Task #1

Problem Statement: The objective of the first task of project 3 is to run a one hour transient simulation of a cylindrical disk for both a laminar and turbulent flow with water and air respectively in a virtually wind tunnel. The inlet velocity is set to 0.002 m/s and 0.5 m/s for water and air respectively with a step size of 1 sec. A fine mesh with a mesh refinement outlined with the red box was used to run the simulation. The following diagrams represents the geometry of the system and the mesh used:

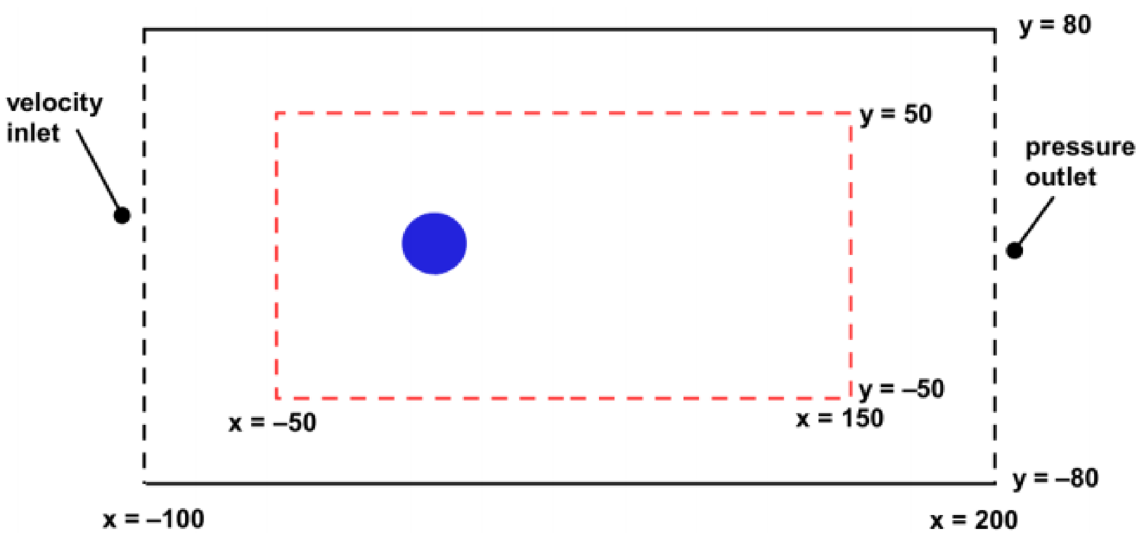


Figure 1: Geometry of task 1

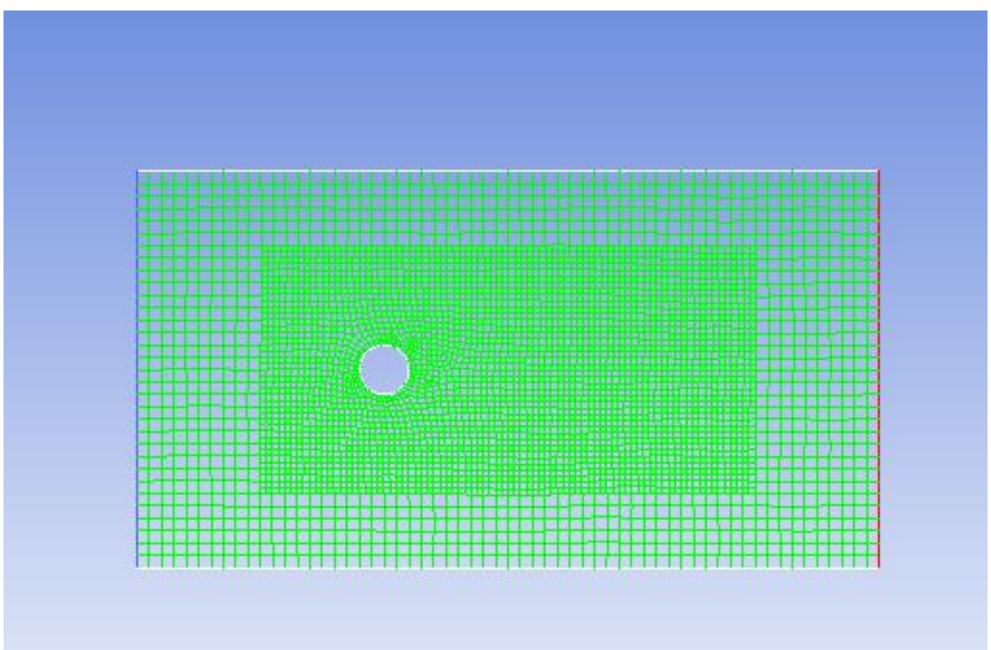


Figure 2: Mesh used for task 1

Results:

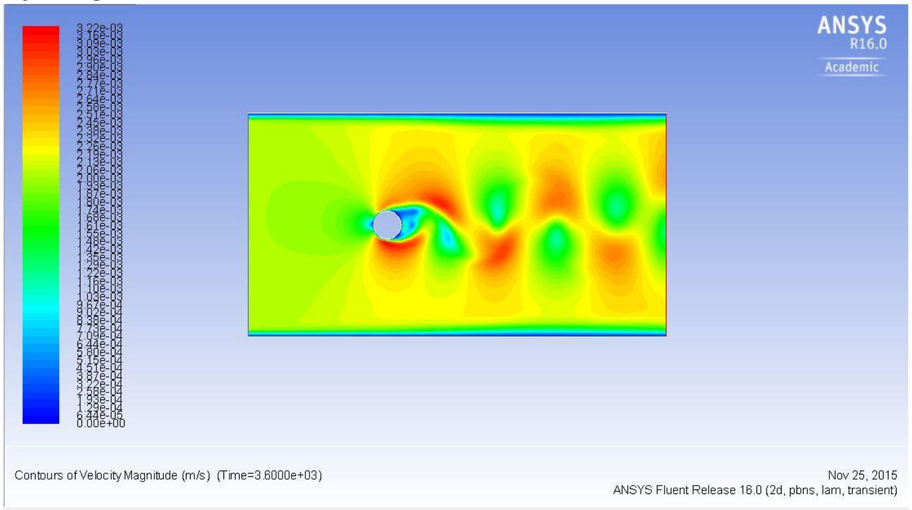
Laminar Case

1. **Reynolds Number**

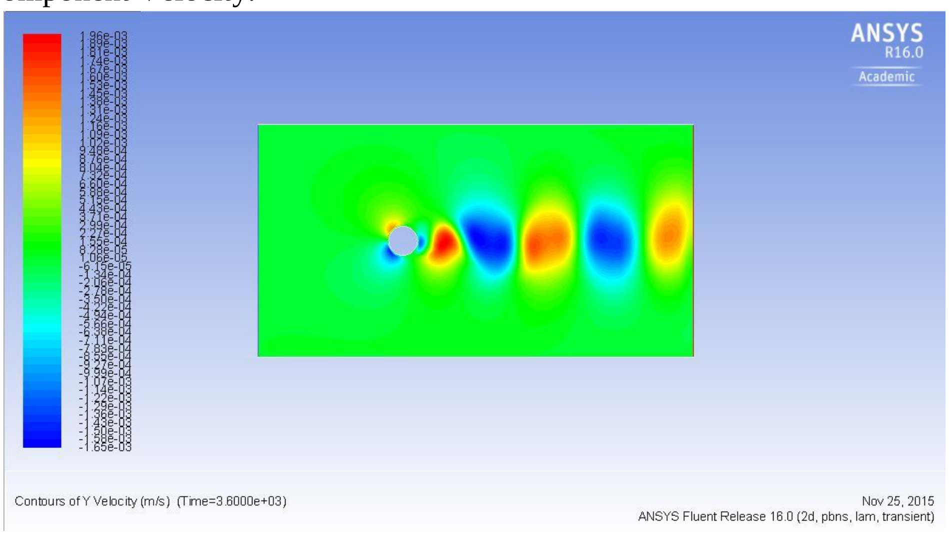
$$Reynolds\ number = \frac{\varphi VD}{\mu} = \frac{998.2\frac{kg}{m^3} * .002\frac{m}{s} * .2m}{0.001003\frac{kg}{ms}} = 398.086$$

2. **Contour Plots:**

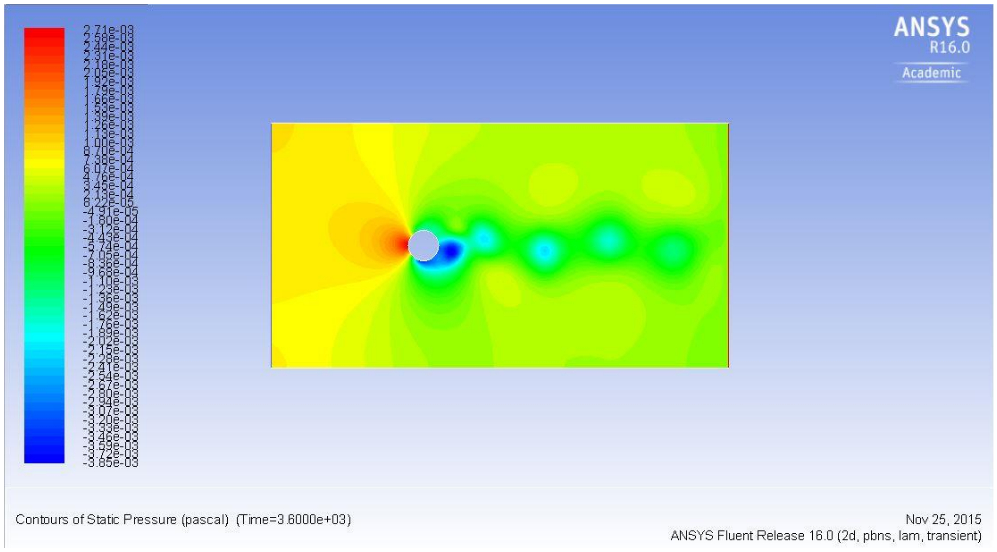
   i. Velocity Magnitude:



   ii. Y-Component Velocity:

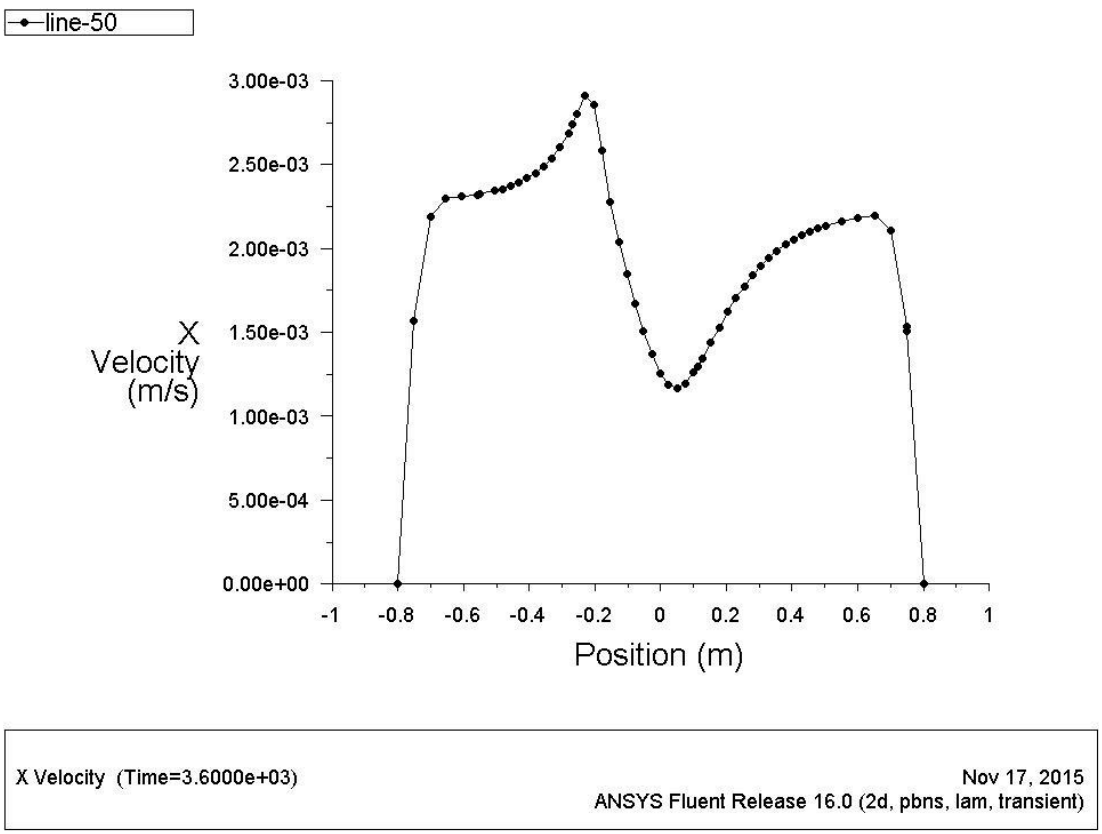


iii. Pressure:

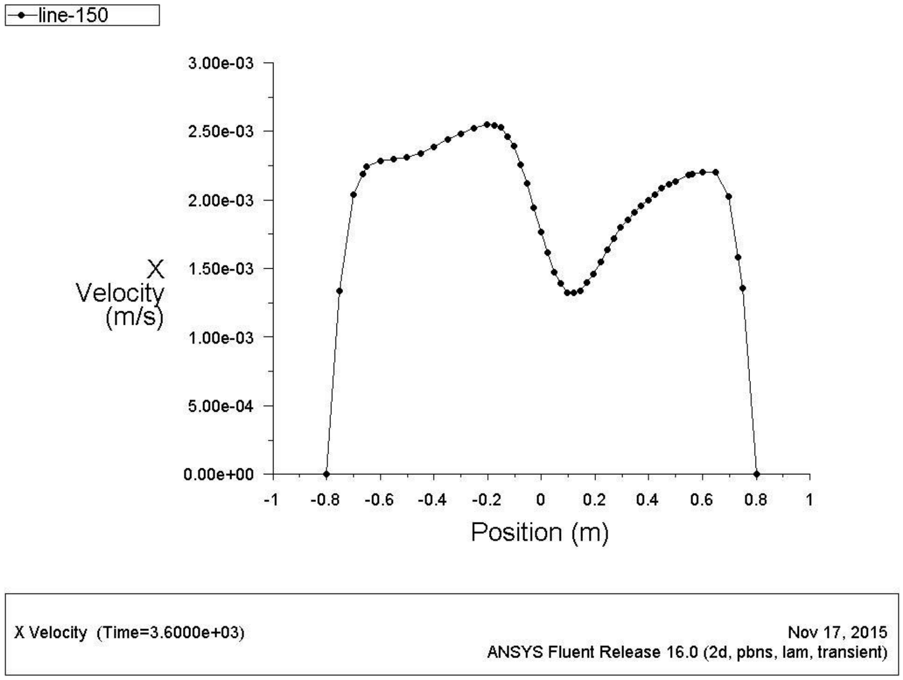


3. Line Plots:

i. x=50 cm.



ii. x=150 cm.



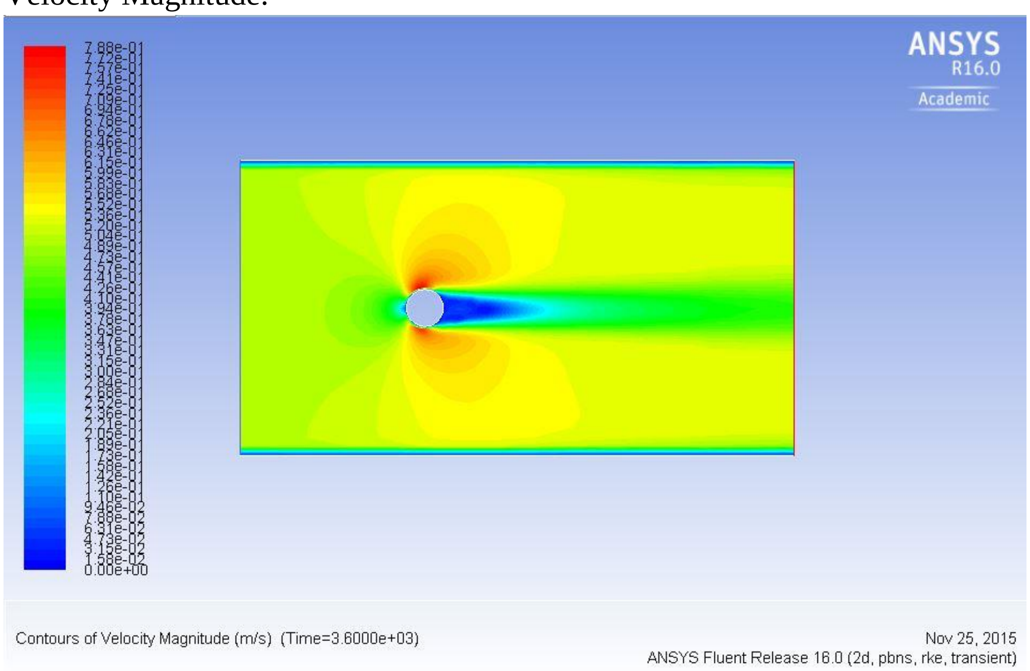
Turbulent Flow

1. Reynolds Number

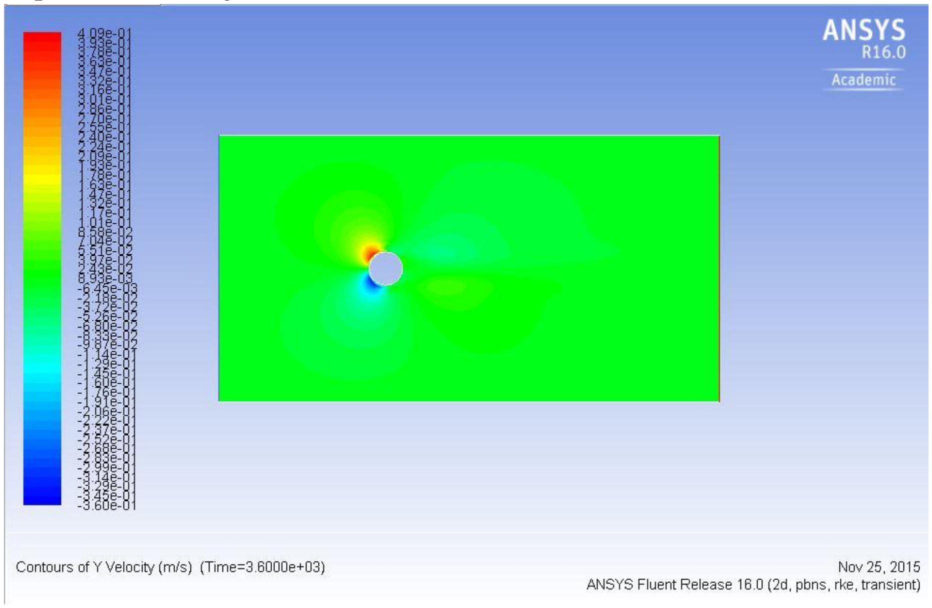
$$Reynolds\ number = \frac{\varphi V D}{\mu} = \frac{1.225\frac{kg}{m^3} * 0.5\frac{m}{s} * .2m}{1.789 * 10^{-5}\frac{kg}{ms}} = 6845.87$$

2. Contour Plots

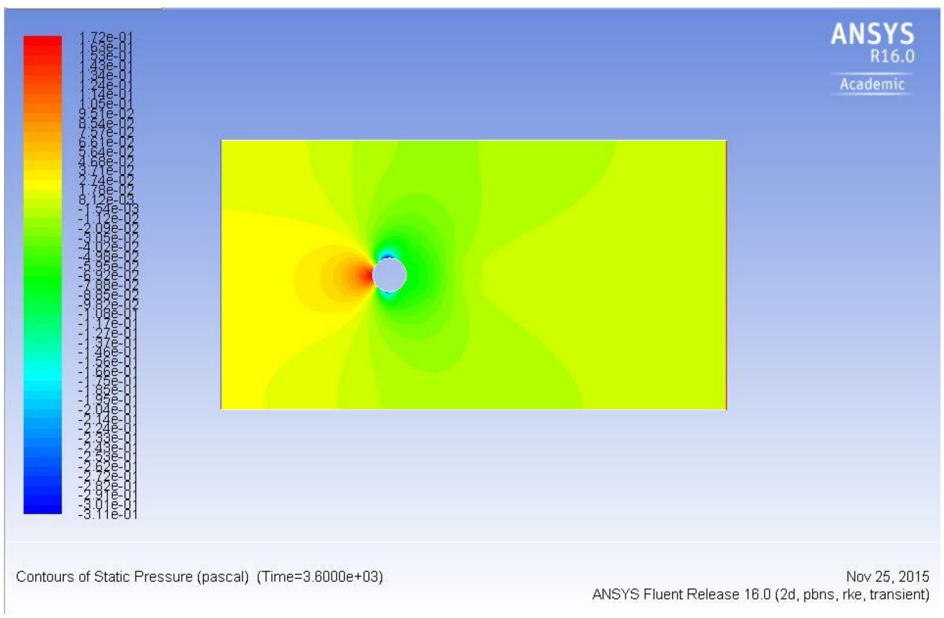
i. Velocity Magnitude:



ii. Y-Component Velocity

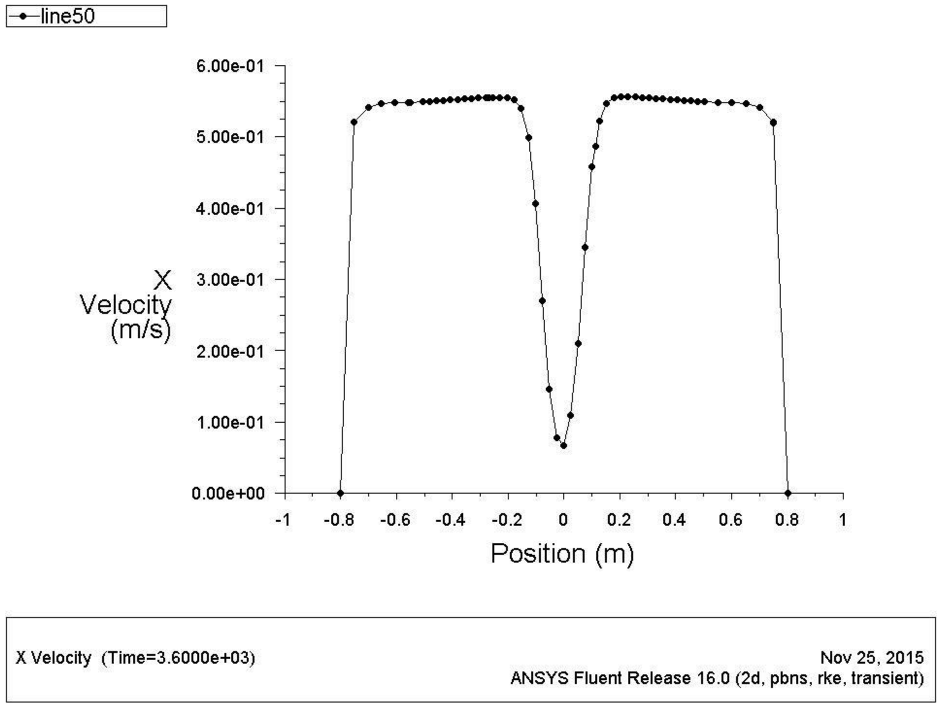


iii. Pressure:

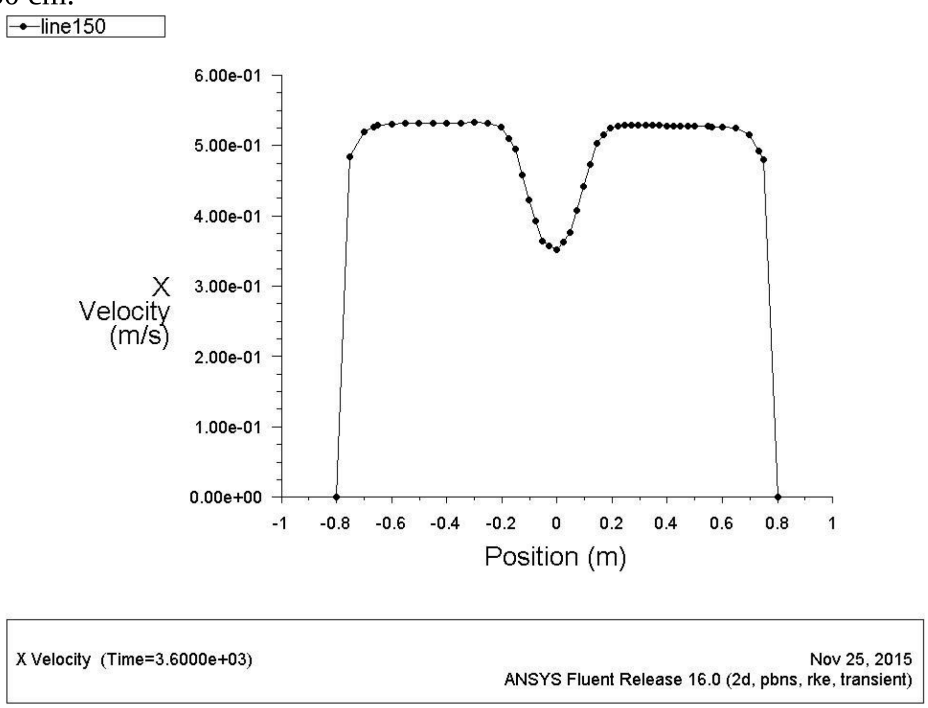


3. Line Plots

i. x=50 cm.

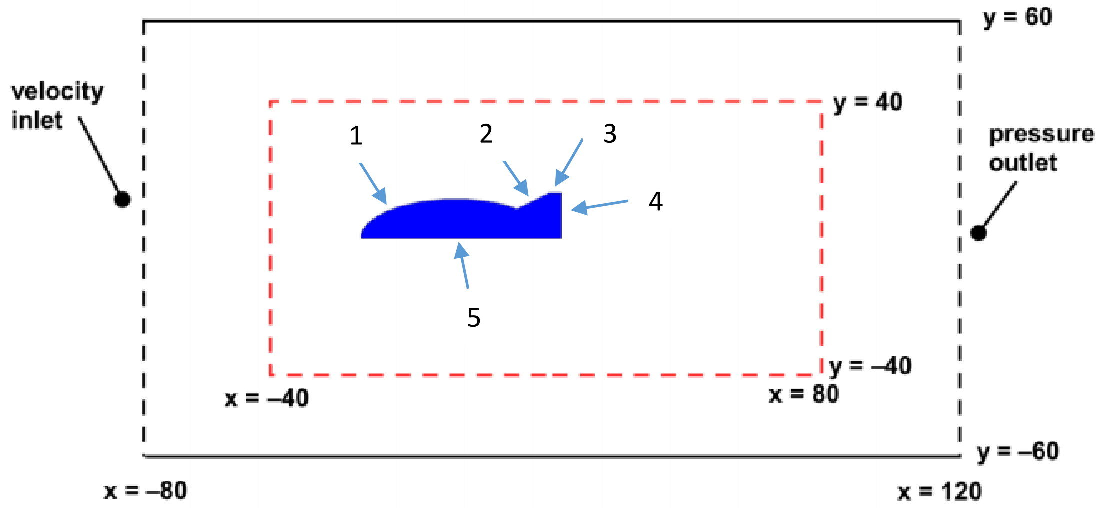


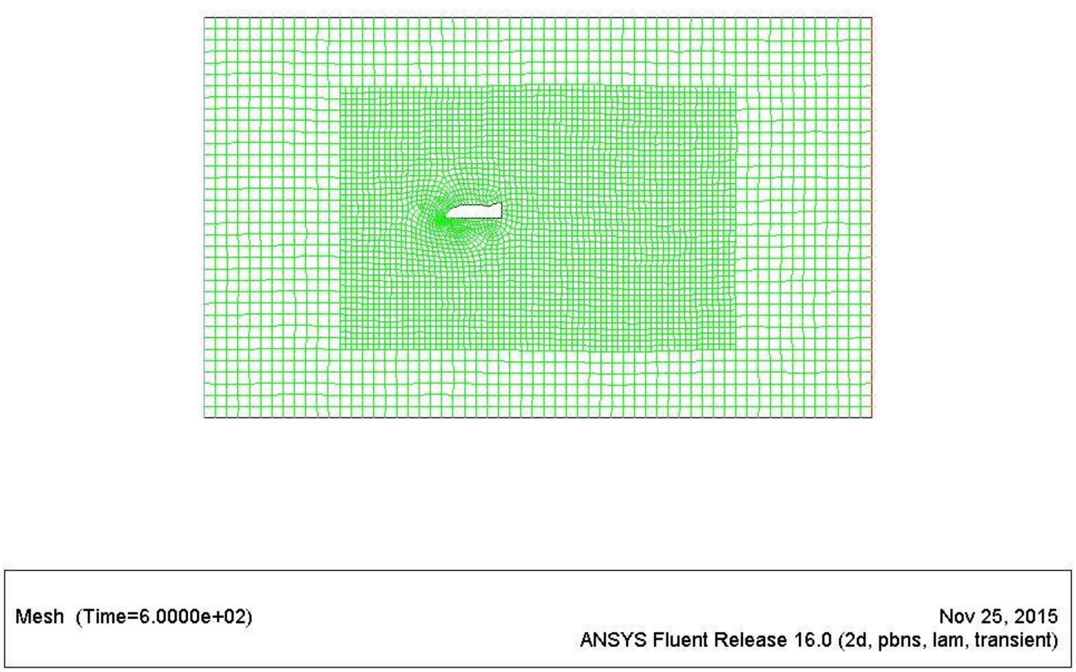
ii. x=150 cm.



Task #2

Problem Statement: The objective of task 2 is to run a virtual wind tunnel simulation for a half fish model with an x-velocity of 0.0025 m/s. The simulation was ran for 10 minutes with a 1 second time step. The setup is very similar to that of task 1 except it is only ran with a laminar flow with water as the fluid. A fine mesh with a refinement similarly to that of task 1 was used. The following figures represent both the geometry and the mesh:

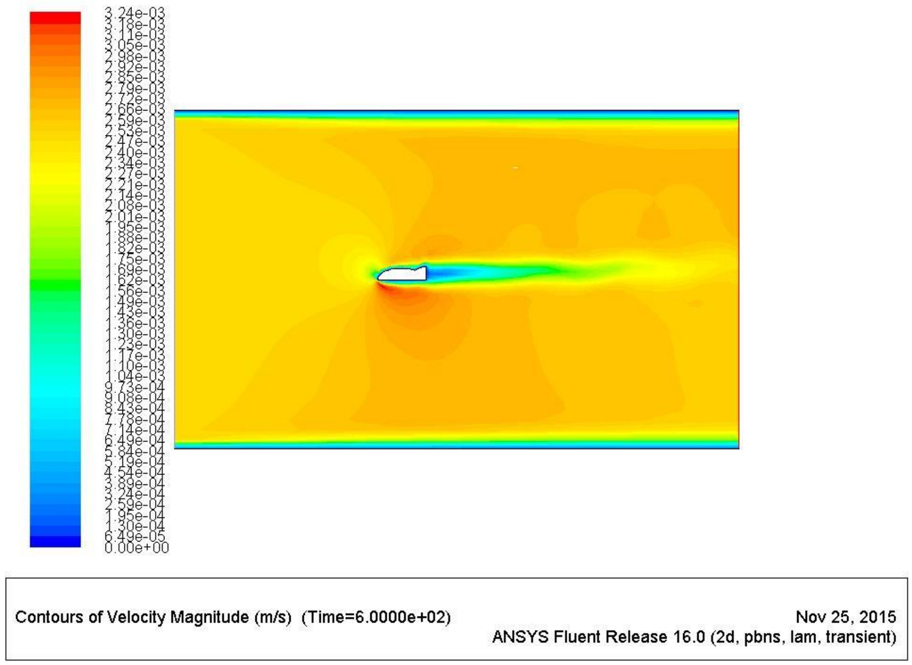




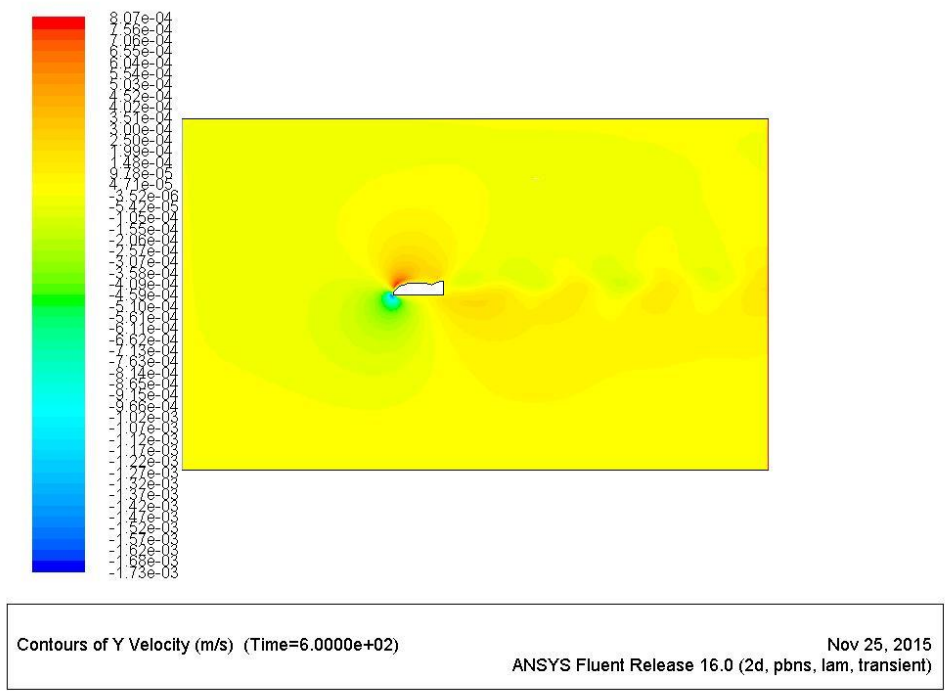
Results:

1. **Contour Pots:**

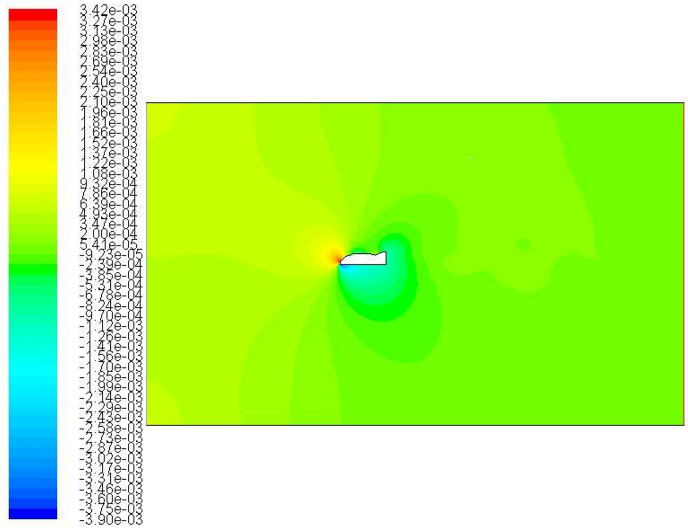
    i. Velocity Magnitude:



    ii. Y-Component Velocity:



iii. Pressure:



Contours of Static Pressure (pascal) (Time=6.0000e+02) Nov 25, 2015
ANSYS Fluent Release 16.0 (2d, pbns, lam, transient)

2. Lift:

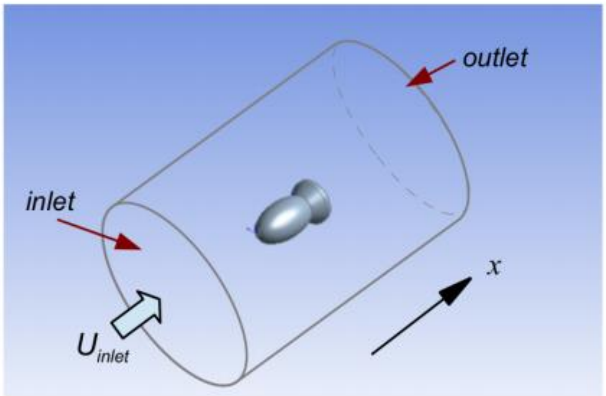
```
------------------------ --------------------------------------- --------------------------------------- ------------------
Net                      (0.00010870101 -0.0002946569 0)         (6.0682389e-05 2.0703587e-06 0)         (0.0001693834 -0.

Forces - Direction Vector (0 1 0)
                         Forces (n)                                Coefficients
Zone                     Pressure        Viscous         Total           Pressure        Viscous         Total
wall-1                   -1.5796018e-05  7.1155982e-06   -8.6804203e-06  -2.5789418e-05  1.1617303e-05   -1.4172115e-05
wall-2                   2.2948411e-06   -7.9315828e-07  1.5016828e-06   3.7466794e-06   -1.2949523e-06  2.4517271e-06
wall-3                   7.0227857e-06   4.0289274e-07   7.4256784e-06   1.1465773e-05   6.5778406e-07   1.2123557e-05
wall-4                   0               6.1059018e-07   6.1059018e-07   0               9.9688192e-07   9.9688192e-07
wall-5                   -0.0002881785   -5.2655641e-06  -0.00029344407  -0.00047049552  -8.5968394e-06  -0.00047909236
------------------------ --------------  --------------  --------------  --------------  --------------  --------------
Net                      -0.0002946569   2.0703587e-06   -0.00029258654  -0.00048107248  3.3801775e-06   -0.00047769231
```

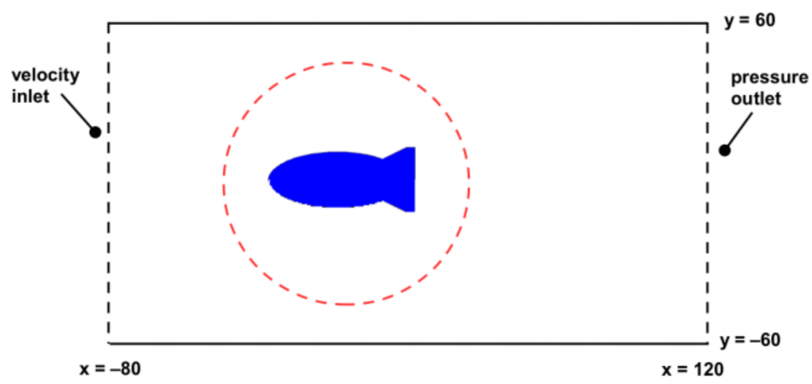
Wall Number	Pressure (N)	Viscous (N)	Total (N)
Wall-1	-1.5796e-05	7.1155982e-06	-8.6804203e-06
Wall-2	2.2948411e-06	-7.9315828e-07	1.5016828e-06
Wall-3	7.0227857e-06	4.0289274e-07	7.4256784e-06
Wall-4	0	6.1059018e-07	6.1059018e-07
Wall-5	-.0028817585	-5.2655641e-06	-0.00029344407
Net	-.0002946569	2.0703587e-06	-0.00029258654

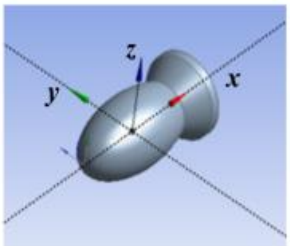
Discussion: From the results of the pressure contour it can be seen that the net lift on the object is in the negative downward direction. This makes sense when looking at the contour plot because it shows the pressure on the top wall is much higher than the pressure on the bottom wall. The table also confirms this with the value of the force caused by the pressure on wall 1 being negative meaning it is acting in the downward direction. Another interesting thing worth noting the net pressure force on the bottom wall is also negative, this is due to the low pressure sucking down the object and generating a downward force. Also wall 4 has zero net force from pressure which makes sense since the fluid would not be hitting that wall. Overall the contour plots match the table's results.

Task #3

Problem Task: The objective to task 3 is to change the 2D half fish model to a full 3D model with a cylinder as a wind tunnel. A viscous-laminar model with water as the working fluid was used. A transient simulation was ran for 5 minutes with a 1 second step size. A fine mesh was used along with a mesh refinement in the shape of a 20 cm radius sphere centered at (x,y,z)=(0,0,0). The below diagrams represent the geometry of the system:

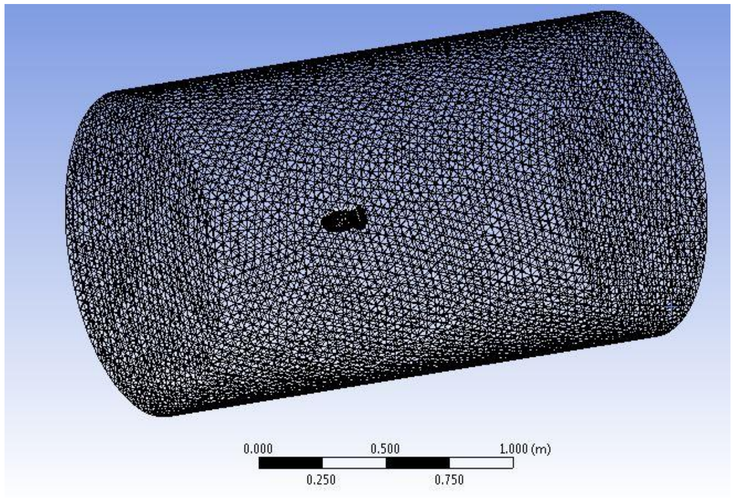


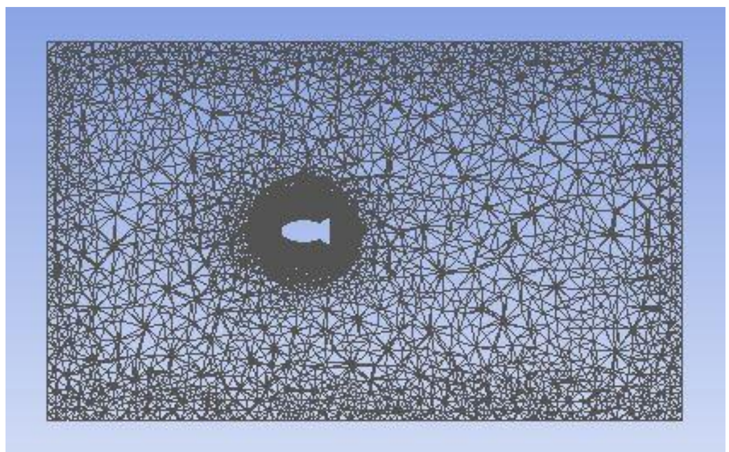




Results:

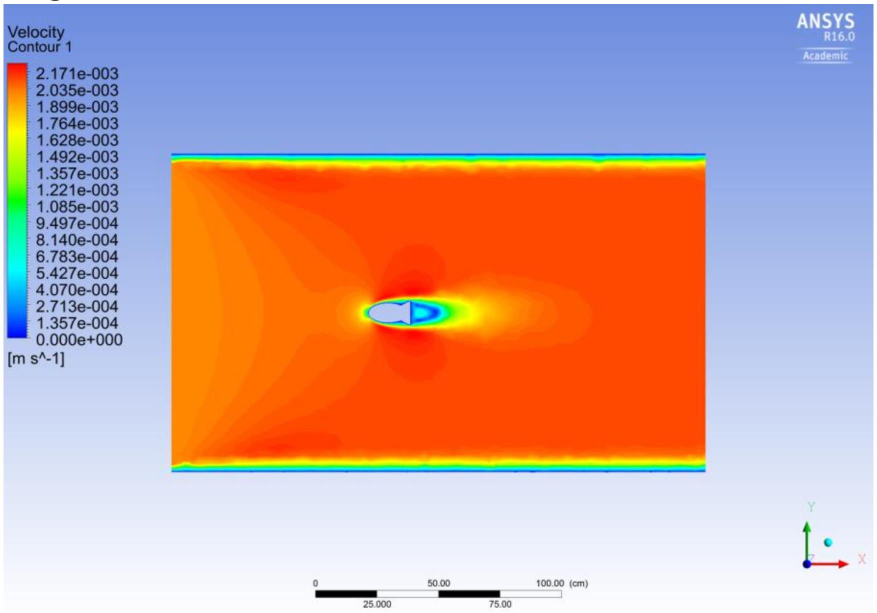
1. **Mesh:**



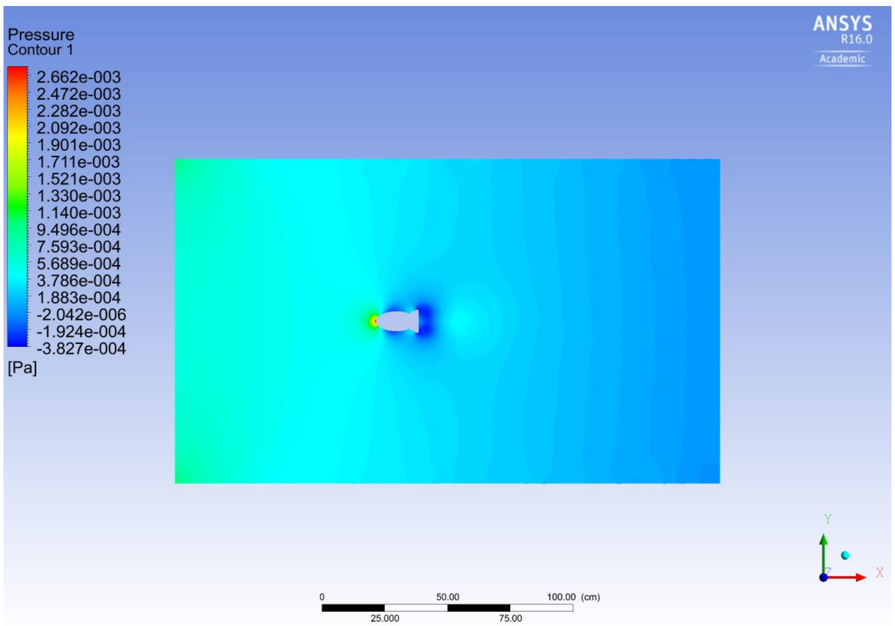


2. Contour Plots:

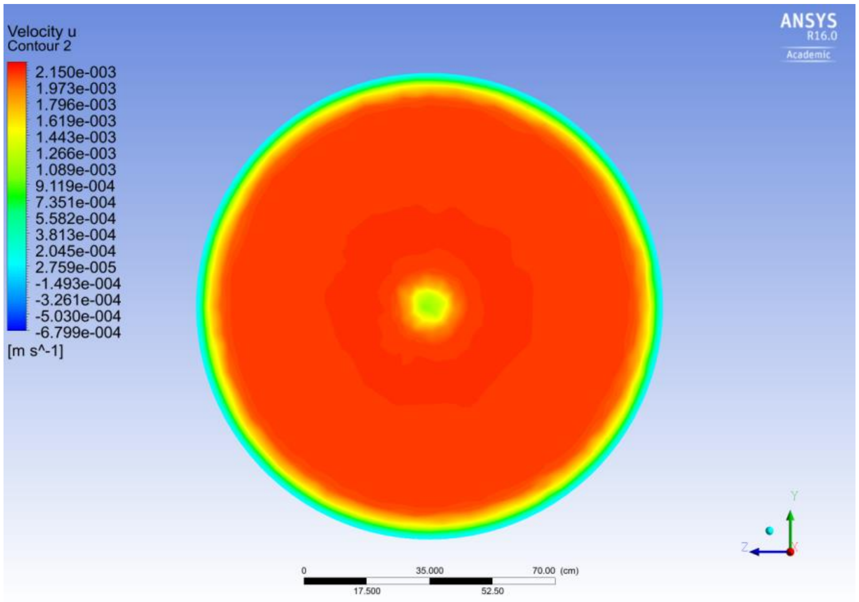
i. Velocity Magnitude:



ii. Pressure:



iii. X-Component Velocity at x=25 cm.



Discussion: From the above plots it can be seen that the symmetrical shape of the half model produces a symmetrical flow and pressure variation around to body. It can be seen that as the fluid travels around the object it increases in velocity which is expected due to the continuity principle. Another interesting thing to point as is that at the locations (top and bottom surface) of where the velocity increases the pressure decreases which is expected due Bernoulli's principle.

Task #4

Problem Statement: The objective to task 4 is change the uniform inlet velocity of task 3 to a parabolic profile. The same inlet velocity and setup of task 3 was used including the mesh refinement and time step. To change the velocity profile a user defined function (UDF) was used. The following equation was programed using Visual Studio:

$$V(r) = V_{max}\left(1 - \left(\frac{r}{R}\right)^2\right)$$

$$V_{max} = 2 * V$$

$$r = y^2 + z^2$$

Results:

1. **User Defined Function:** Using Visual Studio and Challenge 2 as an example the following UDF was constructed:

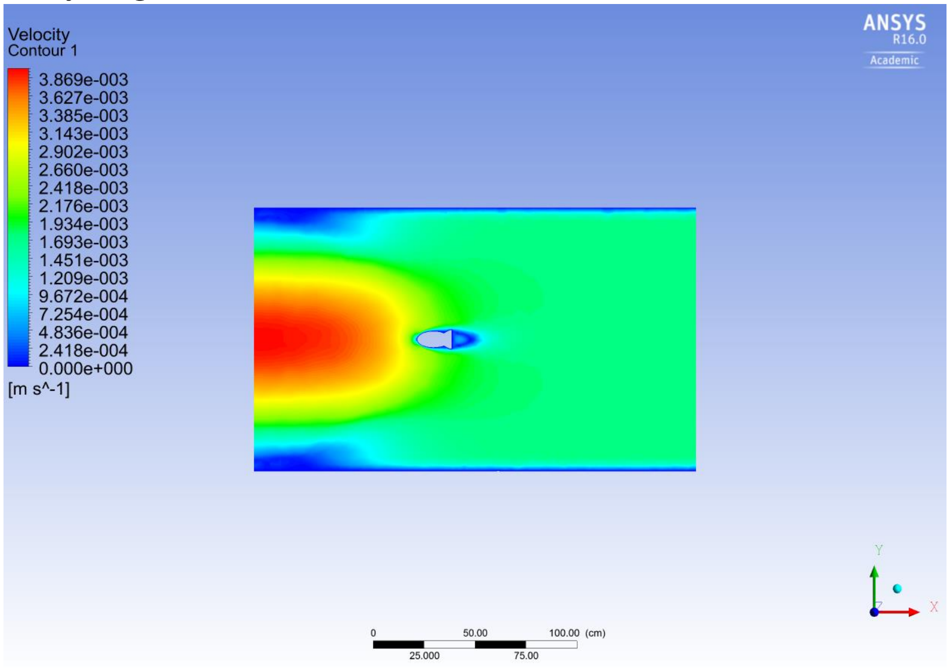
velotask4.c

(Unknown Scope)

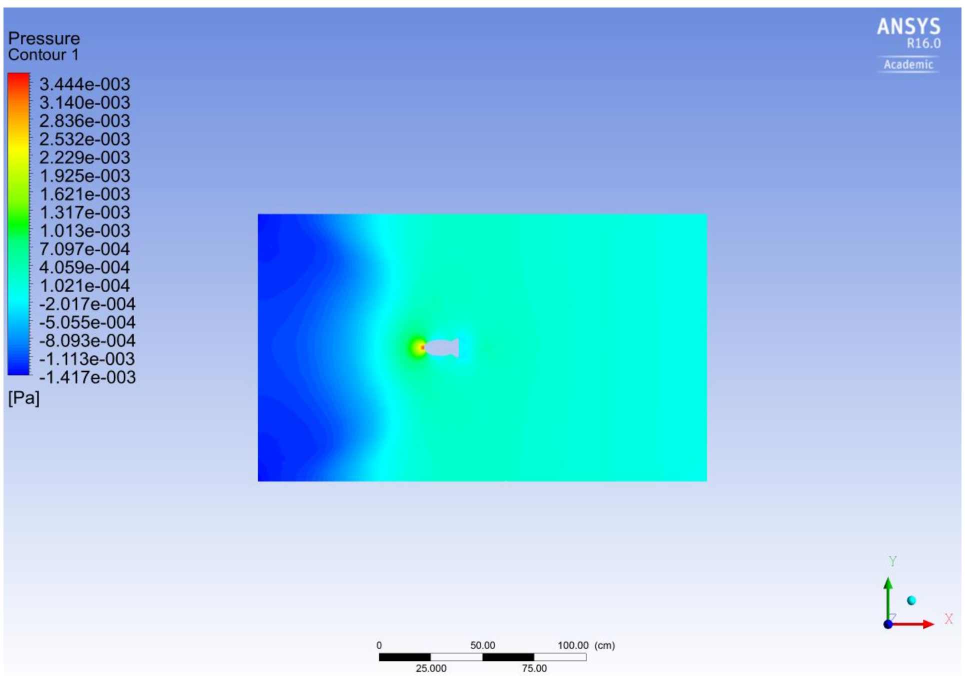
```
#include "udf.h"
DEFINE_PROFILE(inlet_x_velocity, thread, position)
{
    face_t f;
real x[ND_ND]; /* this will hold the position vector */
real y;
real z;
real r=0.6; /*defining the radius of the inlet*/
real V_max=2*0.0002; /*defining the max velocity*/
begin_f_loop(f,thread) /*initializing the loop*/
{
    F_CENTROID(x,f,thread);
    y=x[1]; /*setting y*/
    z=x[2]; /*setting z*/
    F_PROFILE(f,thread,position)=0.004*(1-((y*y+z*z)/(r*r))); /*calculating the V(r) value*/
}
end_f_loop(f,thread)
}
```

2. Contour Plots:

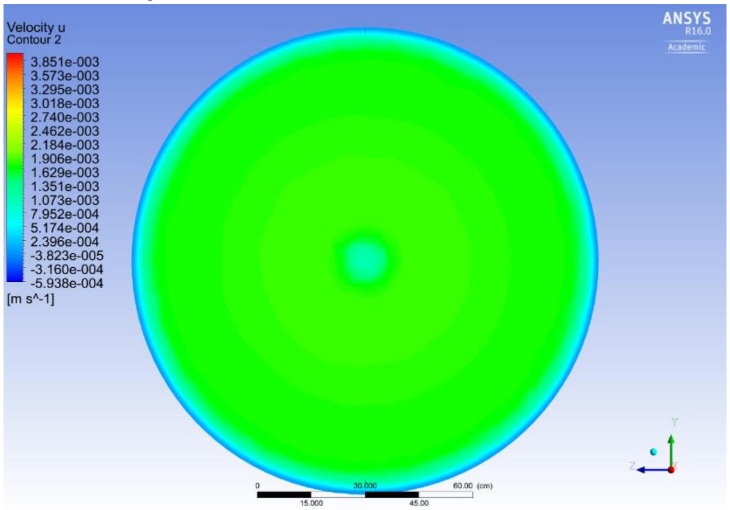
i. Velocity Magnitude:



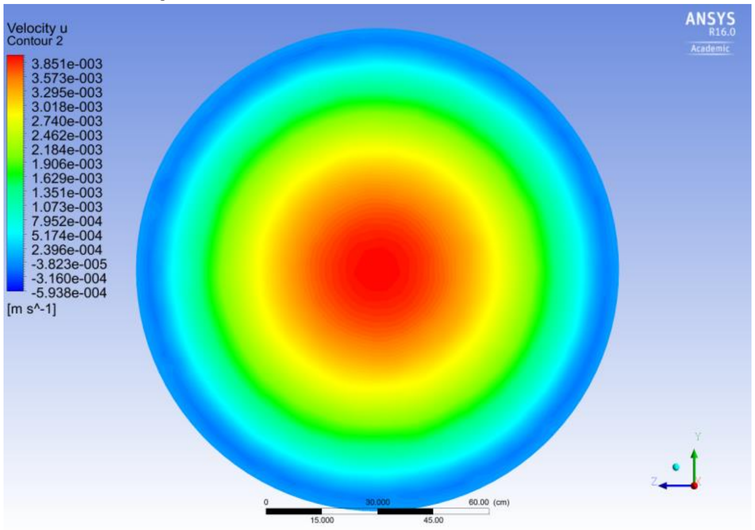
ii. Pressure:



iii. X-Component Velocity at x=25 cm.



iv. X-Component Velocity at Inlet



Discussion: When comparing the plots from task 3 to task 4 it is clear that changing the profile effects the results in a major way. The velocity decreases drastically when travelling from front to back in task 4 compared to task 3. This mainly due to the fact that the velocity is twice as high at the center in task 4. Secondly, the pressure at the tip of the half fish is a lot higher in task 4.