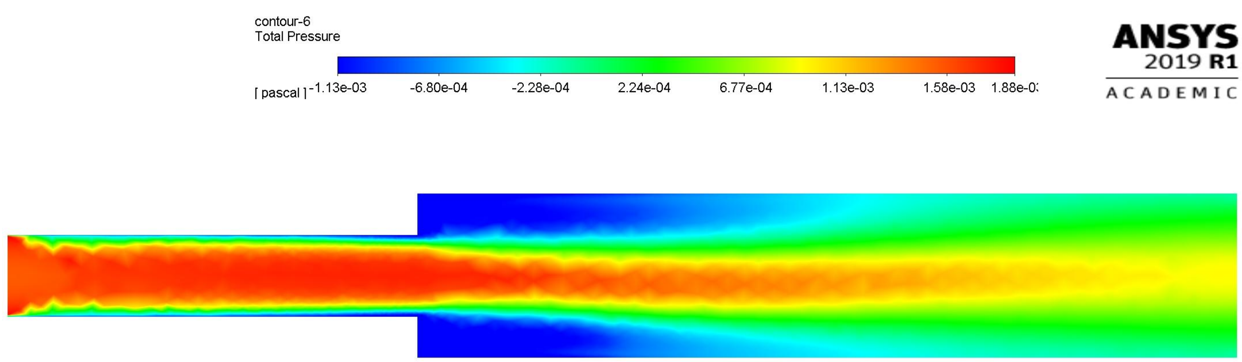

Figure 8*:The total pressure for the entire channel can be interpreted from the image shown.*

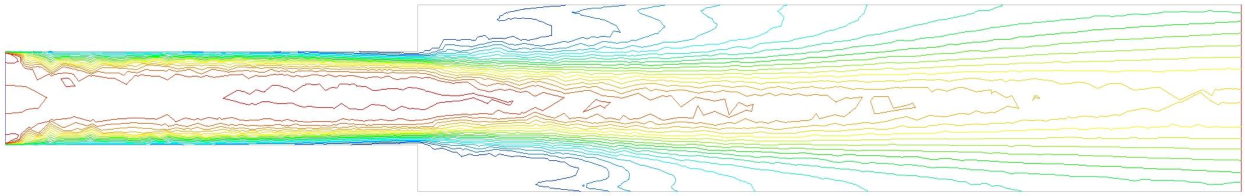

Figure 9*:This image demonstrates the total pressure contour lines filled and unfilled.*

Turbulence Flow (Velocity=0.2m/s).

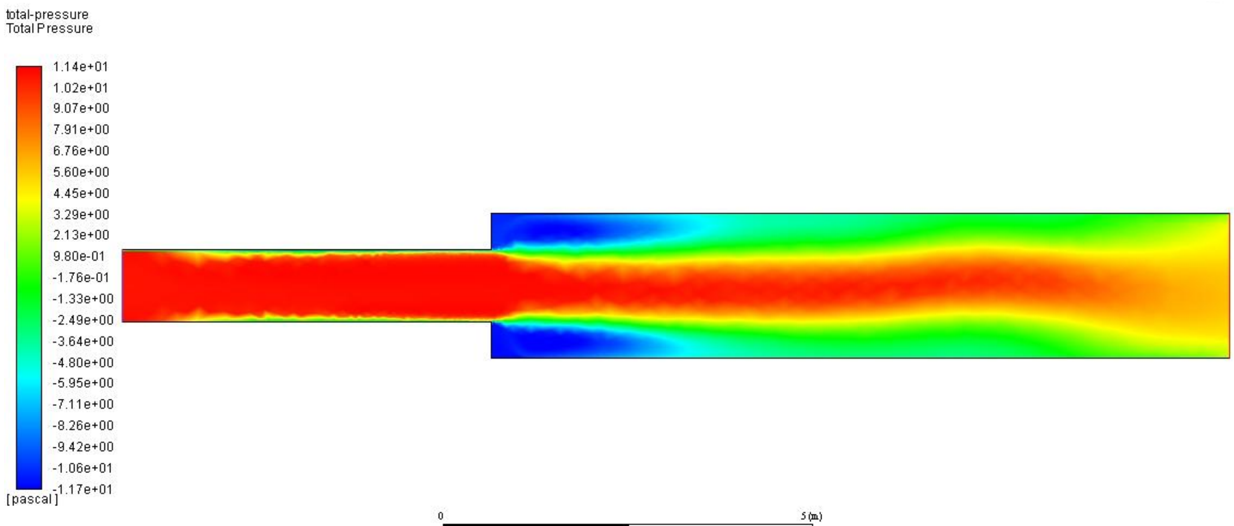

Figure 10*:The total pressure for the entire channel can be interpreted from the image shown.*

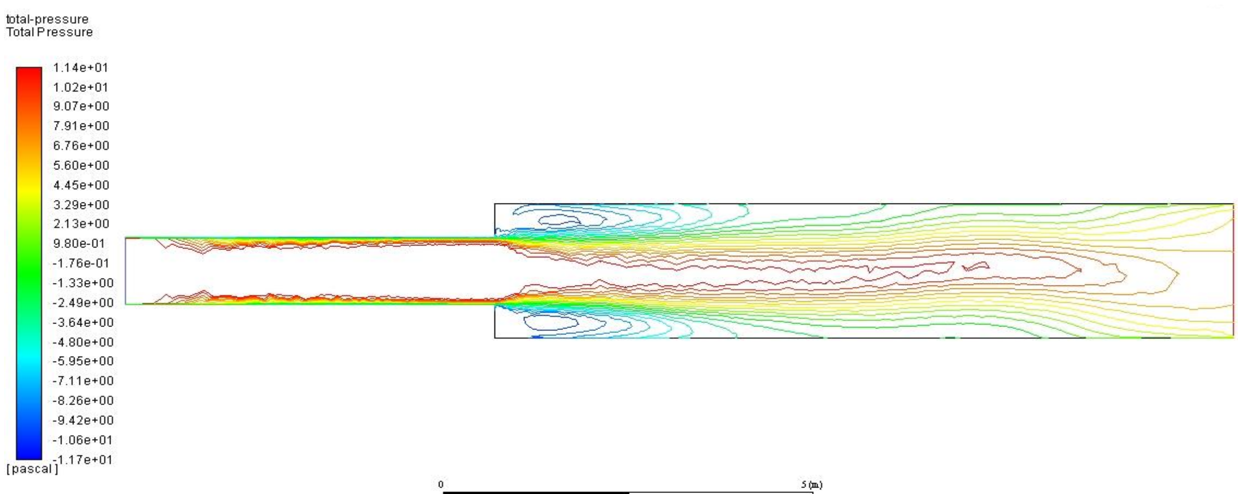

Figure 11*:This image demonstrates the total pressure contour lines unfilled with a better visualization of where the vortices are located.*

Based on the above figures, it can be seen that the total pressure within the channel varies as indicated by the color spectrum, where red is the largest total pressure, while the dark blue areas represent the lowest total pressure. From the filled contour line image, we noticed that the total pressure is greatest during the 1 inch entrance, however unlike the contour image for the laminar flow (Figure 8), the red area of the turbulent flow is smoother and does not gradually decrease in pressure until the very end of the larger cross sectional area. There is also less blue area which have the lowest total pressure. As we start from the left of the channel at position x=-5 m, the pressure is 11.4 Pa. The blue areas in the channel have a very low pressure with -11.7 Pa since the velocity is also low in that region.

III-C Vector Fields of the Flow.

Laminar Flow (Velocity = 0.002 m/s)

As we can see in figure 12 below, the length of the arrows represents the magnitude or size of the vector and the arrows point in their appropriate direction. The arrows also vary by color as shown in the color spectrum with red indicating greatest velocity magnitude, and blue indicating the lowest velocity magnitude. At the entrance of the channel, the arrows have the highest velocity magnitude at the center and decrease gradually as we got closer to the top and bottom wall due to the shear stresses created by the contact between the fluid and the walls. To further analyze these velocity vectors, a closer look was taken, as shown in figure 13, which illustrates a more zoomed in version of the image shown in the previous figure. After passing the 1-inch entrance, the velocity vectors at the bottom left corner of the channel show that the vortices have low magnitudes with an average of 0.002 m/s, but each magnitude varies in terms of directions. It can also be seen that with this change in cross-sectional area, some of the vectors start to diverge away from the center red velocity magnitude vectors and also decrease in speed.

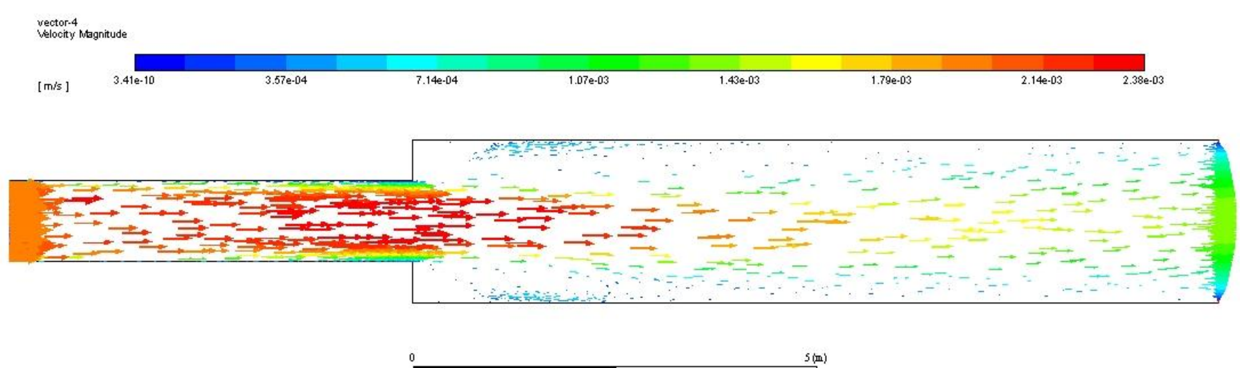

Figure 12:*This image shows the velocity magnitude vectors in the channel.*

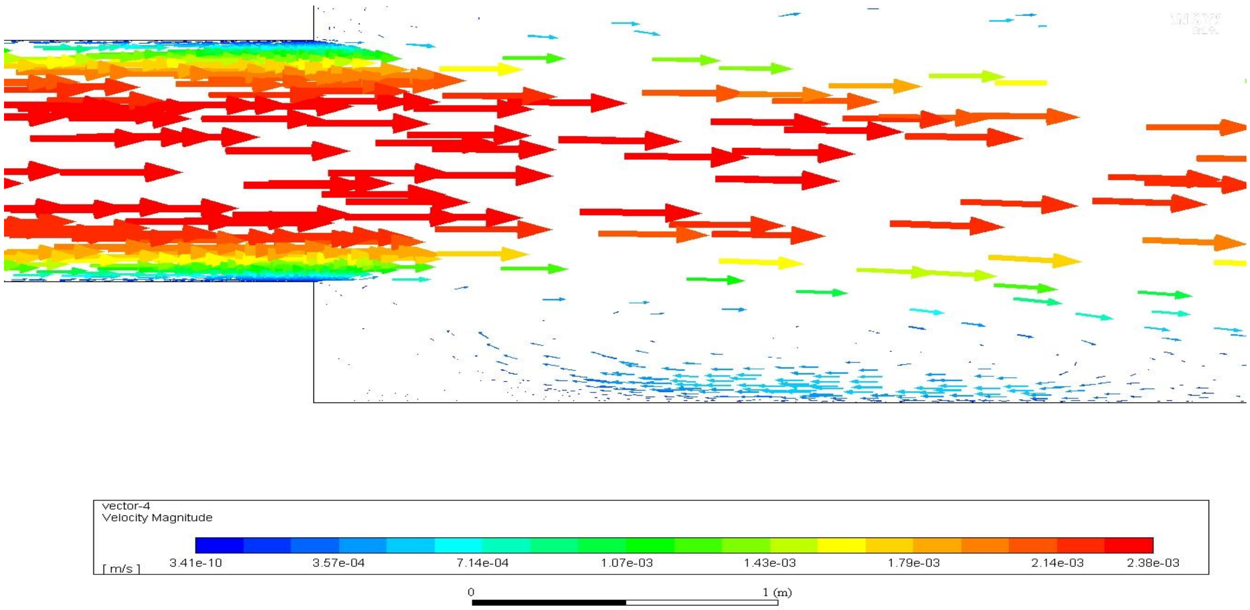

Figure 13*:A closer image of figure 11 in greater detail for further analysis of the velocity magnitude vectors.*

Turbulence Flow (Velocity = 0.2 m/s)

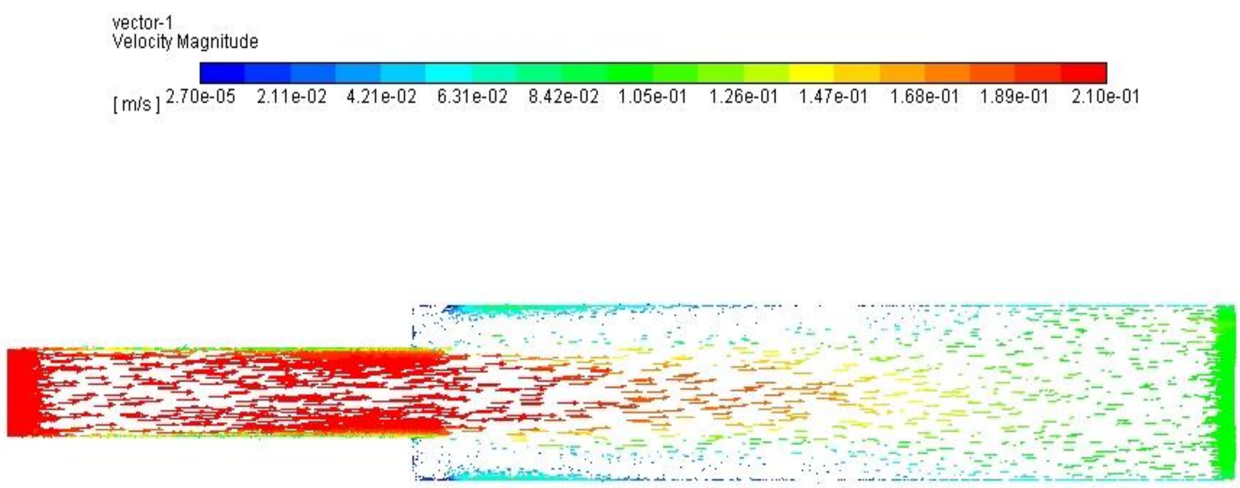

Figure 14*:This image shows the velocity magnitude vectors in the channel.*

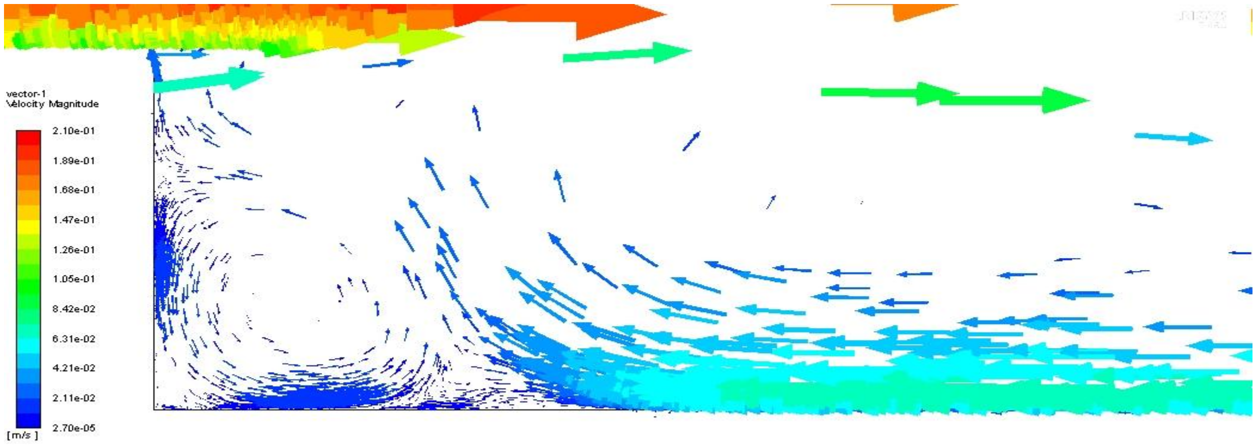

Figure 15*:A closer image of figure 13 for further analysis of the velocity magnitude vectors.*

As seen in figure 14, the length of the arrows represent the magnitude or size of the vector and the arrows point in their appropriate direction. The arrows also vary by color as shown in the color spectrum at the top of figure 14 with red indicating the greatest velocity magnitude, while blue indicates the lowest velocity magnitude. Just as the laminar flow image (figure 12 and 13) The arrows have the highest velocity magnitude at the entrance of the channel since all the flow force is being spread from the entrance, however, the amount of arrows is greater in turbulent flow not just at the entrance but the whole channel. In figure 15, the vectors in the bottom left corner after passing the 1-inch main entrance, show that the vortices are unsteady with many different sizes and have low magnitudes with an average of 0.02 m/s but each magnitude varies in terms of directions. It can also be seen that with this change in cross sectional area, some of the vectors start to diverge away from the center red velocity vectors and also decrease in speed.

III-D Vorticity

Laminar Flow (Velocity = 0.002 m/s)

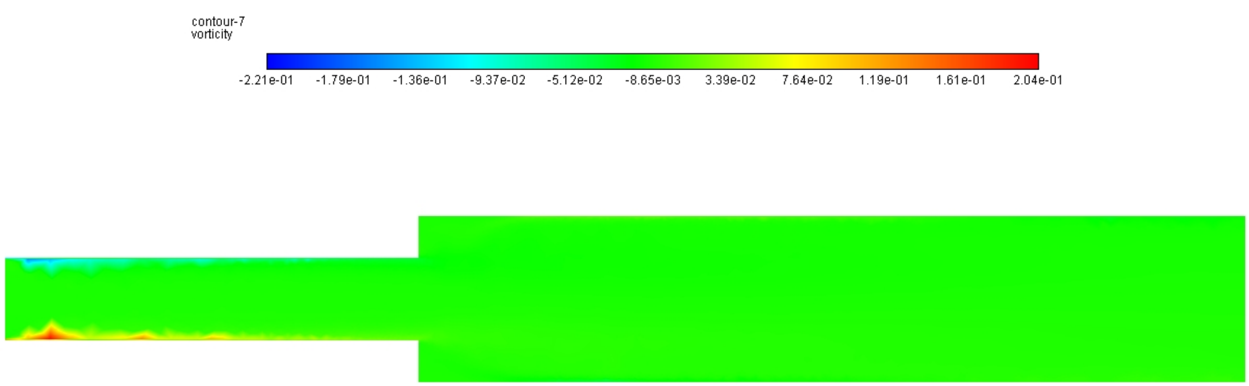

Figure 16*:This image shows the vorticity fields of the flow in the channel before removing extreme values.*

Figure 16 shows the vorticity of the Laminar flow as is can be seen from this contour that the extreme values on the entrance channel is so high that activity in the region of interest cannot be seen. In order to see activity at the expansion we need to remove extreme values by reducing the range smaller. Doing this produced can be demonstrated in figure 17 below.

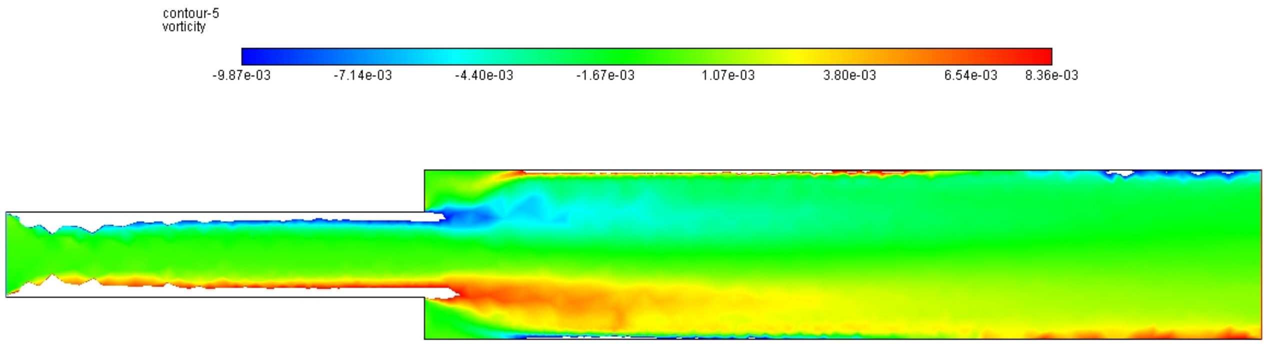

Figure 17*: This image shows the vorticity fields of the flow in the channel after removing extreme values.*

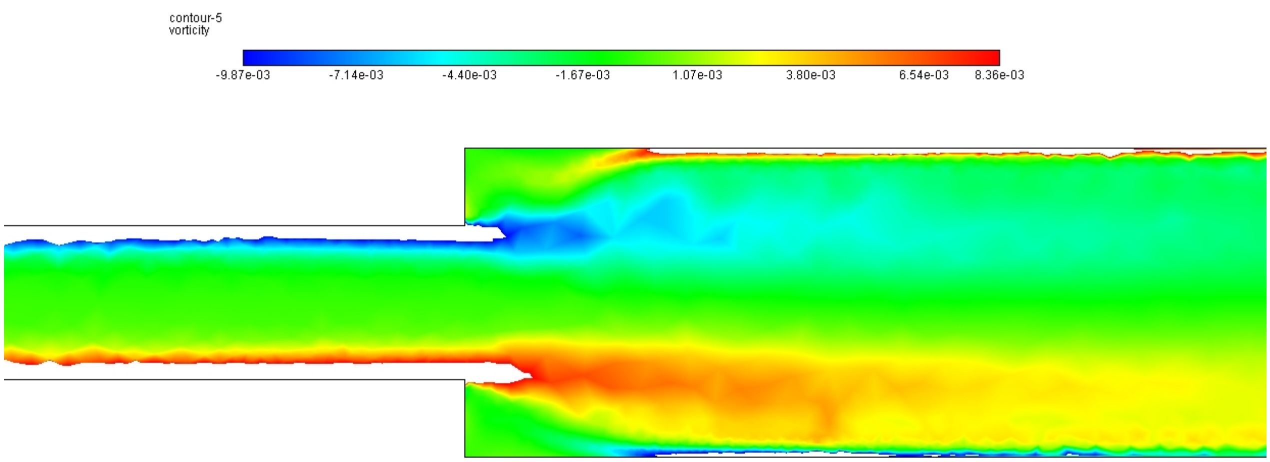

Figure 18*:A zoomed in image of figure 17 used for further analysis of the flow in the channel.*

The vorticity is related to the flow's circulation along a closed path. In this figure, the extreme values were eliminated to clearly show the vorticities at the bottom left and right sides of the channel. As shown in figure 17, the various ranges in color represent the orientation of how the fluid rotates with respect to each local point. The red areas indicate the fluid is rotating positively meaning it is rotating in a clockwise manner, while the blue areas indicate the fluid rotating negatively, or counterclockwise. The green area indicates little to no rotation occurring. Based on figure 18, a zoomed in image of figure 17, it can be seen that little to no rotation is occurring in the center of the channel, which makes sense since the fluid is flowing from left to right in the same direction as it was when it left the 1-inch entrance. As the fluid leaves the entrance, however, it appears that the some of the fluid, in an attempt to diffuse and cover up the now larger area, begins to rotate in the appropriate direction the fluid needs to fill up the whole area, thus we have a counterclockwise rotation at the top of the channel and we have a clockwise rotation at the bottom of the channel.

Turbulence Flow (Velocity = 0.2 m/s)

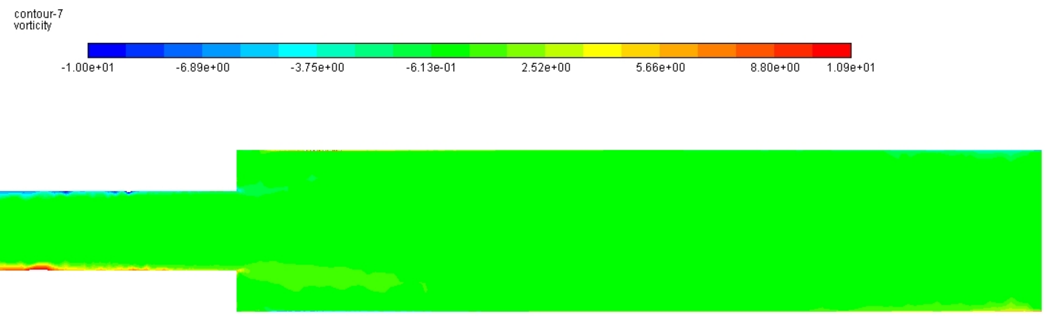

Figure 19*: This image shows the vorticity fields of the flow in the channel before removing extreme values.*

Figure 19 shows the vorticity of the Turbulent flow as is can be seen from this contour that the extreme values on the entrance channel is so high that activity in the region of interest cannot be seen. In order to see activity at the expansion we need to remove extreme values by reducing the range smaller. Once again, doing this produced can be shown in figure 20 below.

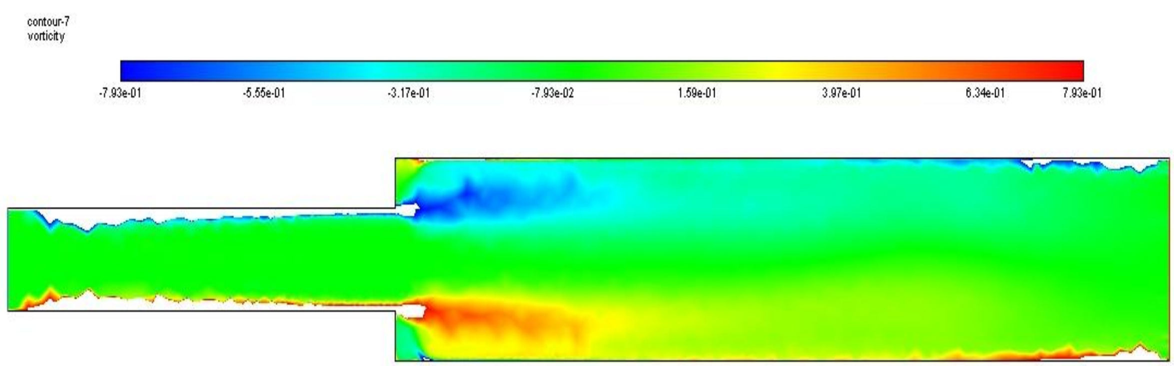

Figure 20: *This image shows the vorticity fields of the flow in the channel after removing extreme values.*

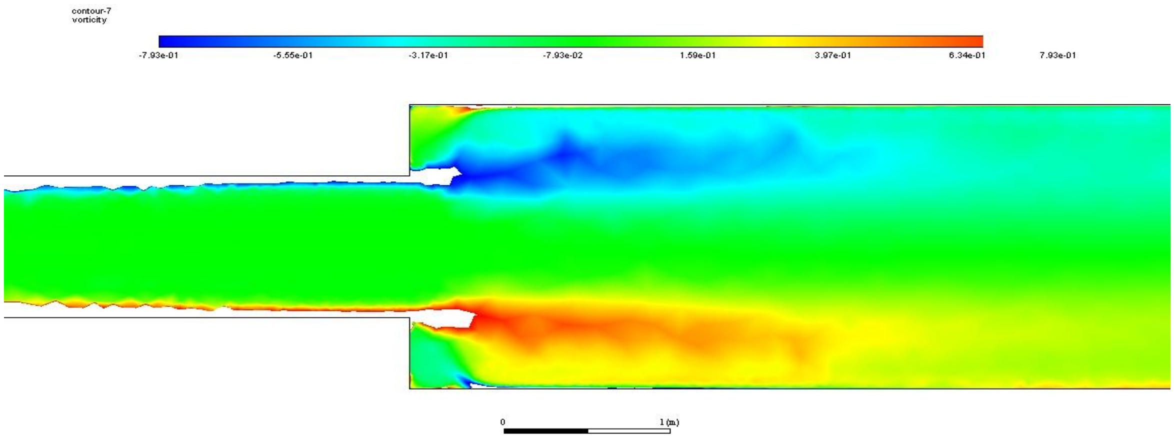

Figure 21: *A zoomed in image of figure 20 for further analysis of the flow in the channel.*

As mentioned earlier, vorticity is related to the flow's circulation along a closed path. In figures 20 and 21, the extreme values were eliminated to clearly show the vorticities at the left side of the channel. As shown in the above images, the various ranges in color represent the orientation of how the fluid rotates with respect to each local point. The red areas indicate the fluid rotating positively meaning it is rotating in a clockwise manner, while the blue areas indicate the fluid rotating negatively, or counterclockwise. The green areas indicate little to no rotation occurring. Based on the images, it can be seen that little to no rotation is occurring in the center of the channel, which makes sense since the fluid is flowing from left to right in the same direction as it was when it left the 1 inch entrance. Similar to the laminar flow, it appears that the fluid, in an attempt to diffuse and cover up the now larger area, begins to rotate in the appropriate direction the fluid needs to fill up the whole area, thus we have counterclockwise rotation at the top of the channel and we have a clockwise rotation at the bottom of the channel.

III-E XY Plot of the X velocity versus x positions.

Laminar Flow (Velocity = 0.002 m/s)

The XY plot for laminar flow of the velocity versus the position shows the distribution of the velocity along a cross-sectional area can be demonstrated in figure 22. It can be seen that this distribution is different for the different positions relative to the entrance and exit. The entrance boundary shows a uniform distribution of the entrance velocity, 0.002 m/s, which starts to follow a parabolic distribution as we move away from the entrance to the exit.

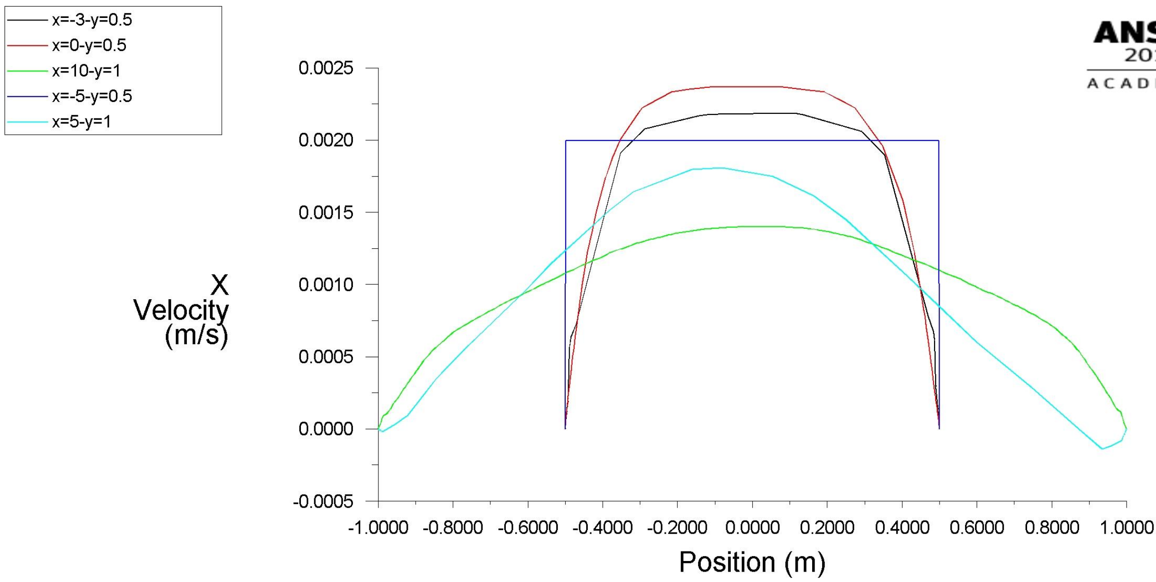

Figure 22*:This graph illustrates the relationship between the velocity (in the x-direction) of the channel and its relative position with an entrance velocity of 0.002 m/s.*

Turbulence Flow (Velocity = 0.2 m/s)

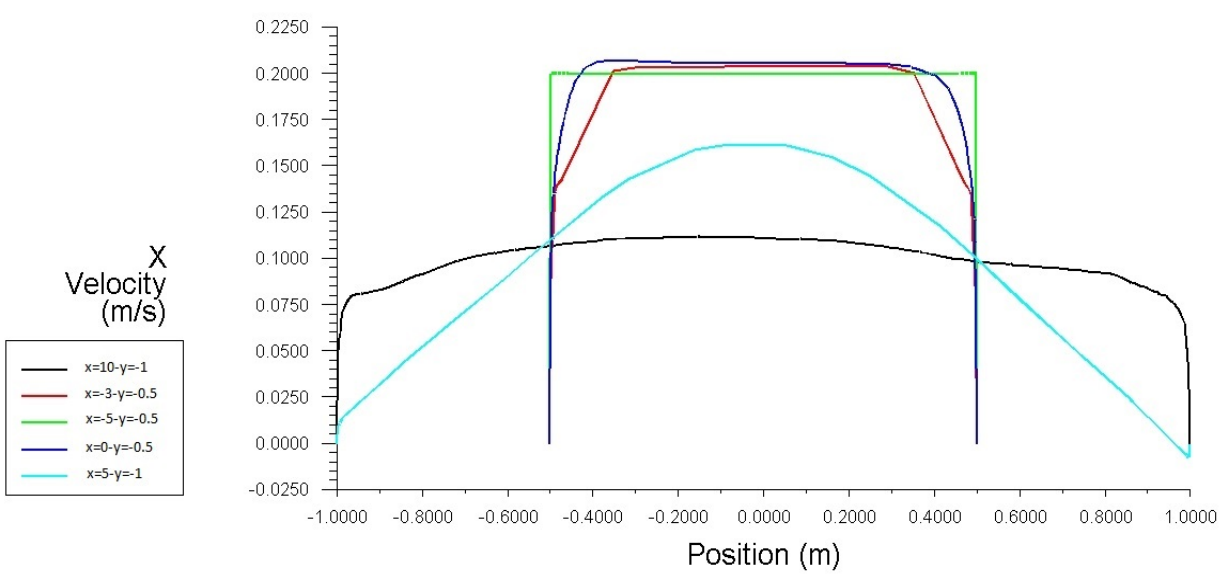

Figure 23: *This graph illustrates the relationship between the velocity (in the x-direction) of the channel and its relative position with an entrance velocity of 0.2 m/s.*

The XY plot for the turbulence flow is quite different from that of the laminar flow. It is clear that the distribution for the velocity along the cross-sectional area followed a more uniform distribution from the entrance to position X=5, after which we started to see the parabolic distribution.

III-F Extracted Pressure values from given boundaries.

Laminar Flow (Velocity = 0.002 m/s)

For the following tables, we were able to extract all the data for dynamic pressure for each boundary. One way to check our data, we analyzed the image from fluent using dynamic pressure contours.

Table 2: *Boundary i: x= -5m and -0.5<y<0.5 using a velocity of 0.002 m/s.*

y(m)	-0.45	-0.2	0	0.2	0.45
p(Pa)	0.00199646	0.00199637	0.00199642	0.00199644	0.00199648

Table 3: Boundary ii-a): -5<x<0 *and y=-0.5 using a velocity of 0.002m/s*

x(m)	-4.5	-4	-3	-2	-1
p(Pa) [$x10^{-7}$]	9.82811	2.26646	1.9892	2.49094	1.74762

Table 4: Boundary ii-b): *-5<x<0 and y=0.5 using a velocity of 0.002m/s*

x(m)	-4.5	-4	-3	-2	-1
p(Pa) [$x10^{-7}$]	0.157832	2.08779	2.7253	2.75317	2.32717

Table 5: Boundary iii): x=0 and -0.5>y>-1 using a velocity of 0.002m/s

y(m)	-0.6	-0.7	-0.8	-0.85	-0.9
p(Pa)[$x10^{-10}$]	4.65484	0.837095	8.37095	0.478813	0.127353

Table 6: Boundary iv-a): 0<x<10 and y=-1 using a velocity of 0.002m/s

x(m)	1	2	3	4	5
p(Pa)($x10^{-9}$)	3.50242	106.885	23.8276	3.94883	1.63014

Table 7: Boundary iv-b): 0<x<10 and y=1 using a velocity of 0.002m/s

x(m)	1	2	3	4	5
p(Pa)(x10^{-8})	0.013642	5.1137	6.16016	2.64188	1.11889

Table 8: Boundary v): x=10 and -1<y<1 using a velocity of 0.002 m/s

y(m)	-0.9	-0.8	-0.7	-0.6	-0.5
p(Pa)	0.0000758945	0.00022157	0.000333842	0.000450478	0.000581216

Turbulence Flow (Velocity = 0.2 m/s)

For turbulence flow, we used the same procedure as for laminar flow. We extracted all the data for dynamic pressure for each boundary and also we made sure that these values made sense by using the dynamic pressure contours.

Table 9: Boundary i: x= -5m and -0.5<y<0.5 using 0.2 m/s

y(m)	-0.45	-0.2	0	0.2	0.45
p(Pa)	19.9655	19.964	19.964	19.9629	19.9625

Table 10: Boundary ii-a: -5<x<0 and y=-0.5 using 0.2 m/s

x(m)	-4.5	-4	-3	-2	-1
p(Pa)	1.89832	1.25267	0.558785	0.866172	0.863044

Table 11: Boundary ii-b: -5<x<0 and y=0.5 using 0.2 m/s

x(m)	-4.5	-4	-3	-2	-1
p(Pa)	1.98909	1.46397	0.929999	0.557427	0.632024

Table 12: Boundary iii) When x=0 and -0.5>y>-1 using 0.2 m/s

y(m)	-0.6	-0.7	-0.8	-0.85	-0.9
p(Pa)	0.476677	0.00146302	0.00276381	0.00128224	0.000411342

Table 13: Boundary iv-a: 0<x<10 and y=-1 using 0.2 m/s

x(m)	1	2	3	4	5
p(Pa)	0.221988	0.178607	0.059831	0.0257568	0.0095564

Table 14: Boundary iv-b) When 0<x<10 and y=1 using 0.2 m/s

x(m)	1	2	3	4	5
p(Pa)	0.271125	0.201125	0.0883228	0.00805397	0.0496779

Table 15: Boundary v) When x= 10 and -1<y<1 using 0.2 m/s

y(m)	-0.9	-0.8	-0.7	-0.6	-0.5
p(Pa)	3.46543	4.20675	4.9883	5.32503	5.60469

III-G Calculating Shear Stress.

Laminar Flow (Velocity = 0.002 m/s)

Shear stress is defined and the amount of force per unit area. In fluids shear stress is due to friction between particles caused by the viscosity of the fluids. The shear stress at the various flow boundary was calculated using the observed equations for the XY plot of X velocity versus position. Which was used to find du/dy and using the dynamic viscosity of water we were able to apply this to the following equation to find shear stress and various points along the flow boundary: $\tau = \mu du/dy$. Since the velocity followed a parabolic relation we found the tangential velocity at two points that found the average velocity between these two points. These two points are referred to as upper and lower interval. To have a better idea of how the following values were obtained, the equations for each boundary can be found in Appendix C of this report.

Table 16: Shear Stress for Boundary 1, x=10 (Laminar)

Lower Interval (m)	Upper Interval (m)	Lower Interval slope (1/s)	Upper Interval Slope (1/s)	Average slope (1/s)	Shear Stress (Pa)
0.1	0.2	-0.0002119	-0.000374	-0.00029295	2.93536E-07
0.2	0.3	-0.000374	-0.0005859	-0.00047995	4.8091E-07

0.3	0.4	-0.0005859	-0.0008272	-0.00070655	7.07963E-07
0.4	0.5	-0.0008272	-0.0011075	-0.00096735	9.69285E-07
0.5	0.6	-0.0011075	-0.0014364	-0.00127195	1.27449E-06
0.6	0.7	-0.0014364	-0.0018235	-0.00162995	1.63321E-06
0.7	0.8	-0.0018235	-0.0022784	-0.00205095	2.05505E-06
0.8	0.9	-0.0022784	-0.0028107	-0.00254455	2.54964E-06
0.9	1	-0.0028107	-0.00343	-0.00312035	3.12659E-06

Table 17: Shear Stress for Boundary 2, x=5 (Laminar)

Lower Interval (m)	Upper Interval (m)	Lower Interval slope (1/s)	Upper Interval Slope (1/s)	Average slope (1/s)	Shear Stress (Pa)
0.1	0.2	-0.0011394	-0.0012152	-0.0011773	1.18E-06
0.2	0.3	-0.0012152	-0.0016938	-0.0014545	1.457E-06
0.3	0.4	-0.0016938	-0.0020416	-0.0018677	1.871E-06
0.4	0.5	-0.0020416	-0.002225	-0.0021333	2.138E-06
0.5	0.6	-0.002225	-0.0022104	-0.0022177	2.222E-06
0.6	0.7	-0.0022104	-0.0019642	-0.0020873	2.091E-06
0.7	0.8	-0.0019642	-0.0014528	-0.0017085	1.712E-06
0.8	0.9	-0.0014528	-0.0006426	-0.0010477	1.05E-06
0.9	1	-0.0006426	0.0005	-7.13E-05	7.144E-08

Table 18: Shear Stress for Boundary 3, x=0 (Laminar)

Lower Interval (m)	Upper Interval (m)	Lower Interval slope (1/s)	Upper Interval slope (1/s)	Average slope (1/s)	Shear Stress (Pa)
0.1	0.2	-0.000139872	-0.000835504	-0.000487688	4.88663E-07
0.2	0.3	-0.000835504	-0.003060896	-0.0019482	1.9521E-06
0.3	0.4	-0.003060896	-0.009360528	-0.006210712	6.22313E-06
0.4	0.5	-0.009360528	-0.02458	-0.016970264	1.70042E-05

Table 19: Shear Stress for Boundary 4, x=-3 (Laminar)

Lower Interval (m)	Upper Interval (m)	Lower Interval slope (1/s)	Upper Interval slope (1/s)	Average slope (1/s)	Shear Stress (Pa)
0.1	0.2	-0.001359166	-0.001049712	-0.00120444	1.20685E-06
0.2	0.3	-0.001049712	-0.000290438	-0.00067007	6.71415E-07
0.3	0.4	-0.000290438	-0.006175984	-0.00323321	3.23968E-06
0.4	0.5	-0.006175984	-0.03460875	-0.02039237	2.04332E-05

Table 20: Shear Stress for Boundary 5, x=-5 (Laminar)

Lower Interval (m)	Upper Interval (m)	Lower Interval slope (1/s)	Upper Interval Slope (1/s)	Average slope (1/s)	Shear Stress (Pa)
0.1	0.2	0.0001	0.0002	0.00015	1.503E-07
0.2	0.3	0.0002	0.0003	0.00025	2.505E-07

0.3	0.4	0.0003	0.0004	0.00035	3.507E-07
0.4	0.5	0.0004	0.0005	0.00045	4.509E-07

Tables 16 to 20 show the shear stress in the fluid as it moves the channel from the entrance to exit for the laminar flow velocity =0.002 m/s. It can be clearly seen that for each interval along each of the boundary flow the shear stress increased as the interval got closer to the wall. This indicates that maximum shear stress is experienced along the wall of the channel as expected.

Turbulence Flow (Velocity = 0.2 m/s)

Shear Stress is primarily caused by friction between fluid particles, due to fluid viscosity. In order to calculate the shear stress through the entire channel, we have to look at our boundary lines as we saw in part E. Since we know that the equation for shear is $\tau=\mu(du/dy)$ where μ is the dynamic viscosity constant with the value of 0.001002 N *s/m^2. The value for du/dy we can obtained by calculating the equation of each boundary line using excel with their corresponding xy values. To have an accurate result, we had to try different polynomial fittings. Once we have completed this step, we can take the derivative of each equation with respect to y. Finally, we had to set our lower and upper to take the average using the derived equations which in other words would be the value for our du/dy. The data gathered for each boundary are being shown in the tables 21 to 25 and all the equations needed are shown in Appendix C of this report.

Table 21: Shear Stress Calculation for Boundary 1, x= 10 (Turbulent)

Lower Interval(m)	Upper Interval(m)	Lower interval Slope(1/s)	Upper Interval Slope(1/s)	Average Slope(1/s)	Shear Stress(Pa)
0.1	0.2	-0.027653224	-0.035338368	0.031495796	3.15588E-05
0.2	0.3	-0.035338368	-0.034386432	0.0348624	3.49321E-05
0.3	0.4	-0.027653224	-0.024880576	0.0262669	2.63194E-05
0.4	0.5	-0.024880576	-0.010575	0.017727788	1.77632E-05
0.5	0.6	-0.027653224	-1.3824E-05	0.013833524	1.38612E-05
0.6	0.7	-1.3824E-05	-0.007649968	0.003831896	3.83956E-06
0.7	0.8	-0.027653224	-0.054964032	0.041308628	4.13912E-05
0.8	0.9	-0.054964032	-0.171583176	0.113273604	0.0001135

Table 22: Shear Stress Calculation for Boundary 2, x= 5 (Turbulent)

Lower Interval(m)	Upper Interval(m)	Lower interval Slope(1/s)	Upper Interval Slope(1/s)	Average Slope(1/s)	Shear Stress(Pa)
0.1	0.2	-0.0588246	-0.1022248	0.0805247	8.06857E-05
0.2	0.3	-0.1022248	-0.1419302	0.1220775	0.000122322
0.3	0.4	-0.1419302	-0.1762704	0.1591003	0.000159419
0.4	0.5	-0.1762704	-0.203575	0.1899227	0.000190303
0.5	0.6	-0.203575	-0.2221736	0.2128743	0.0002133
0.6	0.7	-0.2221736	-0.2303958	0.2262847	0.000226737
0.7	0.8	-0.2303958	-0.2265712	0.2284835	0.00022894
0.8	0.9	-0.2265712	-0.2090294	0.2178003	0.000218236

Table 23: Shear Stress Calculation for Boundary 3, x= 0 (Turbulent)

Lower Interval(m)	Upper Interval(m)	Lower interval Slope(1/s)	Upper Interval Slope(1/s)	Average Slope(1/s)	Shear Stress(Pa)
0.1	0.2	-0.16032025	-0.0803112	0.120315725	0.000120556
0.2	0.3	-0.0803112	0.17461835	0.047153575	4.72479E-05
0.3	0.4	0.17461835	-0.1323976	0.021110375	2.11526E-05
0.4	0.5	-0.1323976	-2.73770625	1.435051925	0.001437922

Table 24: Shear Stress for Boundary 4, x= -3 (Turbulent)

Lower Interval(m)	Upper Interval(m)	Lower interval Slope(1/s)	Upper Interval Slope(1/s)	Average Slope(1/s)	Shear Stress(Pa)
0.1	0.2	-0.108188	-0.210752	0.15947	0.000159789
0.2	0.3	-0.210752	-0.312092	0.261422	0.000261945
0.3	0.4	-0.312092	-0.412208	0.36215	0.000362874
0.4	0.5	-0.412208	-0.5111	0.461654	0.000462577

Table 25: Shear Stress for Boundary 5, x=-5 (Turbulent)

Lower Interval(m)	Upper Interval(m)	Lower interval Slope(1/s)	Upper Interval Slope(1/s)	Average Slope(1/s)	Shear Stress(Pa)
0.1	0.2	-0.0101	-0.0202	0.01515	1.51803E-05
0.2	0.3	-0.0202	-0.0303	0.02525	2.53005E-05
0.3	0.4	-0.0303	-0.0404	0.03535	3.54207E-05
0.4	0.5	-0.0404	-0.0505	0.04545	4.55409E-05

IV. Comparing the Flow with each Entrance Velocity.

In Fluid Mechanics, it is important in learning how fluids flow in a particular environment. When a fluid is flowing in a pipe or channel, it can be described as either a laminar or turbulent flow depending on its velocity. The goal of this exercise was to examine the differences of each flow when run on FLUENT with their respective entrance velocities, 0.002 m/s for laminar and 0.2 m/s for turbulent flow.

The primary characteristic of laminar flow is a streamlined flow, lacking any swirls or cross currents. If one imagines different layers of a fluid, divided into rows/cylinders with varying radii, the layers of the fluid wouldn't mix in any manner. The fluid would flow without interference or disturbance, and the path of the flow wouldn't have any swirls or cross currents. Based on Figures 21 and 22, it can be seen that at such low velocity (0.002m/s) the fluid seemed to flow without lateral mixing and the adjacent layers slide past one another. The layers of the fluids flowed at various speeds as indicated by the different colors of the flow with red representing the maximum velocity and blue represents almost zero velocity. It can be seen that the layers in the center of the channel had a greater velocity compared to the layers on the outer edges of the channel. Overall, however, the velocity of the fluid flow is very low.

In this flow the fluid is assumed to be passing in layers through passage (here we assume a cylindrical pipe). This flow occurs when the Reynolds number of the fluid relating velocity and characteristic length of pipe is less than(<) 2100. The action of friction between the layers decreases radially inwards as the friction provided by the pipe would be maximum which is ceases the flow of the outermost layer which tries to cease the layer next to it but never negotiates the motion. Finally the innermost layer would be having the highest velocity as it is far from wall of the pipe.

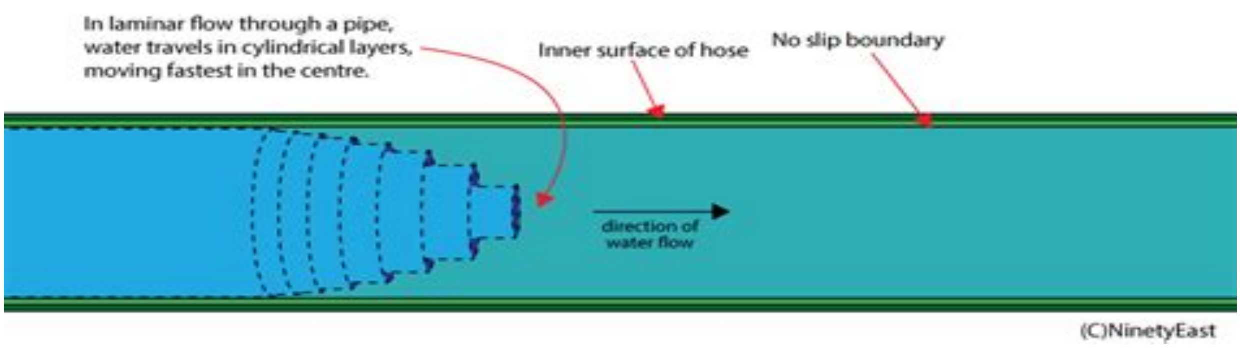

***Figure 24**: Laminar flow through a pipe.*

But if we consider the thickness (x) of layers to be minimum (dx−>0) then the trajectory of the fluid radially would be parabolic along the length of the pipe as shown in the figure below.

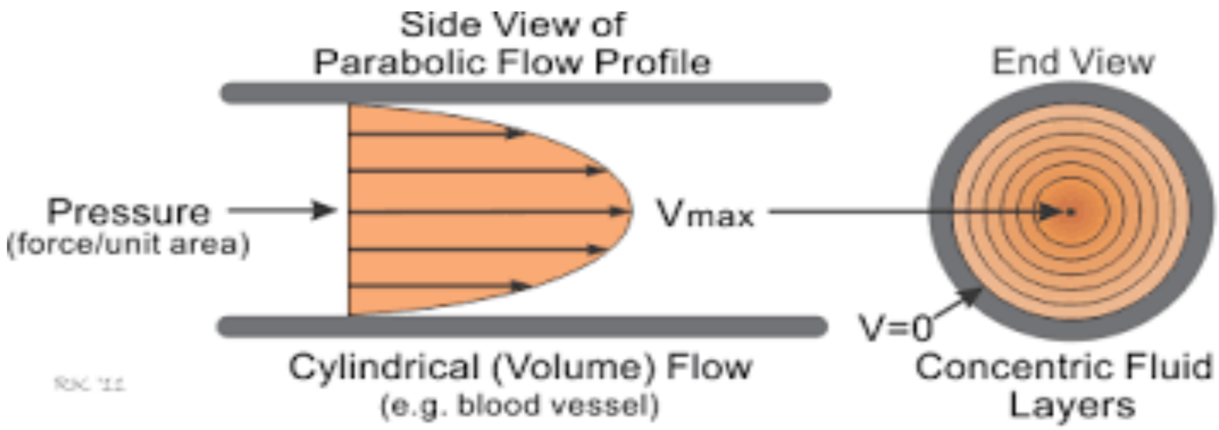

Figure 25: *A side view of parabolic flow profile for a laminar flow(fluids and fluids flow).*

Turbulent flow, on the other hand, can be described as a flow that is chaotic. Many scientists and researchers today still haven't fully understood the way turbulent flows due to how random and out of order the fluid flows. Unlike laminar flow, the fluid experiencing turbulent flow has layers that intersect each other and do not run parallel to one another. They occur in channels of large diameters where the velocity is high. Turbulent flow is any pattern of fluid motion characterized by chaotic changes in pressure and flow velocity. Turbulence is caused by excessive kinetic energy in parts of a fluid flow, which overcomes the damping effect of the fluid's viscosity.

When the velocity of the fluid reaches a value such that the Reynolds number gets a value greater than (>) 4000. There would be intermixing between the layers which causes its deviation from the laminar flow.

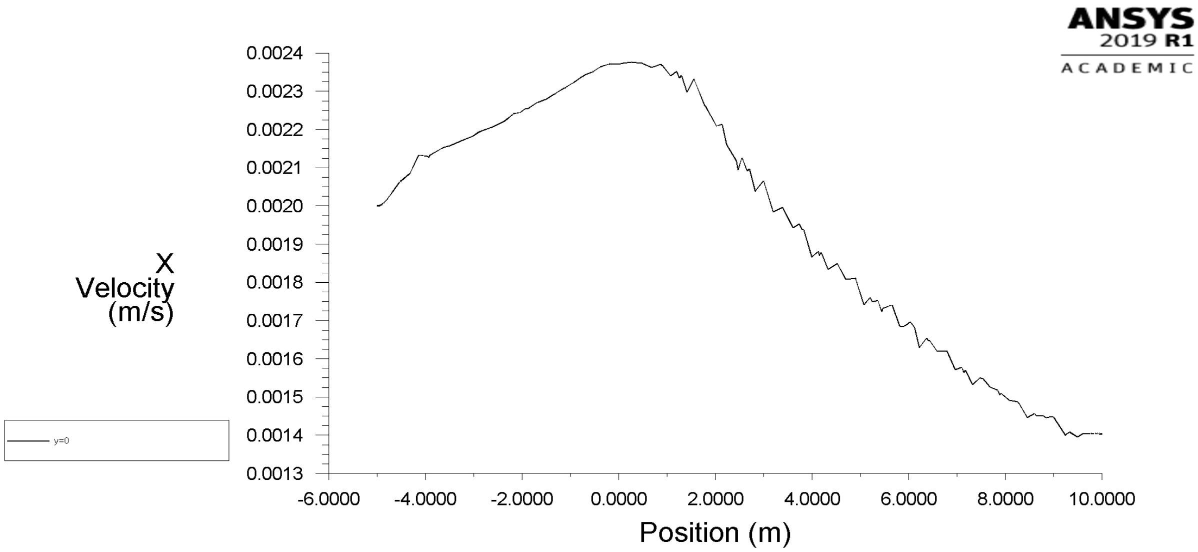

Table 26: *Showing the velocity vs x position along the channel for Laminar Flow.*

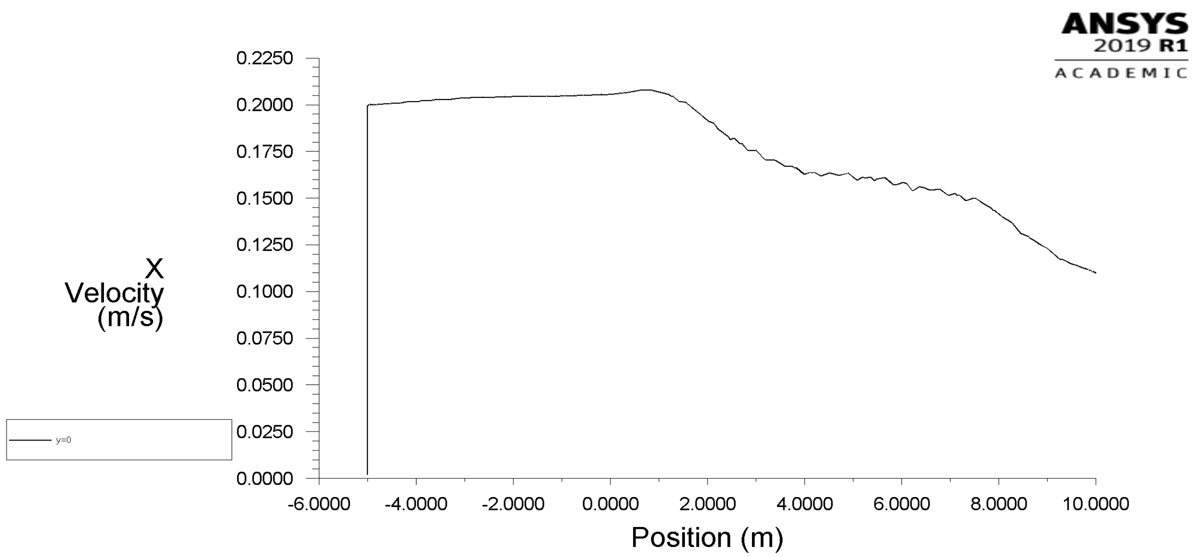

Table 27 *Showing the velocity vs x position along the channel for turbulent flow.*

Figure 26 and figure 27 shows us how the velocity changes for each type of flow from entrance to exit. We can clearly see that both figures follow a distinct pattern. Figure 26, shows the velocity changes for Laminar flow. We clearly see that the velocity starts to increase form the entrance to where the channel expands which then follows by a rapid decrease. However, figure 27 shows how the velocity changes for Turbulent flow. We can see that the velocity is relatively constant until where the channel expands where the velocity starts to decrease. The decrease in the velocity after the expansion was not as rapid as the decrease observed for laminar flow.

V. Conclusion

The fact that FLUENT is able to model such a hypothetical system is fairly remarkable. It is even more remarkable that it is diverse as it and the scale of accuracy that it is able to handle out. The main principle for the continuous property of fluids holds true under this software rendering of a system and the ability to model such complex principles such as momentum principle. The process of modeling and analyzing this system has produced a more intuitive knowledge of the behavior of fluids and has increased the student's understanding of utilizing such technology in analyzing problems which will serve well in future engineering endeavors.

References

[1]Crowe, C.T., et al. *Engineering Fluid Mechanics.* Wiley, 2006.
[2]*Fluid mechanics fundamentals and applications*.Çengel, Y. A., & Cimbala, J. M. (2014). New York, NY: McGraw-Hill.
[3]Workbench ANSYS 19.2 Fluent Quick Tutorial, CE Dept. CCNY, 2019.
[4] Dong, W. (2019, March 6). Retrieved February 22, 2019, from https://www.youtube.com/watch?v=s5SldjMm-r4&feature=yout

Image Citations

Fluids and Fluid Flow. (n.d.). Retrieved from https://www.engr.colostate.edu/CBE101/topics/fluids_and_fluid_flow.html

What-is-the-difference-between-laminar-and-turbulent-flow.(n.d). Retrieved from https://www.quora.com/What-is-the-difference-between-laminar-and-turbulent-flow.html

Appendix

Appendix A: Derivation for Vorticity Equation

Recalling that du/dt and dv/dt are x and y component momentum equations respectively, we can start our derivation by taking ∂/∂x[dv/dt] - ∂/∂y[du/dt]

$$\frac{\partial}{\partial x}[\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}+fu = -\frac{1}{\rho}\frac{\partial p}{\partial y}] - \frac{\partial}{\partial y}[\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}-fv = -\frac{1}{\rho}\frac{\partial p}{\partial x}]$$

$$\frac{\partial}{\partial x}\frac{\partial v}{\partial t}+u\frac{\partial 2v}{\partial x2}+\frac{\partial v}{\partial x}\frac{\partial u}{\partial x}+v\frac{\partial 2v}{\partial x\partial y}+\frac{\partial v}{\partial y}\frac{\partial v}{\partial x}+w\frac{\partial 2v}{\partial x\partial z}+\frac{\partial v}{\partial z}\frac{\partial w}{\partial x}+f\frac{\partial u}{\partial x}+u\frac{\partial}{\partial x} = -\frac{1}{\rho}\frac{\partial 2p}{\rho\partial x\partial y}+\frac{1}{\rho 2}(\frac{\partial p}{\partial x}\frac{\partial \rho}{\partial x})$$

$$-\frac{\partial}{\partial y}\frac{\partial u}{\partial t}+u\frac{\partial 2u}{\partial x\partial y}+\frac{\partial u}{\partial x}\frac{\partial u}{\partial y}+v\frac{\partial 2u}{\partial y2}+\frac{\partial u}{\partial y}\frac{\partial v}{\partial y}+w\frac{\partial 2u}{\partial y\partial z}+\frac{\partial u}{\partial z}\frac{\partial w}{\partial y}-f\frac{\partial v}{\partial y}-v\frac{\partial f}{\partial y} = -\frac{1}{\rho}\frac{\partial 2p}{\rho\partial x\partial y}+\frac{1}{\rho 2}(\frac{\partial p}{\partial x}\frac{\partial \rho}{\partial y})$$

$$\frac{\partial}{\partial t}(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y})+u\frac{\partial}{\partial x}(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y})+v\frac{\partial}{\partial y}(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y})+w\frac{\partial}{\partial z}(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y})+(\frac{\partial v}{\partial x}-\frac{\partial u}{\partial y})(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y})+f(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}) +(\frac{\partial w}{\partial \square}\frac{\partial v}{\partial z}-\frac{\partial w}{\partial y}\frac{\partial u}{\partial z})+v\frac{\partial f}{\partial y} = \frac{1}{\rho 2}(\frac{\partial p}{\partial y}\frac{\partial \rho}{\partial x}-\frac{\partial p}{\partial x}\frac{\partial \rho}{\partial y})$$

$$\frac{\partial \zeta}{\partial t}+u\frac{\partial \zeta}{\partial x}+v\frac{\partial \zeta}{\partial y}+w\frac{\partial \zeta}{\partial z}+\zeta(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y})+f(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y})+(\frac{\partial w}{\partial x}\frac{\partial v}{\partial z}-\frac{\partial w}{\partial y}\frac{\partial u}{\partial z}) +v\frac{\partial f}{\partial y} = \frac{1}{\rho 2}(\frac{\partial p}{\partial y}\frac{\partial \rho}{\partial x}-\frac{\partial p}{\partial x}\frac{\partial \rho}{\partial y})$$

Vorticity: $\frac{d}{dt}(\zeta+f) = -(\zeta+f)(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}) - (\frac{\partial w}{\partial x}\frac{\partial v}{\partial z}-\frac{\partial w}{\partial y}\frac{\partial u}{\partial z})+\frac{1}{\rho 2}(\frac{\partial p}{\partial y}\frac{\partial \rho}{\partial x}-\frac{\partial p}{\partial x}\frac{\partial \rho}{\partial y})$

Appendix B: Obtaining Shear Stress Equation

Let us consider the parallel motion of fluid where all particles are moving in the same direction, but different layers have different velocities. After a small time Δt the fluid volume abcd moves to a ′ b ′ c ′d ′ as described in Figure 28, where |aa′ | = |bb′ | = u(y + δy)Δt and |cc′ | = |dd′ | = u(y)Δt. The corresponding shear strain is:

$$\gamma = \frac{\Delta x}{\delta y} = \frac{(u(y + \delta y) - u(y))\,\Delta t}{\delta y}$$

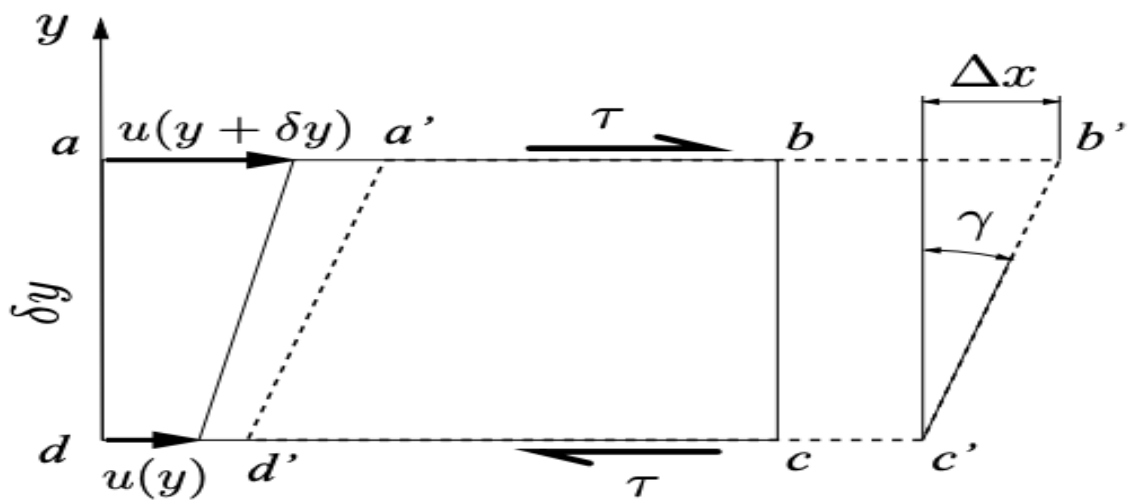

Figure 26: *Control volume for analysis of uniform flow.*

For small change in t, the strain can be expressed via its rate of change as: $\gamma = \frac{d\gamma}{dt}\Delta t$
We can then write: $d\gamma/dt = u(y + \delta y) - u(y)/\delta y$ and for small δy we get $d\gamma$/dt=du/dy
For a parallel flow of a Newtonian fluid shear stress is proportional to the gradient of velocity in the direction perpendicular to the flow that is: $\tau = \mu \frac{du}{dy}$

Appendix C: Necessary Tables to get Shear Stress Results

The following equations show the relationship between the velocity of the flow and the traveled distance. Each equation helped us obtain du/dy for the shear stress.

Table 28: *Boundary Equations for Laminar Velocity(0.002 m/s).*

Position x(m)	Laminar Flow boundaries Equations	Derivative for each Boundary
10	$u = -0.0001y^5 - 0.0004y^4 + 0.0001y^3 - 0.0009y^2 + 2E\text{-}07y + 0.0014$	$du/dy = -0.0005y^4 - 0.0016y^3 + 0.0003y^2 - 0.0018y + 2E\text{-}07$
5	$u = 0.0003y^6 - 2E\text{-}05y^5 + 0.001y^4 + 0.0005y^3 - 0.0031y^2 - 0.0005y + 0.0018$	$du/dy = 0.0018y^5 - 1E\text{-}04y^4 + 0.004y^3 + 0.0015y^2 - 0.0062y - 0.0005$
0	$u = -0.1112y^6 + 0.0084y^5 - 0.0081y^4 - 0.0025y^3 - 0.0005y^2 + 7E\text{-}05y + 0.0024$	$du/dy = -0.6672y^5 + 0.042y^4 - 0.0324y^3 - 0.0075y^2 - 0.001y + 7E\text{-}05$
-3	$u = -0.4086y^6 + 0.0015y^5 + 0.1009y^4 - 0.0003y^3 - 0.0087y^2 + 1E\text{-}05y + 0.0023$	$du/dy = -2.4516y^5 + 0.0075y^4 + 0.4036y^3 - 0.0009y^2 - 0.0174y + 1E\text{-}05$
-5	$u = -0.0005y^2 - 3E\text{-}17y + 0.002$	$du/dy = -0.001y - 3E\text{-}17$

Table 29: *Boundary Equations for Turbulent Velocity(0.2 m/s).*

Position x(m)	Laminar Flow boundaries Equations	Derivative for each Boundary
10	$u = -0.1554y^6 - 0.0262y^5 + 0.1808y^4 + 0.0382y^3 - 0.0785y^2 - 0.013y + 0.1121$	$du/dy=-0.932y^5 -0.131y^4 +0.723y^3 +0.114y^2 -0.157y-0.013$
5	$u = 0.0696y^4 + 0.0059y^3 - 0.2294y^2 - 0.0134y + 0.1609$	$du/dy=0.2784y^3 +0.0177y^2 -0.4588y-0.0134$
0	$u = -46.545y^6 + 0.4909y^5 + 14.065y^4 - 0.1631y^3 - 1.0812y^2 + 0.0071y + 0.2181$	$du/dy=-46.54y^5 +2.45y^4 +56y^3 -0.489y^2 -2.16y+0.0071$
-3	$u = 0.0204y^3 - 0.522y^2 - 0.0044y + 0.2261$	$du/dy=-0.0612y^2 -1.044y-0.0044$
-5	$u = -3E\text{-}14y^3 - 0.0505y^2 + 6E\text{-}14y + 0.2032$	$du/dy=-9E\text{-}14y^2 -0.101y+6E\text{-}14$