Technical Report

ME 412

Numerical Thermo-Fluid Mechs

Third Ansys Project

Author:
Arturo Machado Burgos

Submitted to:
Dr. Surya Pratap Vanka

March 12, 2022

0.1 Problem Description

The objective of this project is to solve the Transient Flow over a cylinder. It consists in the following problem: simulate a 2D inlet flow over the geometry.

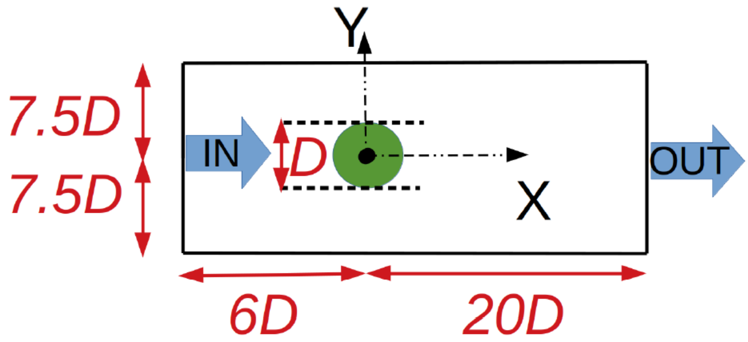

Figure 1: Transient flow over a cylinder

The geometry dimensions of Fig. 1 are given in Tab. 1

Table 1: Geometry elements and its magnitude

Geometry Element	**Magnitude**
D	1 $[m]$

It is required to solve the transient solution at two values of Reynolds number: 20 and 80, by setting appropriate values of U.

The Reynolds number in this problem can be calculated using the given formula:

$$Re = \frac{\rho U D}{\mu} \tag{1}$$

Where:

- $\rho = 1\ kg/m^3$ is the constant density of the fluid
- $\mu = 1\ Pa.s$ is the dynamic viscosity of the fluid

The only observation considering the code is that aiming to reach 20 and 80 as the Reynolds number, considering the values of the other parameters in the formula, the U variable had to be changed according to the relation: $U = 20[m/s]$ corresponds to $Re = 20$ and $U = 80[m/s]$ corresponds to $Re = 80$.

Solve for two values of Reynolds numbers: 20 and 80 for each case. Repeat the following for each of the TWO problems:

1. Plot all residuals lift and drag coefficient: 3 figures
2. Plot contours of X and Y velocities and its magnitude for at least 4 time-steps: $4 \times 3 = 12$ figures

So for each of the two Reynolds numbers, please include the above 15 plots along with a single common mesh plot. Also, include a discussion of the effects of Reynolds number on the flow, lift and drag coefficients.

0.2 Setup Procedure

0.2.1 Geometry Creation

During this project, first it was defined the geometry, according to the measurements what is showed in Fig. 2:

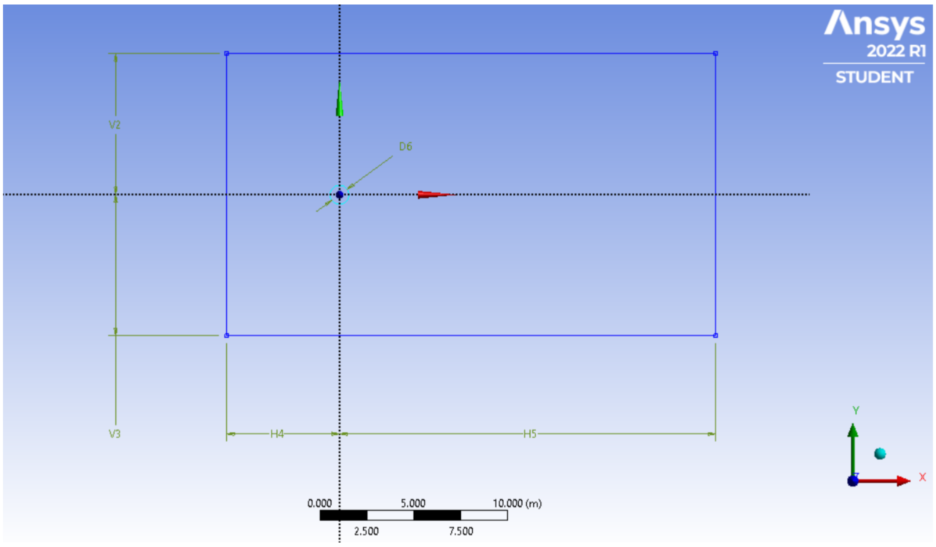

Figure 2: Geometry's constrains

Then it was set the geometry as a fluid conformation after generating the surface.

0.2.2 Mesh

As the next step, it started to create the mesh for the body to be simulated. As seen in in Fig. 3 the specifications were:

- Thickness set to 0.0m
- Methods set to triangular mesh
- The cylinder edge sizing set to 0.025 m
- Mesh refinement according to boundary scoping method, first layer thickness as $2.5 \cdot 10^{-}2$, maximum layers as 40 and growth rate as 2.5

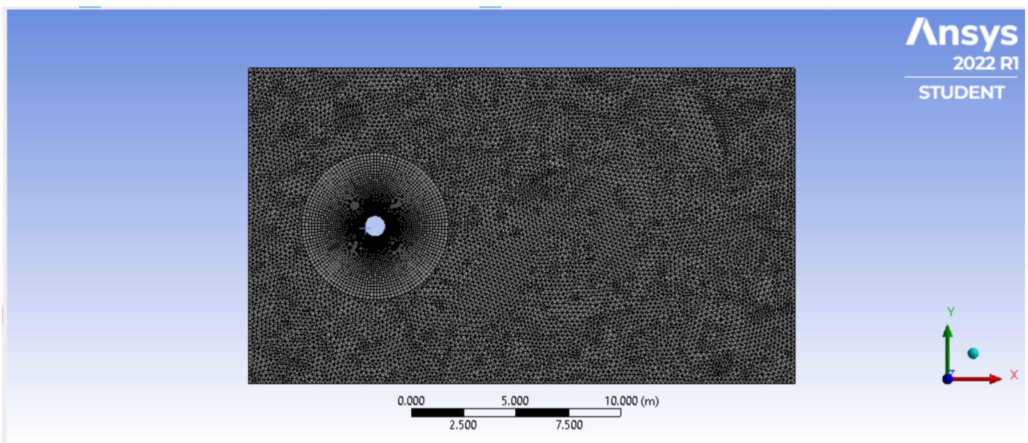

Figure 3: Mesh result according to the properties set above

Important to mention that as seen in Fig. 4 four named selections were set for the boundary conditions.

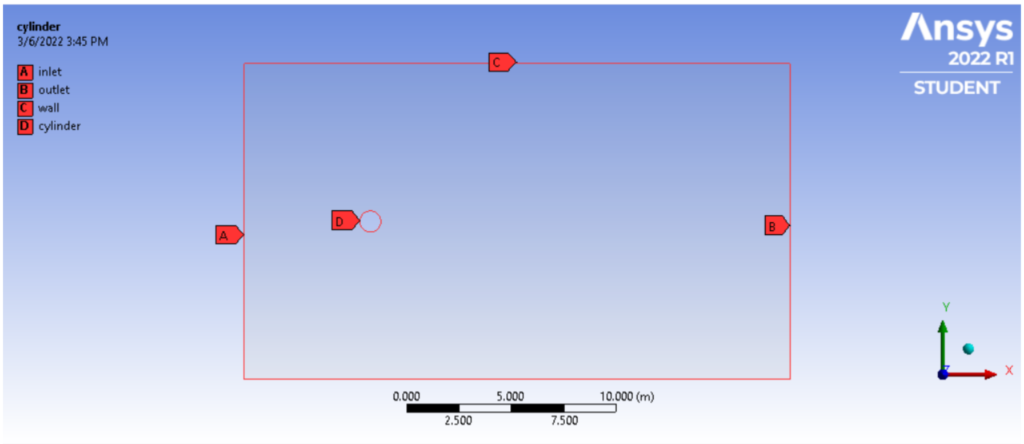

Figure 4: Named selections

0.2.3 Set Fluent Physical Conditions

At this step, it had began the inputs of the problem's physical conditions to be simulated. First, neglect gravity and set time as "transient" as seen in Fig. 5. Apart from that, on models only the Viscous option was set as laminar, the other were off. This can be seen in Fig. 6

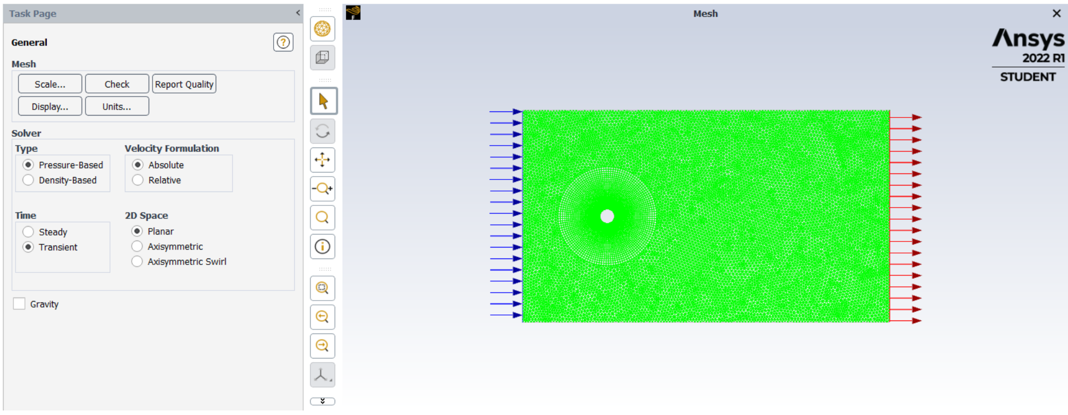

Figure 5: Initial Fluent configuration

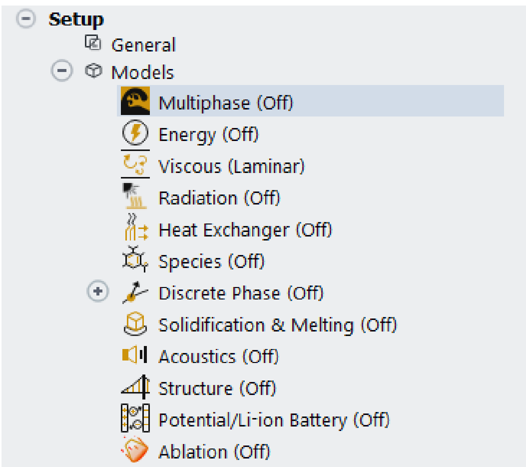

Figure 6: Models setup

As for the fluid selected, it was chosen air with the following properties as show in Fig. 7

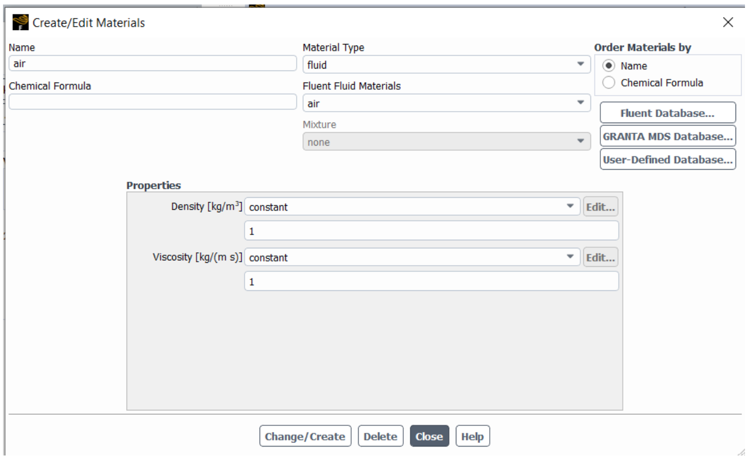

Figure 7: Air properties

For the Momentum Boundary conditions the walls apart form the inlet and outlet and the cylinder were set as stationary ones considering the motion, as well as, no slip regarding shear condition. For the inlet the velocity profile was taken using 20 or 80 m/s. for the outlet wall the condition was set as pressure-outlet.

0.2.4 Set Solution Initialization and Methods

Set the reference values to inlet. Also set the method solution to second order implicit transient formulation. To perform the simulations it was used hybrid initialization.

Important to mention that the the auto-save was configured for each time-step. The maximum number of time iteration was defined as 350, where the $dt = 0.01$ and for each t iteration as for the convergence 100 iterations were set.

0.3 Results and Discussion

Two simulations were performed. The first one with Re = 20. The second with Re = 20.

0.3.1 Residuals

Figures 8 to 9 show the residuals for continuity, and momentum (represented by x-velocity and y-velocity) over the number of iterations required to converge the solution. It is possible to verify oscillations on the convergence of both simulations, this is due the iterations that are done at each progression on time.

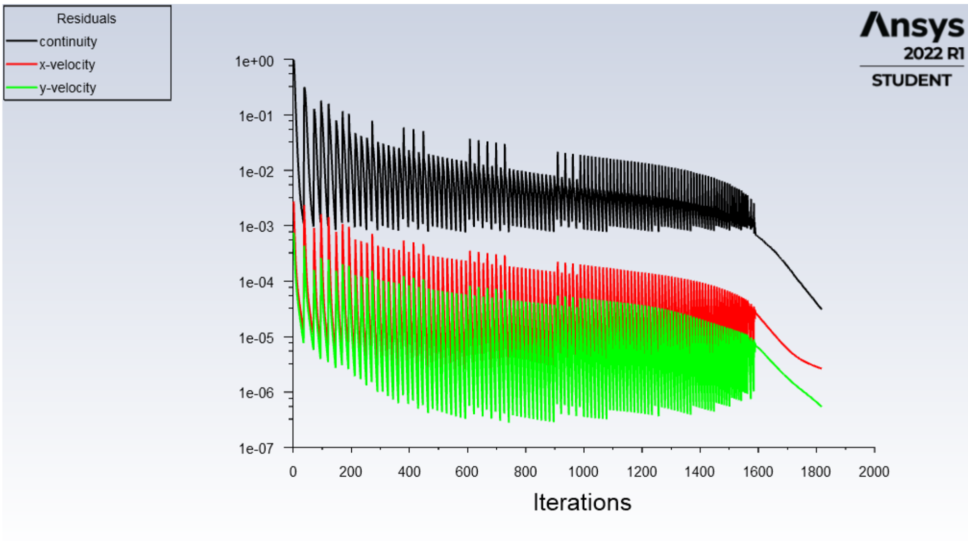

Figure 8: Residuals for Re = 20

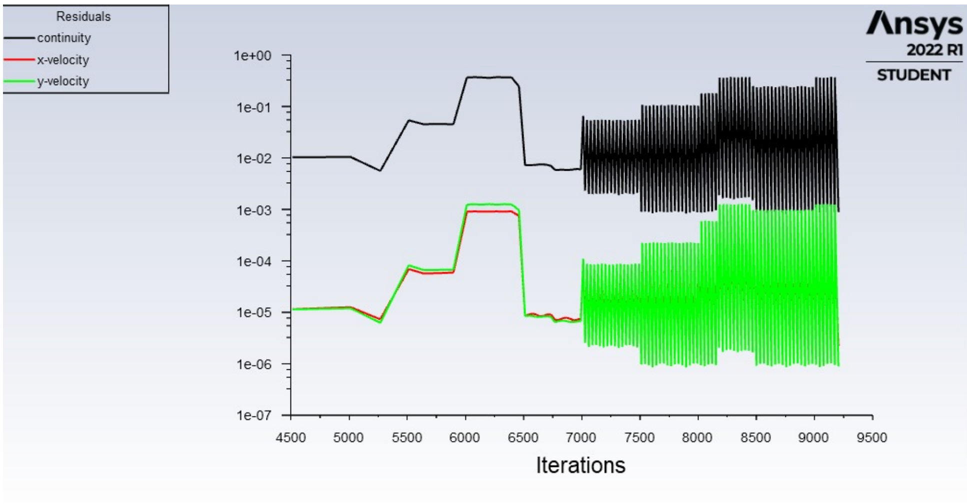

Figure 9: Residuals for Re = 80

Table 2: Number of iterations needed to converge to the solution, applied on each simulation case varying the Reynolds number - those were acquired looking at the Console Log

Simulation	**Number of Total Iterations**
Re = 20	1818
Re = 80	9157

0.3.2 Velocity Contour

Figures 10 to 33 show the velocity contour of all previous simulations. The plots seem correct accordingly to the problem physics. For each simulation four discreet moments in time were captured.

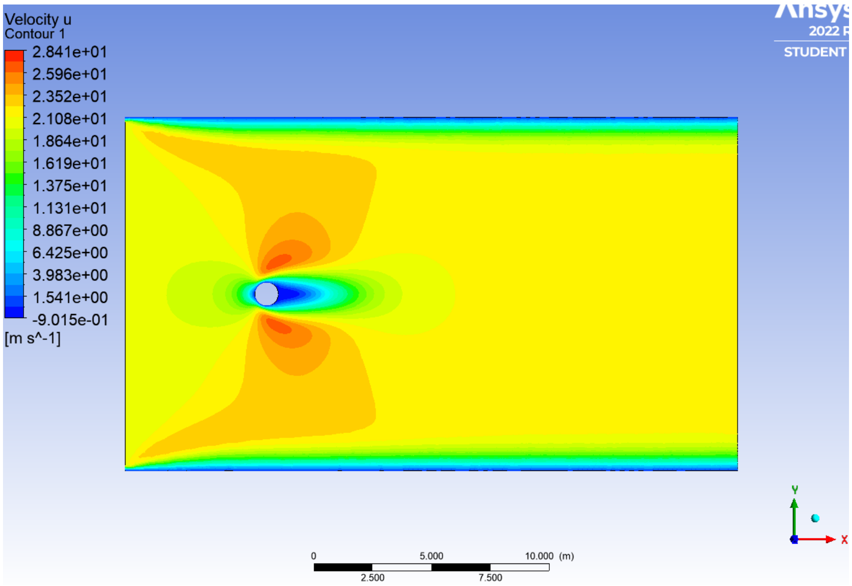

Figure 11: Velocity X contour for Re = 20 at 22nd time-step

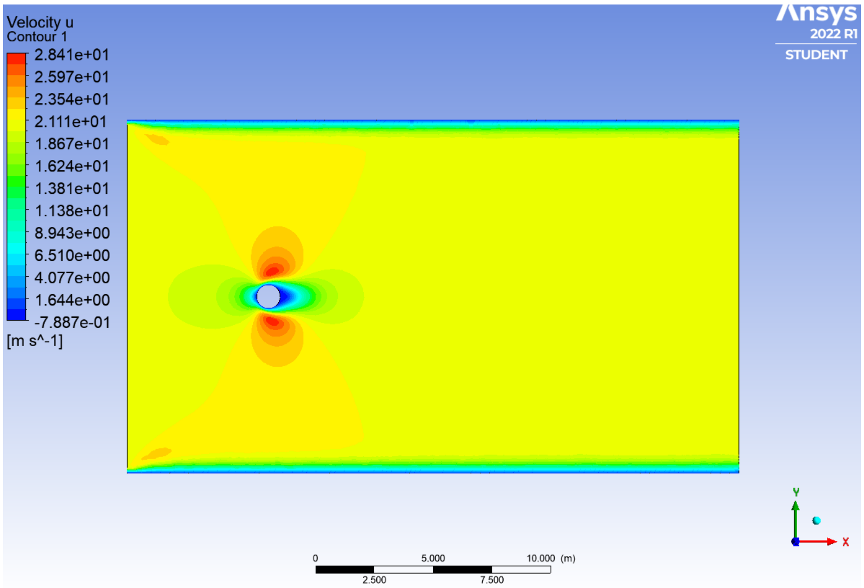

Figure 10: Velocity X contour for Re = 20 at 7th time-step

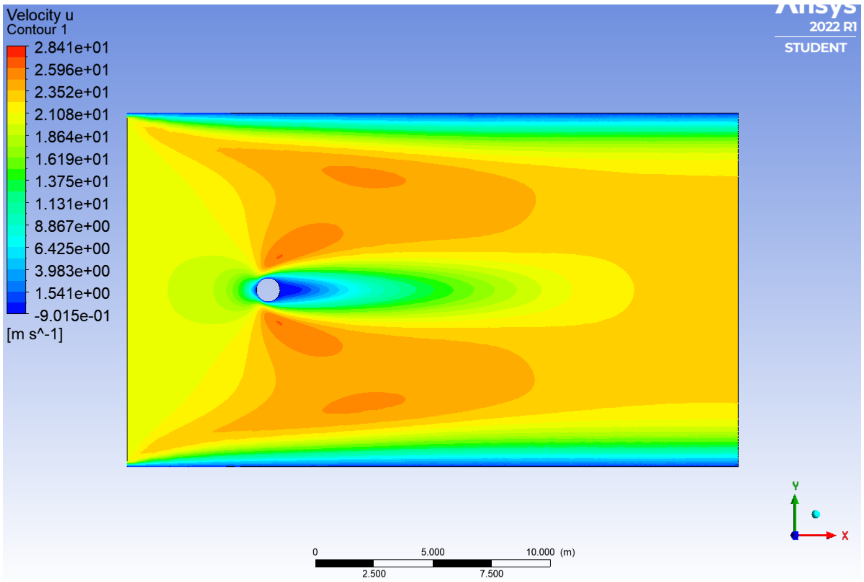

Figure 12: Velocity X contour for Re = 20 at 63rd time-step

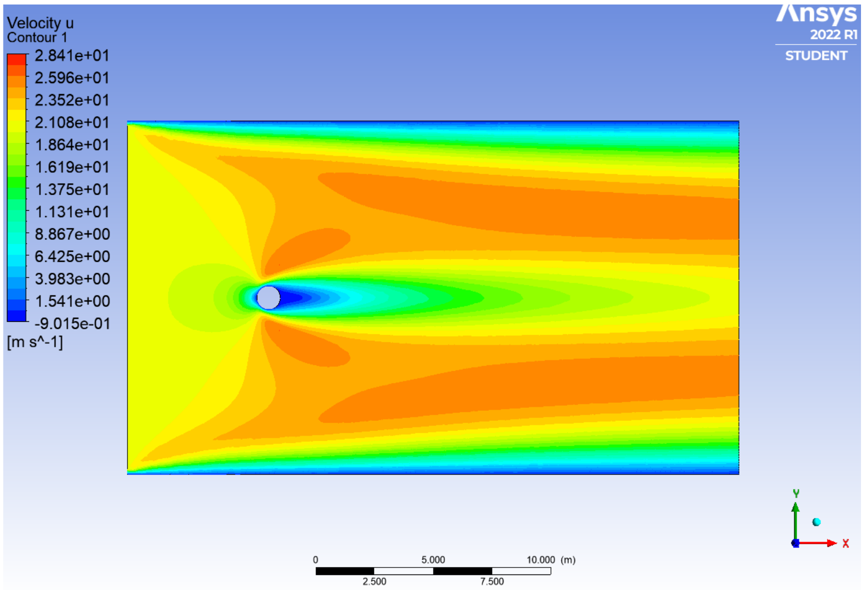

Figure 13: Velocity X contour for Re = 20 at 193rd time-step

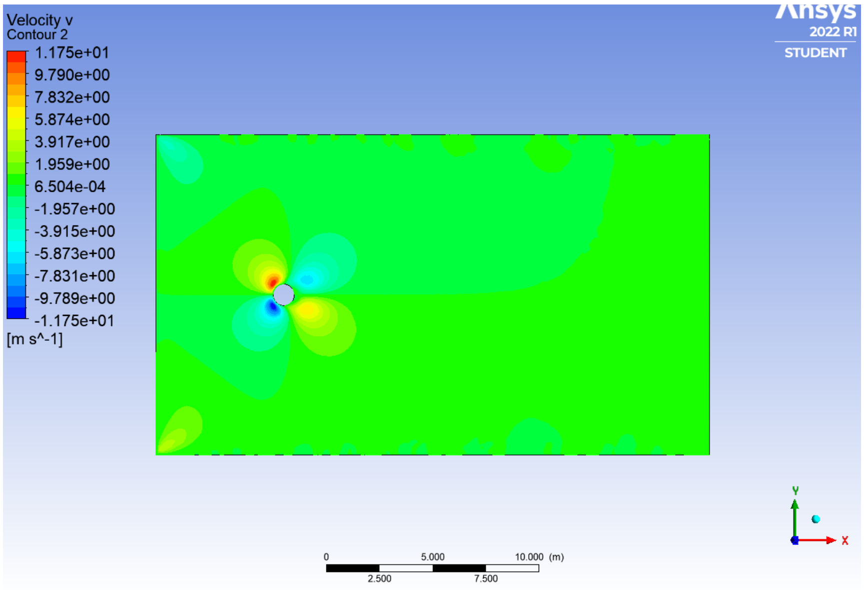

Figure 14: Velocity Y contour for Re = 20 at 7th time-step

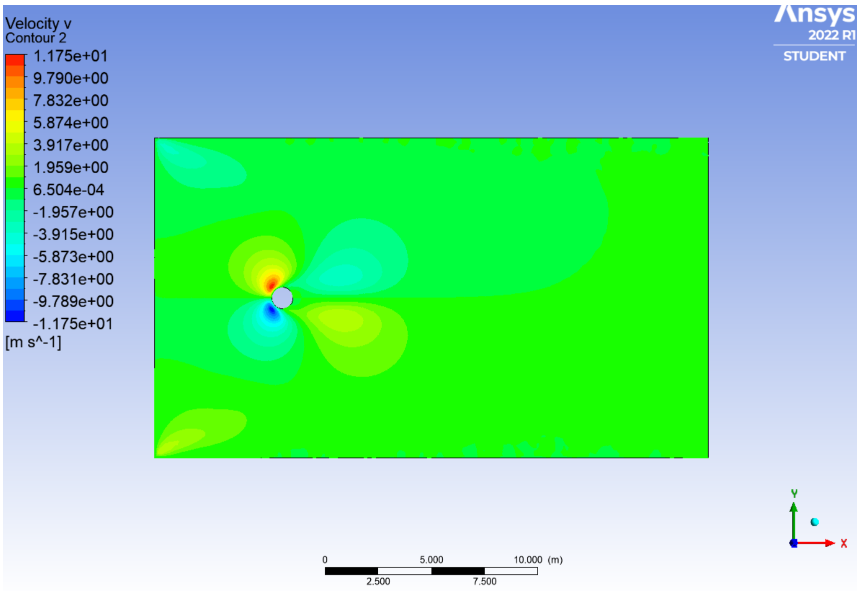

Figure 15: Velocity Y contour for Re = 20 at 22nd time-step

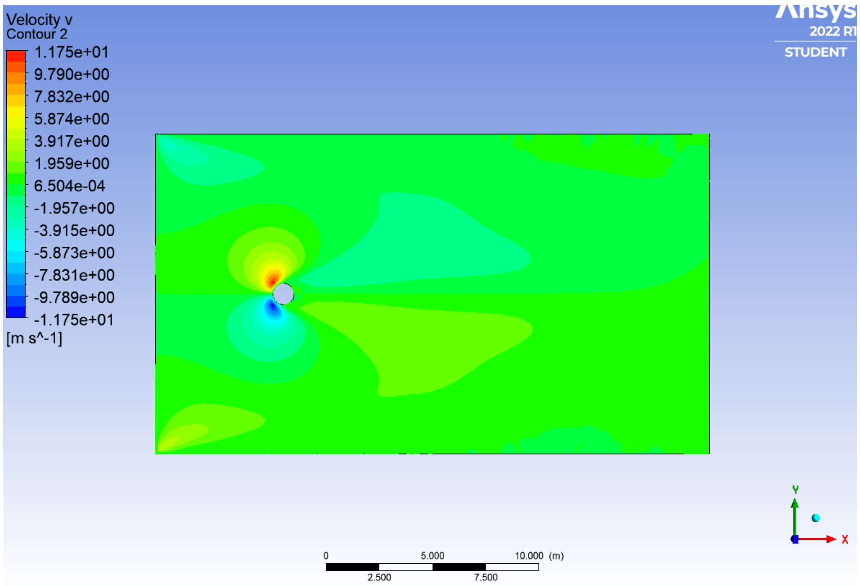

Figure 16: Velocity Y contour for Re = 20 at 63rd time-step

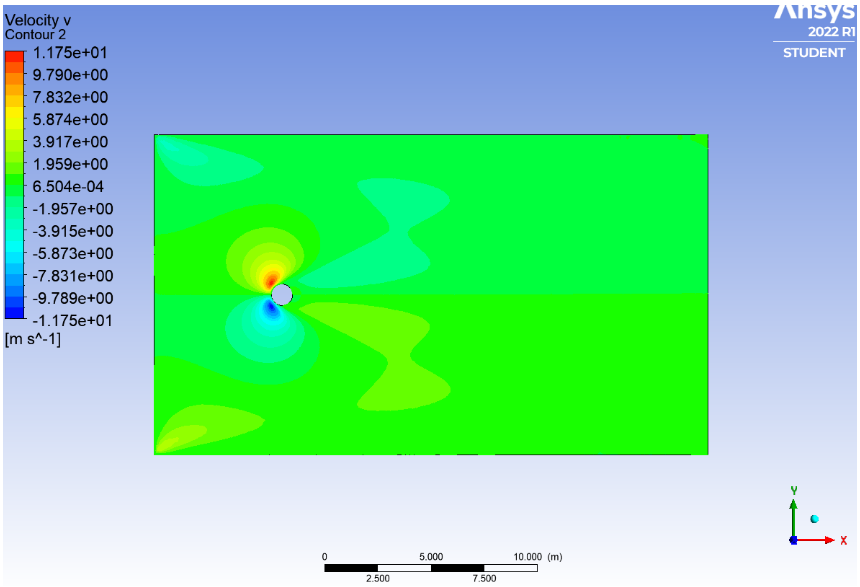

Figure 17: Velocity Y contour for Re = 20 at 193rd time-step

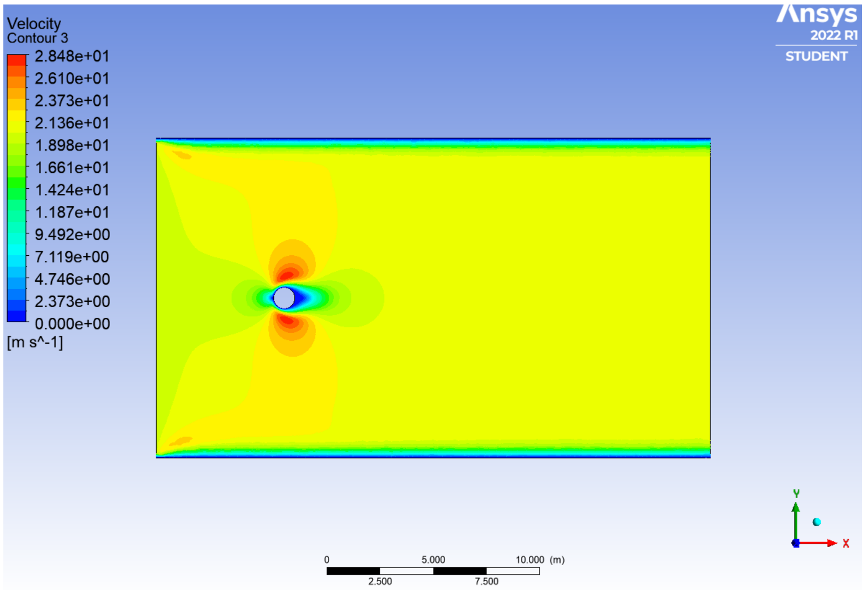

Figure 18: Velocity magnitude contour for Re = 20 at 7th time-step

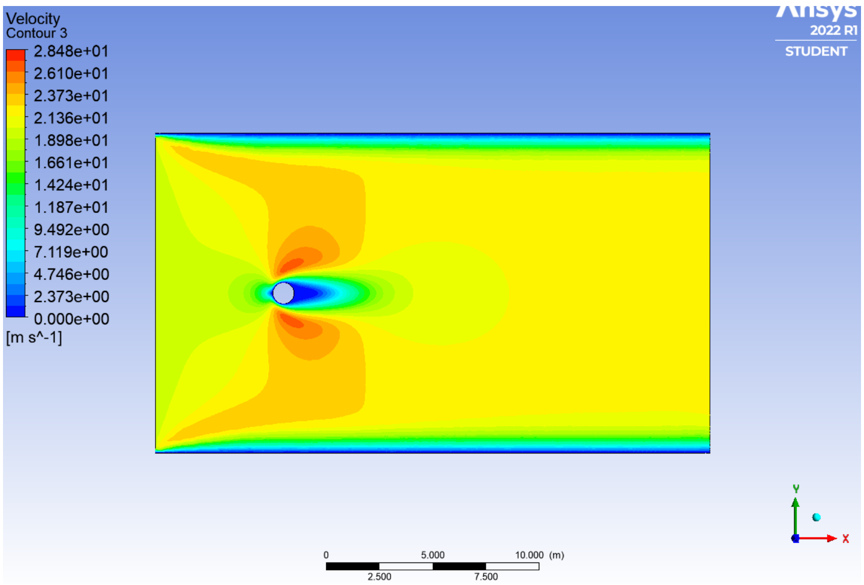

Figure 19: Velocity magnitude contour for Re = 20 at 22nd time-step

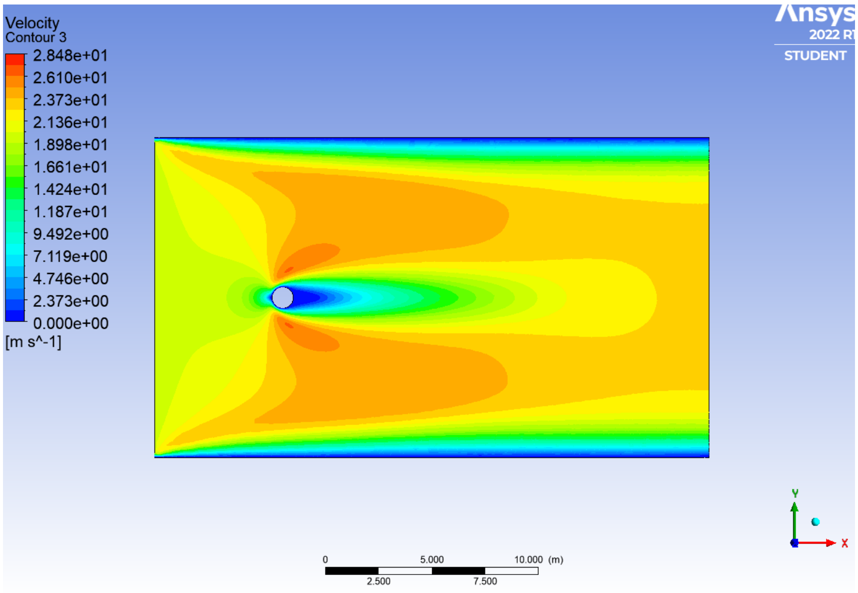

Figure 20: Velocity magnitude contour for Re = 20 at 63rd time-step

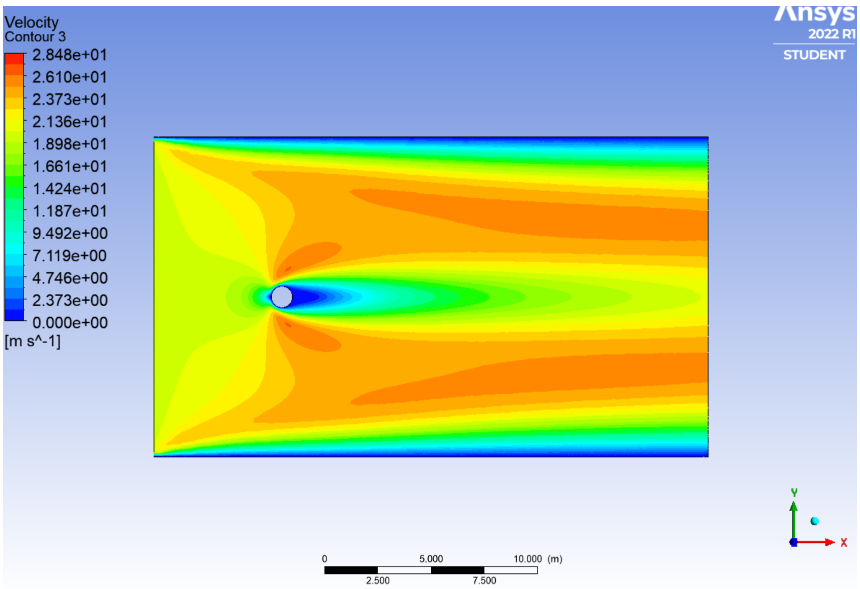

Figure 21: Velocity magnitude contour for Re = 20 at 193rd time-step

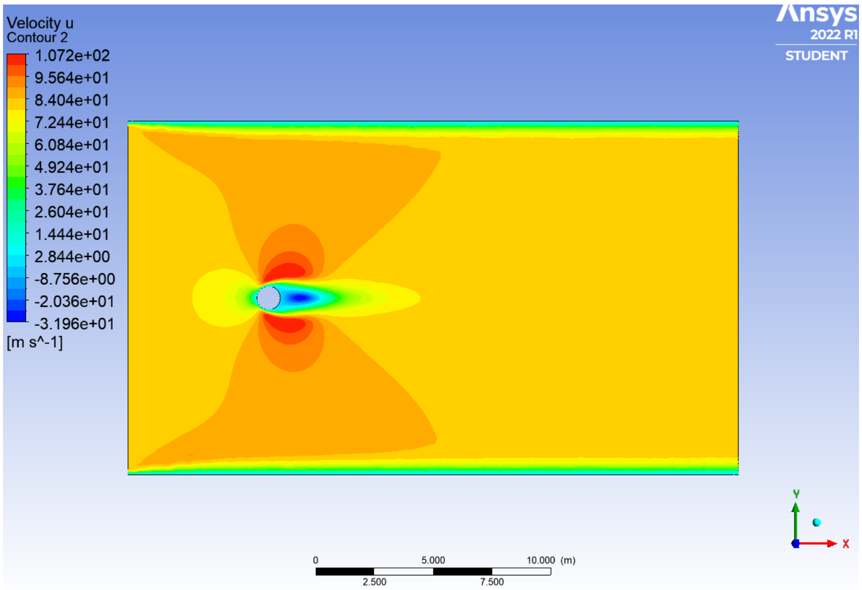

Figure 22: Velocity X contour for Re = 80 at 7th time-step

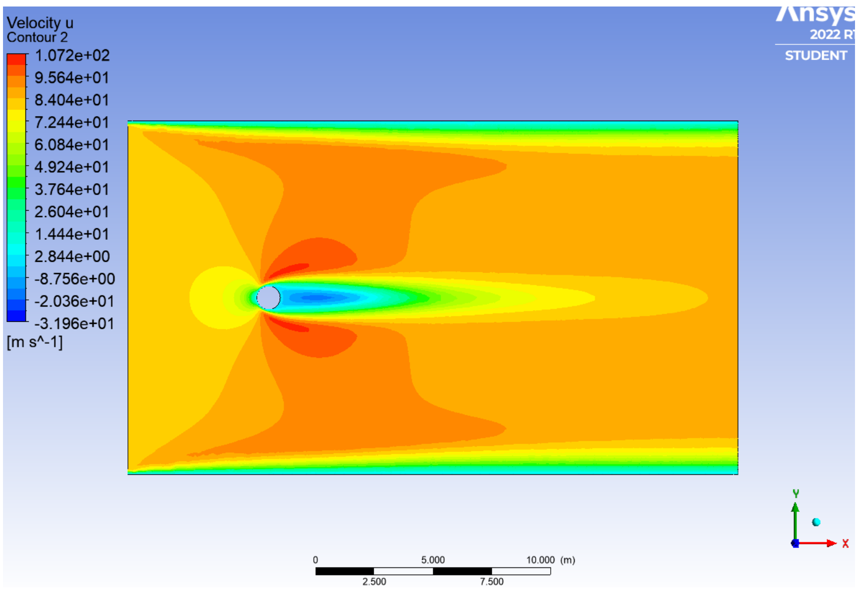

Figure 23: Velocity X contour for Re = 80 at 22nd time-step

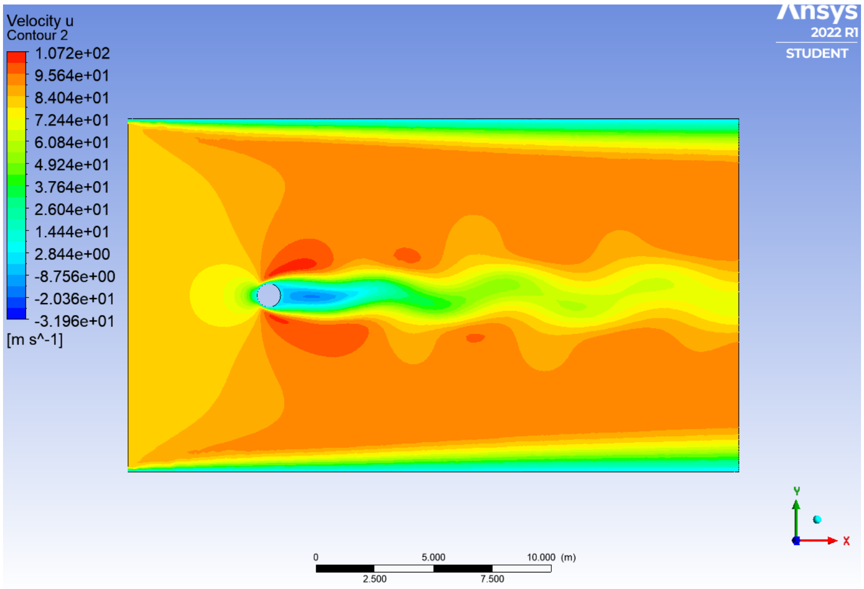

Figure 24: Velocity X contour for Re = 80 at 193rd time-step

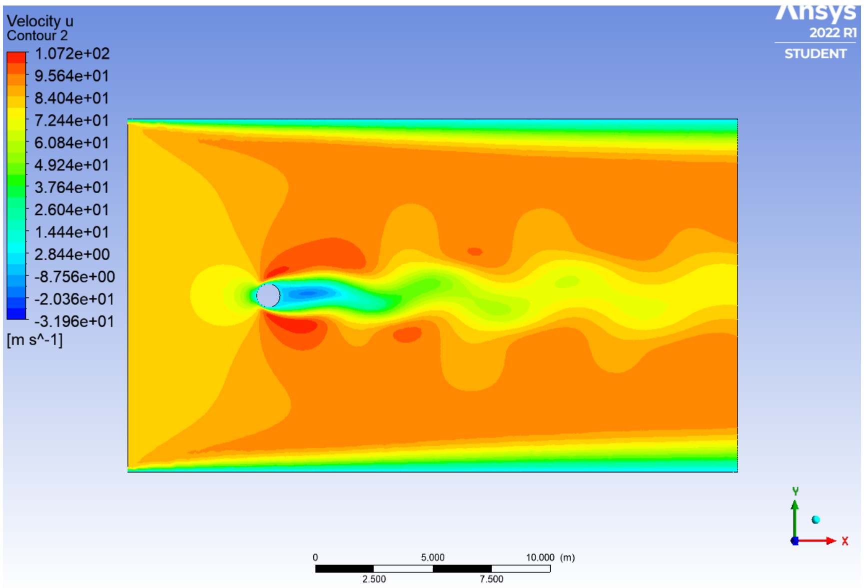

Figure 25: Velocity X contour for Re = 80 at 350th time-step

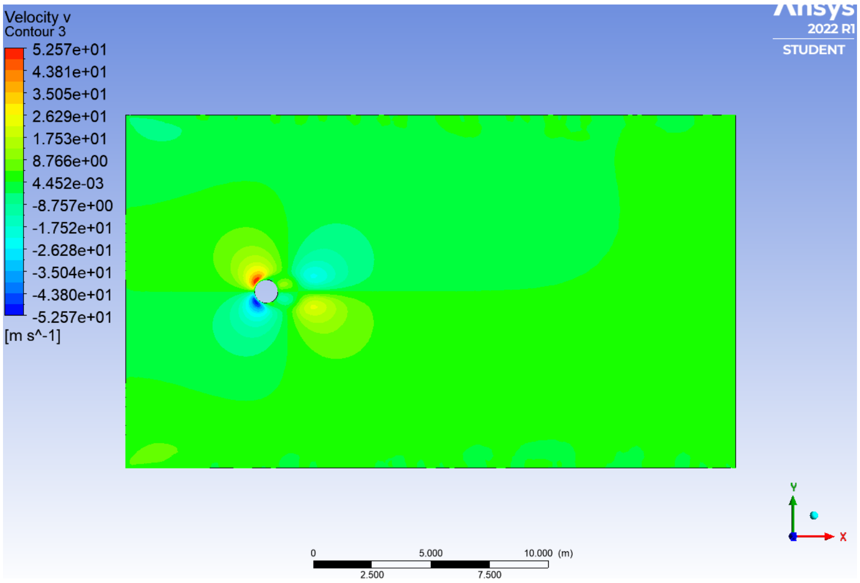

Figure 26: Velocity Y contour for Re = 80 at 7th time-step

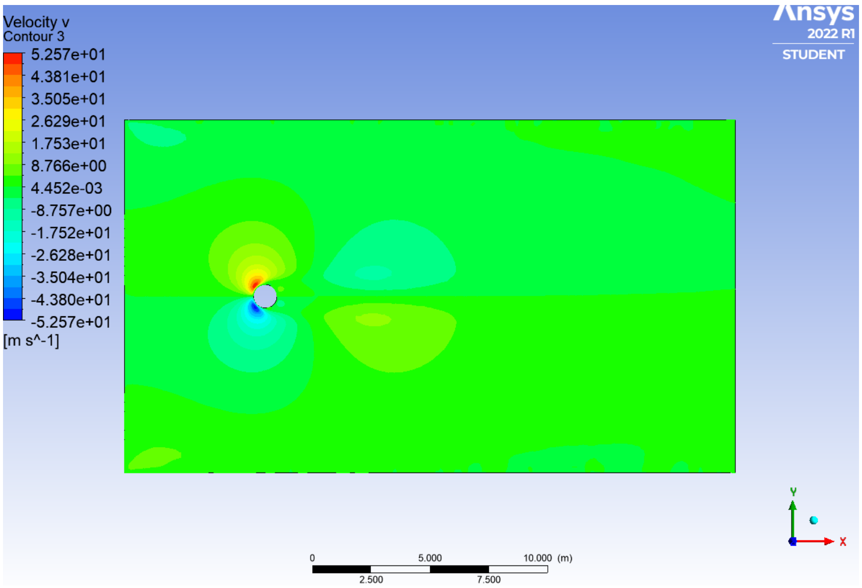

Figure 27: Velocity Y contour for Re = 80 at 22nd time-step

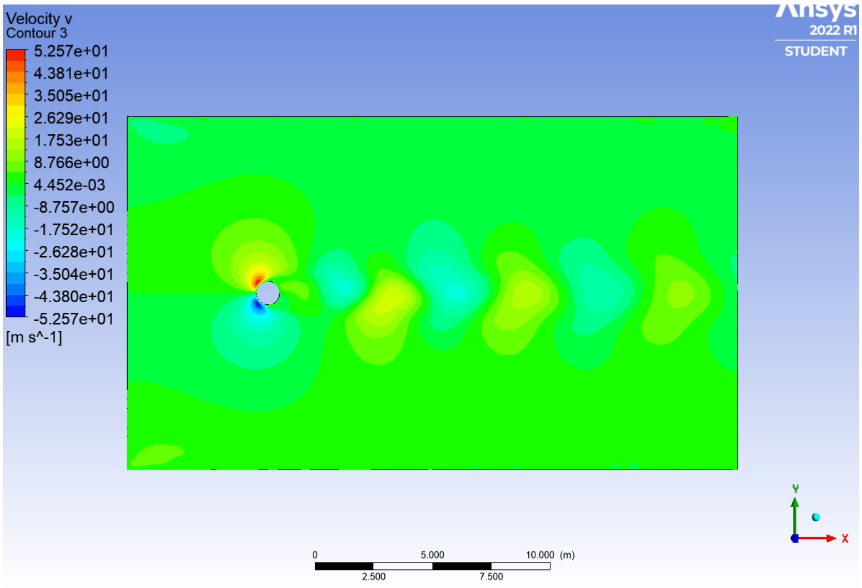

Figure 28: Velocity Y contour for Re = 80 at 193rd time-step

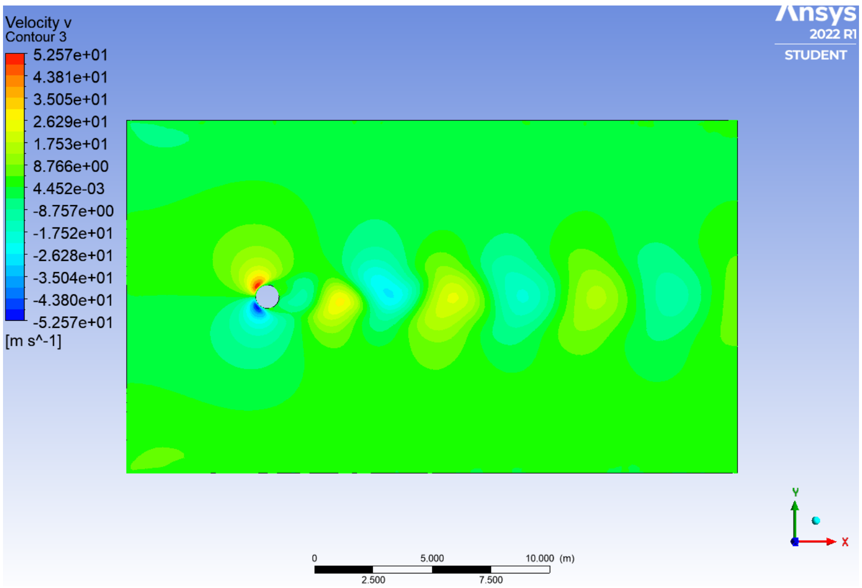

Figure 29: Velocity Y contour for Re = 80 at 350th time-step

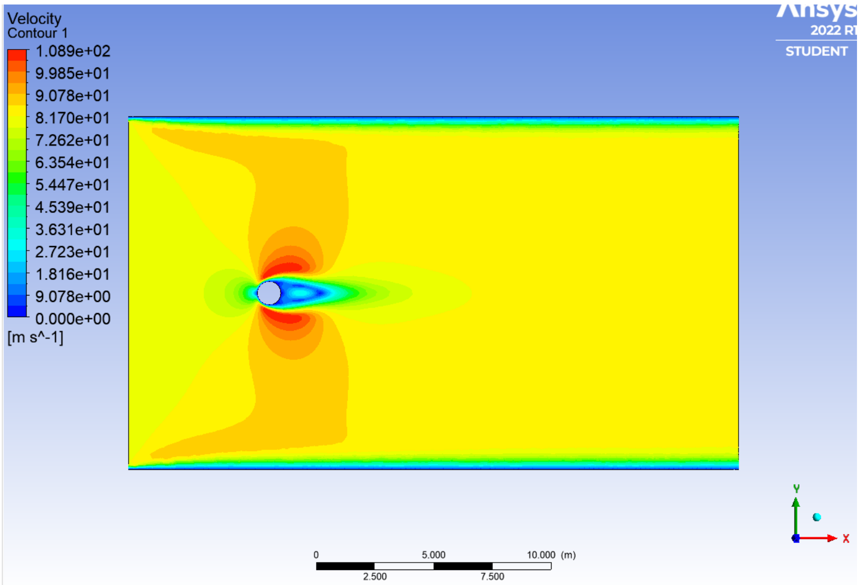

Figure 30: Velocity magnitude contour for Re = 80 at 7th time-step

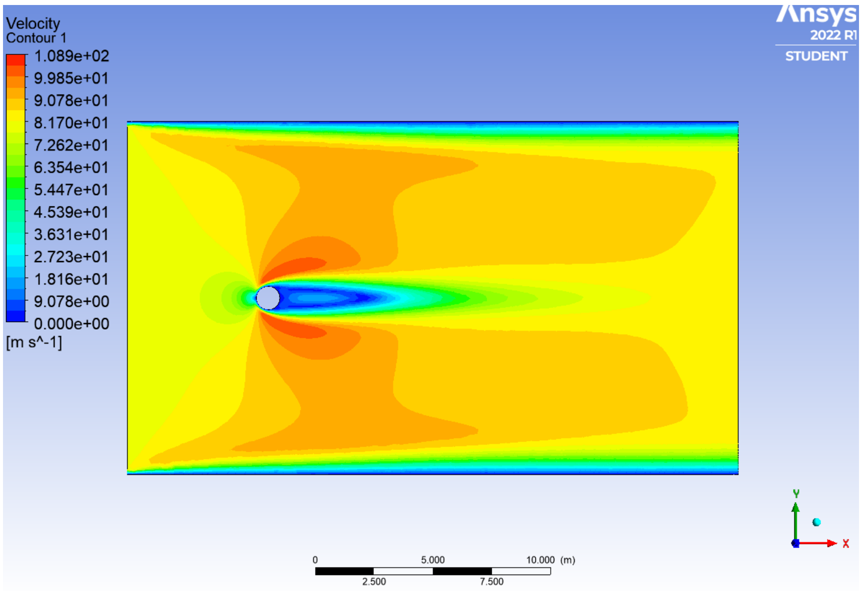

Figure 31: Velocity magnitude contour for Re = 80 at 22nd time-step

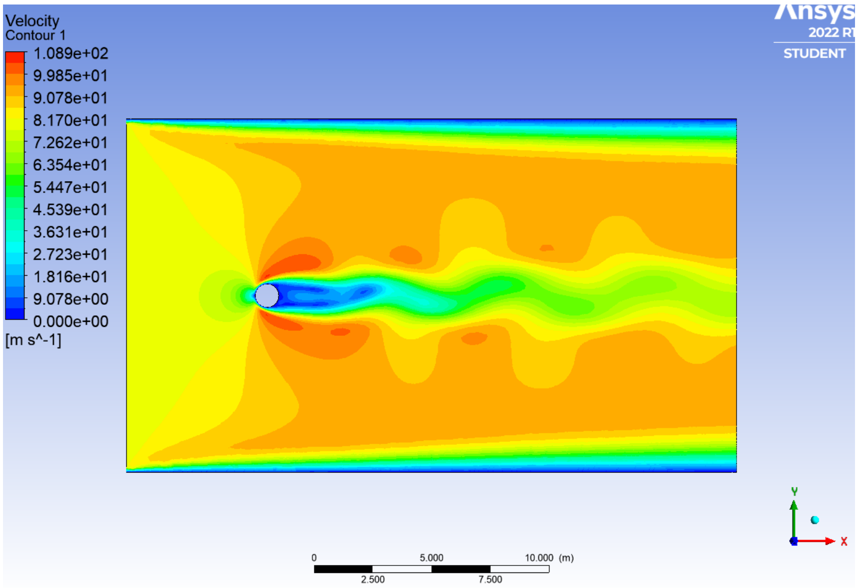

Figure 32: Velocity magnitude contour for Re = 80 at 193rd time-step

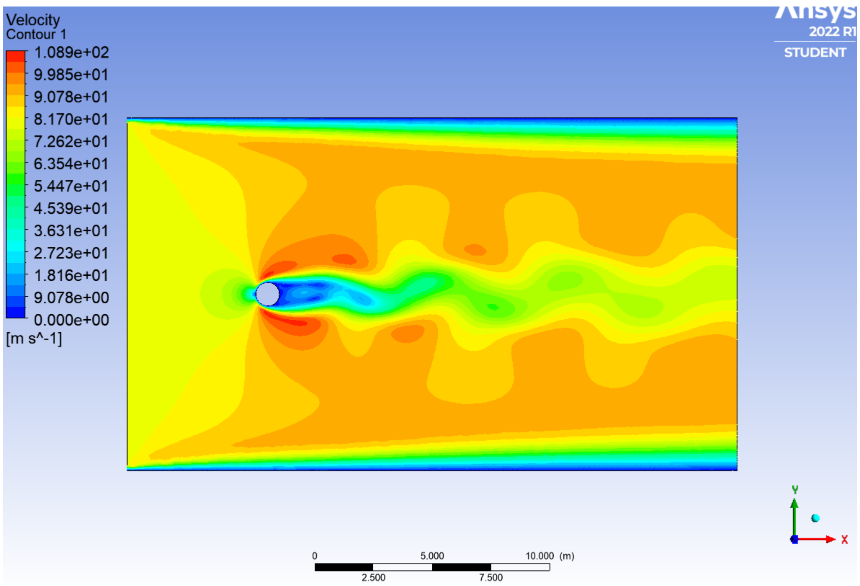

Figure 33: Velocity magnitude contour for Re = 80 at 350th time-step

0.3.3 Drag and Lift Coefficients

Figures 34 to 37 show the lift and drag coefficients through time for both simulations.

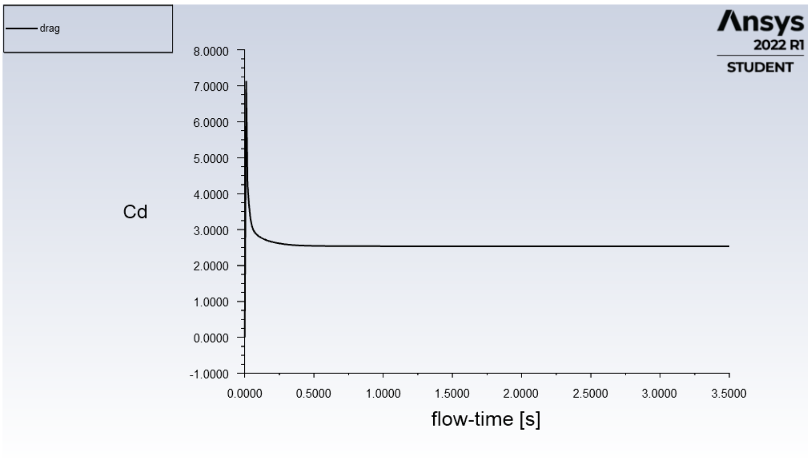

Figure 34: Drag coefficient for Re = 20

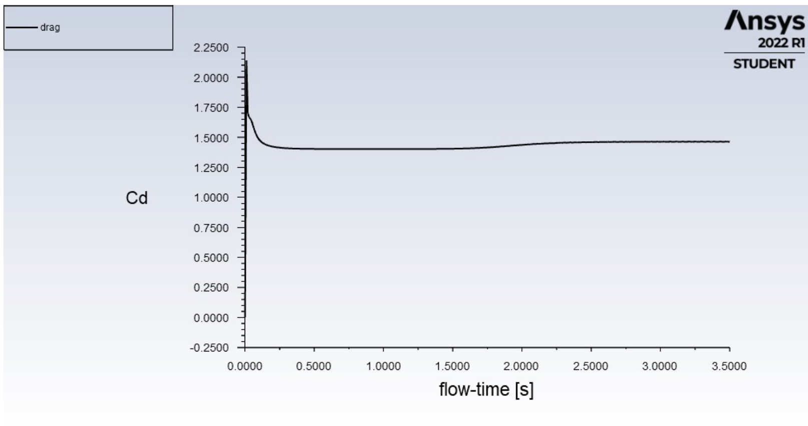

Figure 35: Drag coefficient for Re = 80

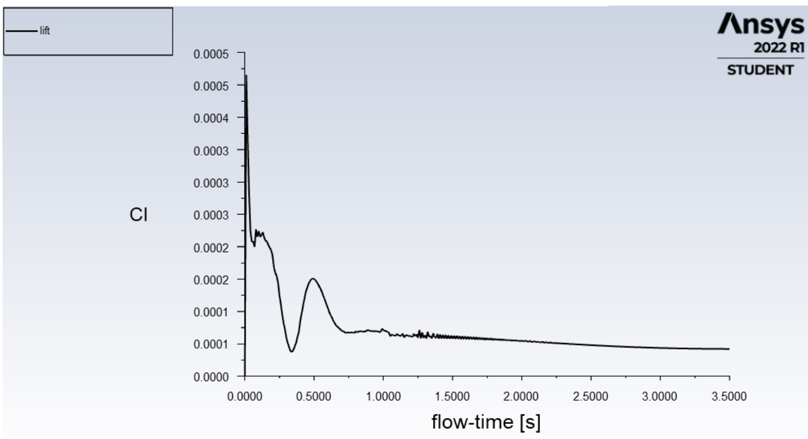

Figure 36: Lift coefficient for Re = 20

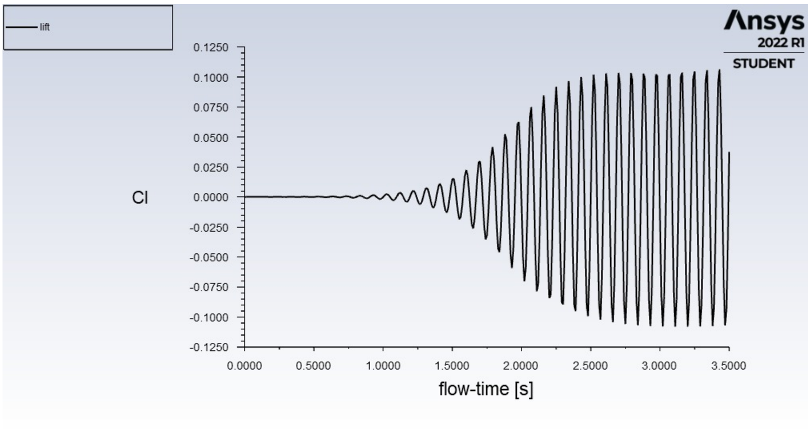

Figure 37: Lift coefficient for Re = 80

0.3.4 Discussion

The plots seem correct accordingly to the problem physics. Looking at the results it is possible to infer that:

- With respect to flow itself with the increase of the Reynolds number it can be seen that there is the emergence of vortexes begins once the pressure gradient allows this phenomena (Re = 80). It is expected that, increasing even more the Reynolds number, more instabilities would have arise making the flow regime towards turbulent conformation.
- With regard drag it is possible to see that the results are in accordance with the physics once the Re = 20 simulation show a bigger drag than Re = 80. Being more accurate the drag from Re = 20 is twice bigger than Re = 80. It is possible to verify the accordance with the Cd formula where $C_d = \frac{d}{1/2.\rho.U_{inf}^2.D}$
- In relation to lift it is possible to note a different behaviour between the two Reynolds numbers. For 20 to not be considered a high Reynolds the shape of lift plot seems correct with the physics of the problem. Aside that in this simulatiopns the contour did not show any vortex emergence. For 80 as the Reynolds number as we have the vortex we notice a plot that shows the lift coefficient oscillates between positive and negative values. Also the norm of the coefficient is becoming bigger trough time.