Technical Report

ME 412

Numerical Thermo-Fluid Mechs

Fourth Ansys Project

Author:
Arturo Machado Burgos

Submitted to:
Dr. Surya Pratap Vanka

April 2, 2022

1 Problem Description

The objective of this project is to solve the Turbulent Backward Facing Step Problem. It consists in the following problem: simulate a 2D inlet flow over the geometry.

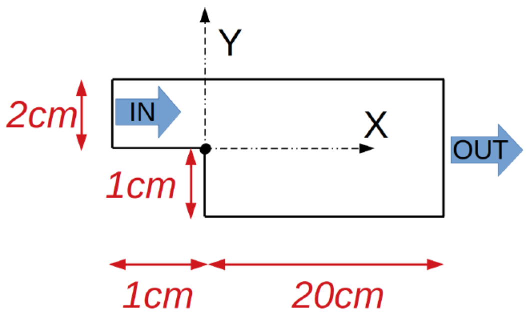

Figure 1: Turbulent Backward facing step

It is required to solve the problem at $Re = 10^4$.
The Reynolds number in this problem can be calculated using the given formula:

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

Where:

- $\rho = 1.225\ kg/m^3$ is the constant density of the fluid
- $\mu = 1.7849 \times 10^{-5}\ Pa.s$ is the dynamic viscosity of the fluid
- $H = 0.01\ m$ is the step

U is the velocity that is set using the above formula. In this case: $U = 14.606\ m/s$
Solve for two values of Reynolds numbers: 50 and 100 for each case. Repeat the following for each of the TWO problems:

1. Mesh
2. 5 residuals as a function of iterations
3. Pressure Coefficient as a function of iterations
4. Skin Friction as a function of iterations
5. Streamlines
6. X velocity contours
7. Y velocity contours
8. Velocity magnitude contours
9. Turbulent viscosity contours
10. Turbulent kinetic energy contours
11. Pressure coefficient as a function of X on lower wall
12. Skin friction coefficient as a function of X on lower wall
13. X velocity plots at X = H, X = H/2
14. Y velocity plots at Y = H, Y = H/2, Y = 0, Y = -H/2

Note down some key observations.

2 Setup Procedure

2.1 Geometry Creation

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

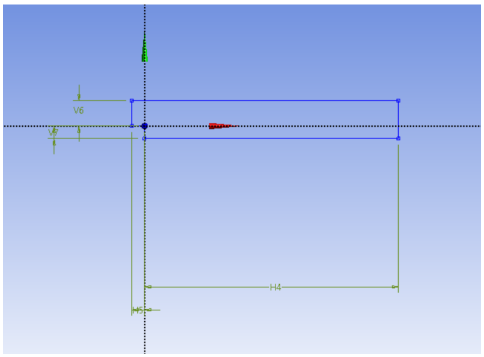

Figure 2: Geometry's constrains

Then the divisions were set accordingly as in Fig. 3

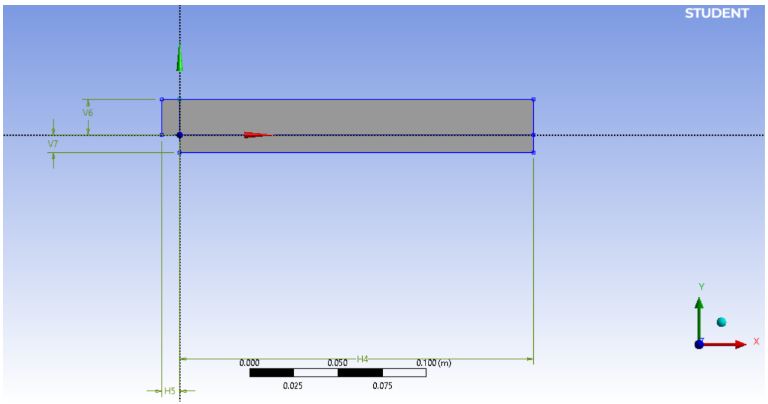

Figure 3: Divisions

2.2 Mesh

As the next step, it started to create the mesh for the body to be simulated. As seen in Figs. 4 to 6, using Face Mesh conformation.

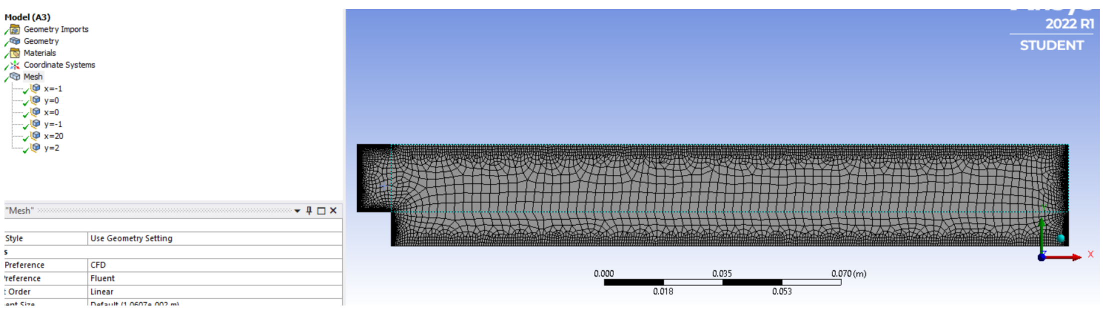

Figure 4: Geometry's mesh before face mesh, only assigning the mesh at the edges

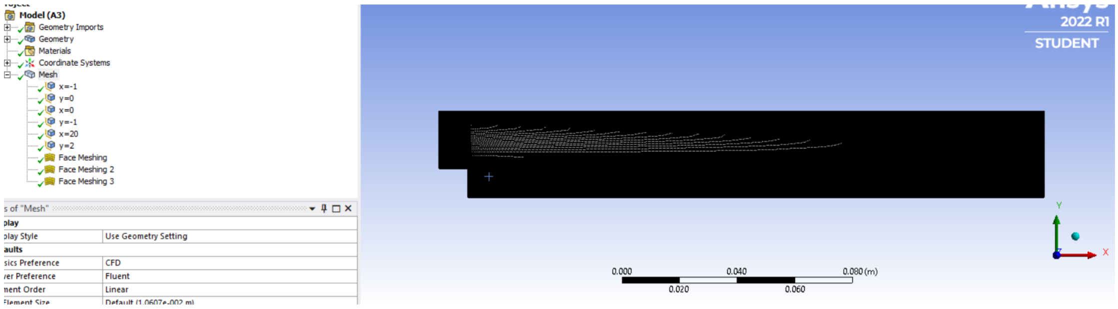

Figure 5: Geometry's mesh after face mesh

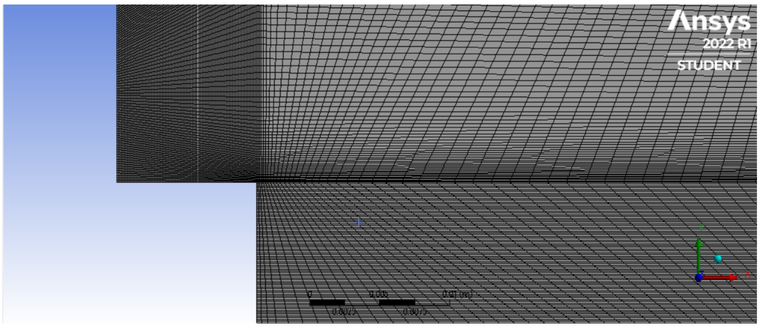

Figure 6: Zoom at the final mesh

Then it was defined the named selections:

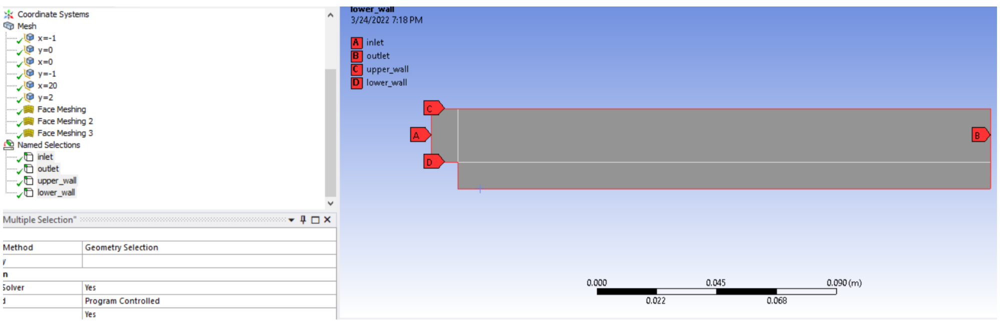

Figure 7: Named selections

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 as seen in Fig. 8. Apart from that, on models only the Viscous option was set as k-epsilon, the other were off. This can be seen in Fig. 9

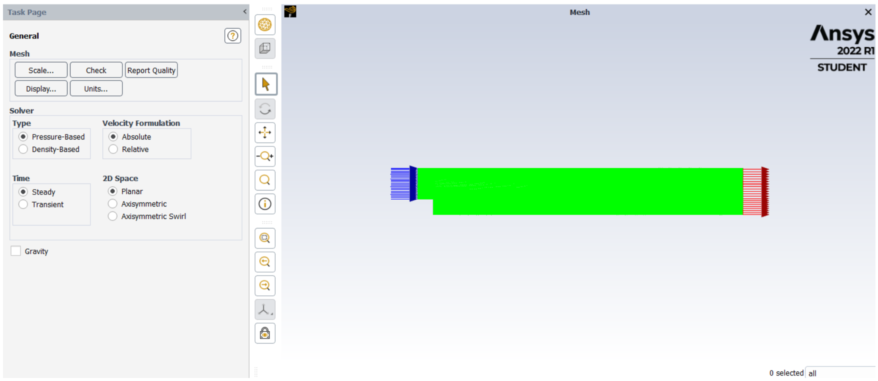

Figure 8: Initial Fluent configuration

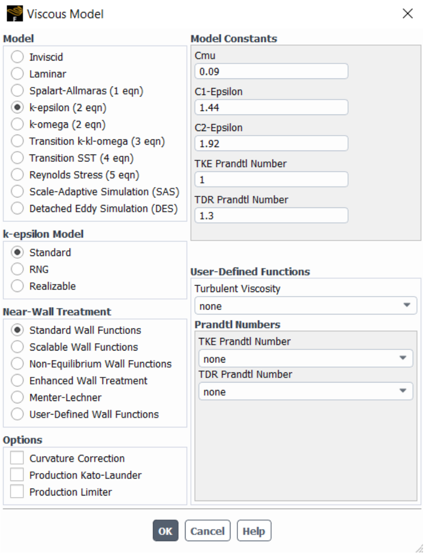

Figure 9: Models setup

As for the fluid selected, it was chosen air with the following properties as show in Fig. 10

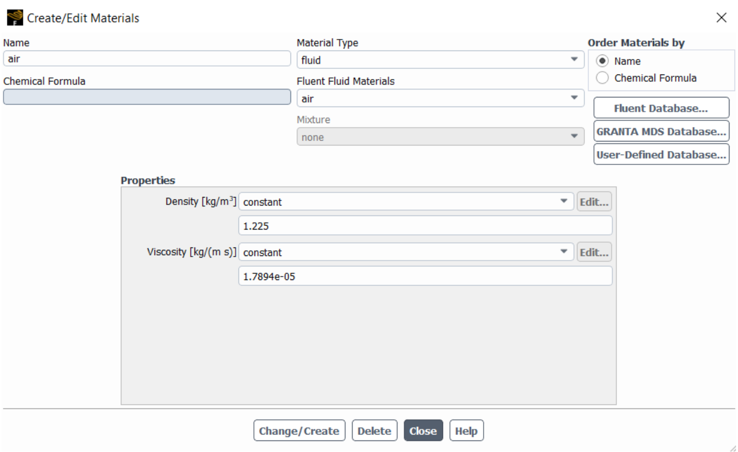

Figure 10: Air properties

Figure 11 show the Boundary Conditions used:

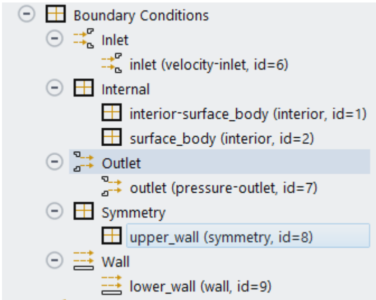

Figure 11: Boundary Conditions

The inlet velocity was set accordingly to what was previous calculated, as can be seen in Fig. 12:

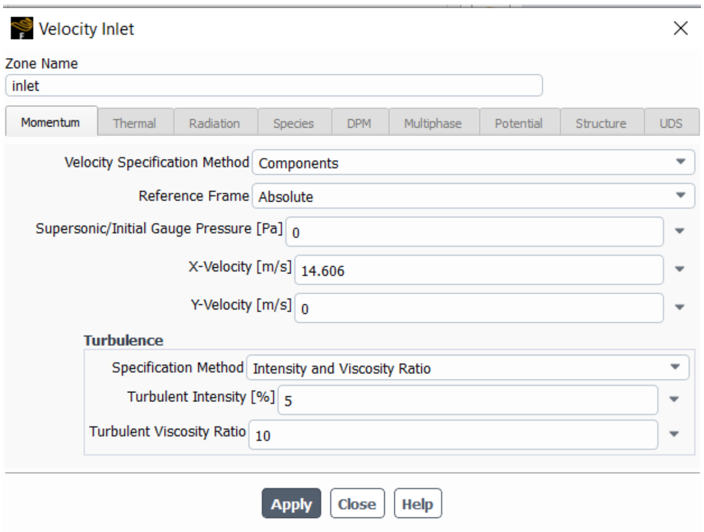

Figure 12: Boundary Conditions

2.4 Set Solution Initialization and Methods

Figures 13 and 14 show the reference values and the methods used during the procedure. The solution initialization was hybrid.

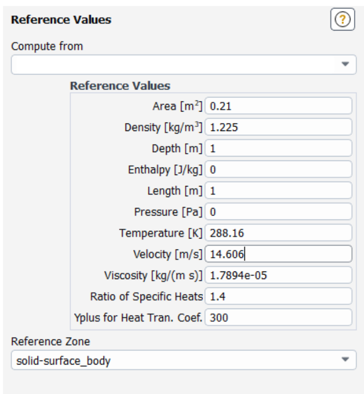

Figure 13: Reference values

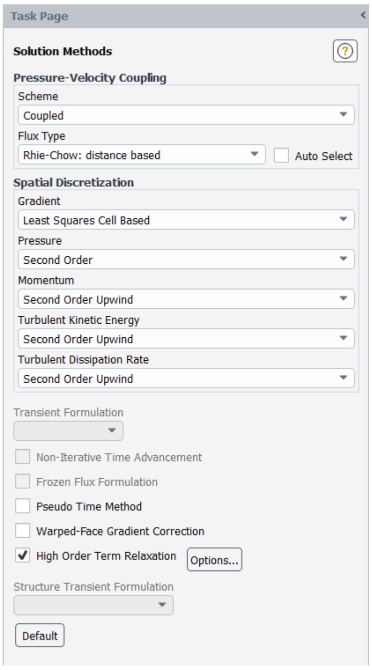

Figure 14: Solution methods

Regarding the "Run Calculations" it was set the maximum number of iterations as 1000, what can be seen in Fig. 15.

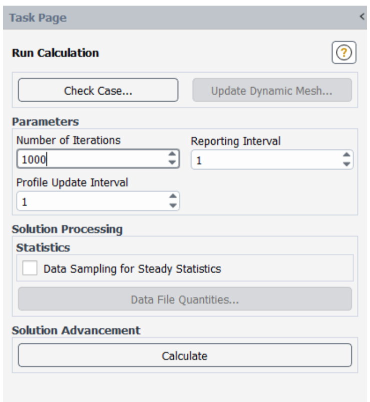

Figure 15: Run Calculations

Also the solution controls and the absolute criteria residuals can be seen in the following figures:

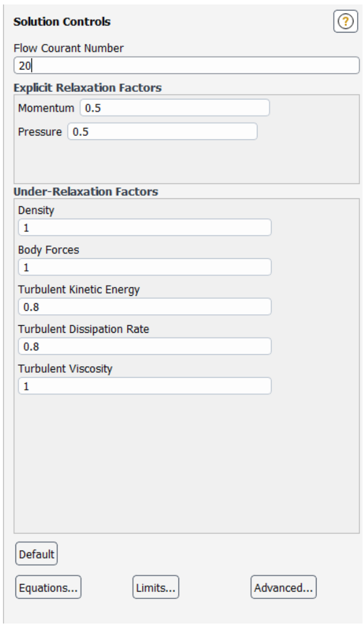

Figure 16: Solution controls

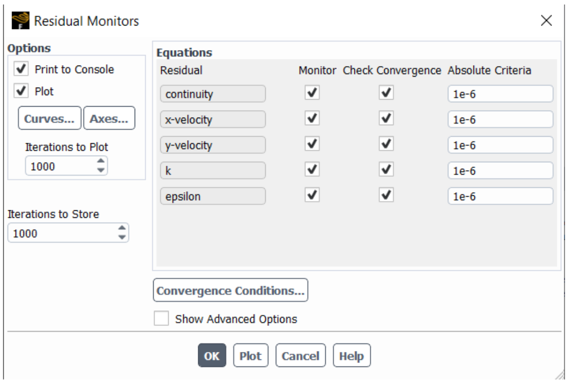

Figure 17: Residuals criteria

3 Results and Discussion

One simulation was performed.

3.1 Residuals

Figure 18 shows the residuals over the number of iterations required to converge the solution. The convergence criteria chosen was 1e-6. It is possible to verify two main aspects: first, all the residuals converged to small values and second during all the simulations despite 1000 as the maximum number of iterations most of the simulations did not converge to the expected tolerance since 1000 iterations seems to be not sufficient.

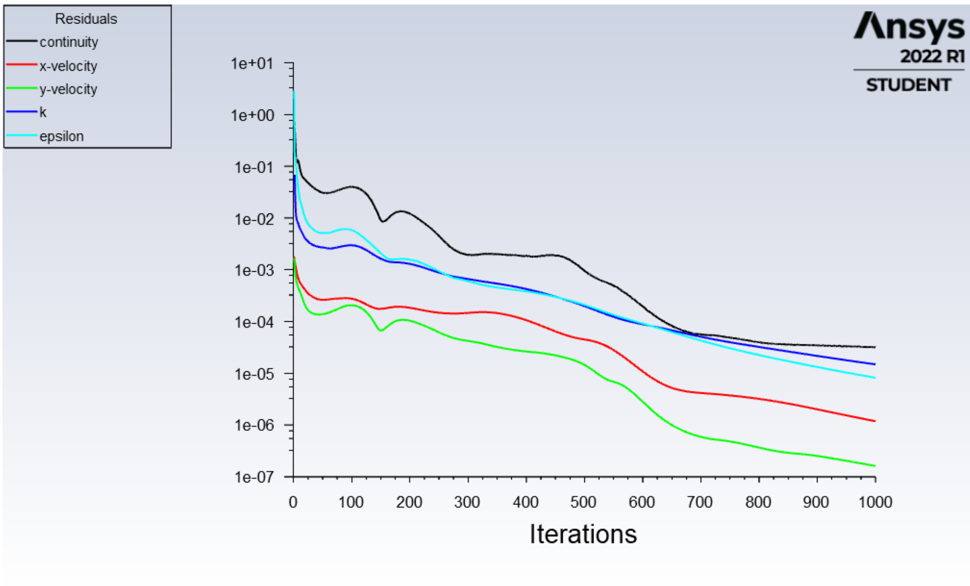

Figure 18: Residuals

iter	continuity	x-velocity	y-velocity	k	epsilon	pressure_c	skin_frict	time/iter	
991	2.8930e-05	1.2101e-06	1.8811e-07	2.0090e-05	1.1151e-05	-2.2716e-01	1.0853e-02	0:00:02	9
992	2.8699e-05	1.2039e-06	1.8734e-07	2.0018e-05	1.1105e-05	-2.2717e-01	1.0854e-02	0:00:01	8
993	2.8640e-05	1.1978e-06	1.8652e-07	1.9947e-05	1.1060e-05	-2.2718e-01	1.0854e-02	0:00:01	7
994	2.8926e-05	1.1919e-06	1.8569e-07	1.9875e-05	1.1015e-05	-2.2720e-01	1.0854e-02	0:00:01	6
995	2.8486e-05	1.1860e-06	1.8485e-07	1.9803e-05	1.0970e-05	-2.2721e-01	1.0855e-02	0:00:00	5
996	2.8838e-05	1.1799e-06	1.8401e-07	1.9732e-05	1.0926e-05	-2.2722e-01	1.0855e-02	0:00:01	4
997	2.8825e-05	1.1740e-06	1.8329e-07	1.9662e-05	1.0879e-05	-2.2723e-01	1.0855e-02	0:00:01	3
998	2.8620e-05	1.1681e-06	1.8247e-07	1.9592e-05	1.0835e-05	-2.2725e-01	1.0856e-02	0:00:00	2
999	2.8882e-05	1.1622e-06	1.8175e-07	1.9522e-05	1.0791e-05	-2.2726e-01	1.0856e-02	0:00:00	1
1000	2.8792e-05	1.1561e-06	1.8100e-07	1.9452e-05	1.0747e-05	-2.2727e-01	1.0856e-02	0:00:00	0

Figure 19: Console iterations

3.2 Pressure Coefficient and Skin Friction Coefficient

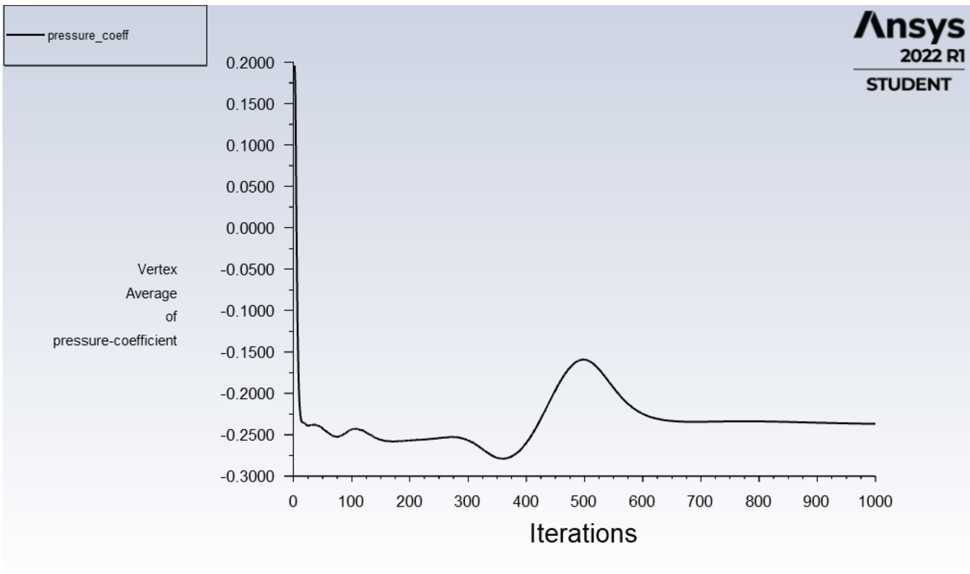

Figure 20: Pressure coefficient as a function of iterations

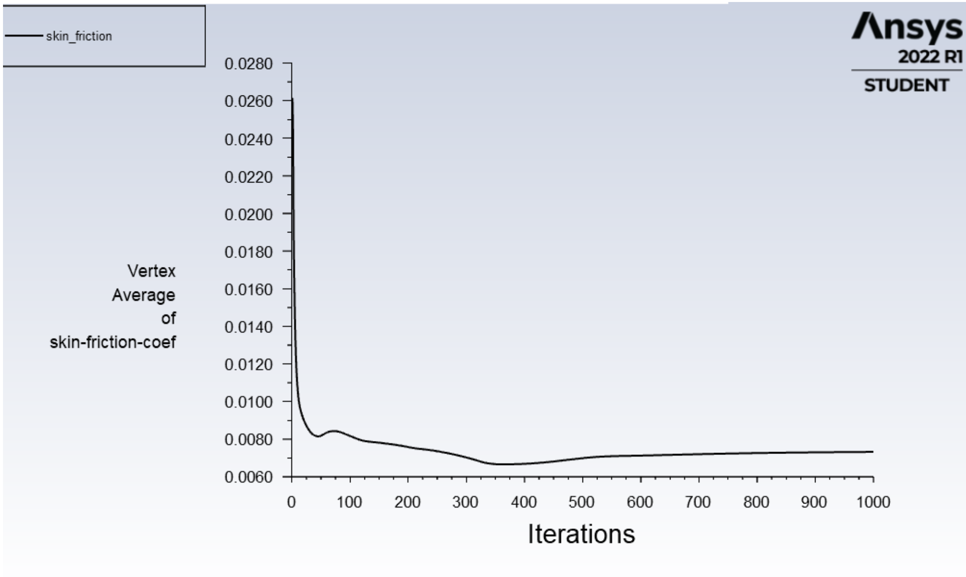

Figure 21: Skin friction coefficient as a function of iterations

3.3 Streamlines

It is possible to notice a re-circulation flow after the step due to the adverse pressure gradient.

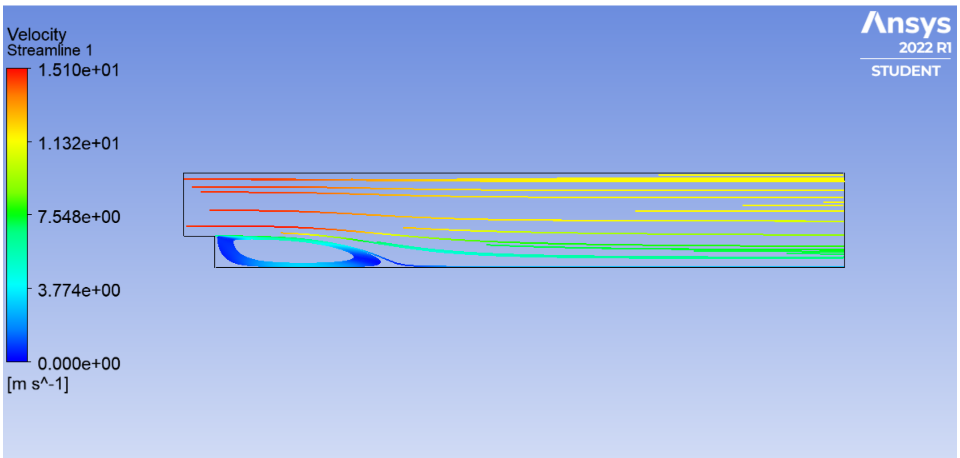

Figure 22: Streamlines

3.4 Velocity Contour

Figures 23 to 25 show the velocity contour of all previous simulations. The plots seem correct accordingly to the problem physics.

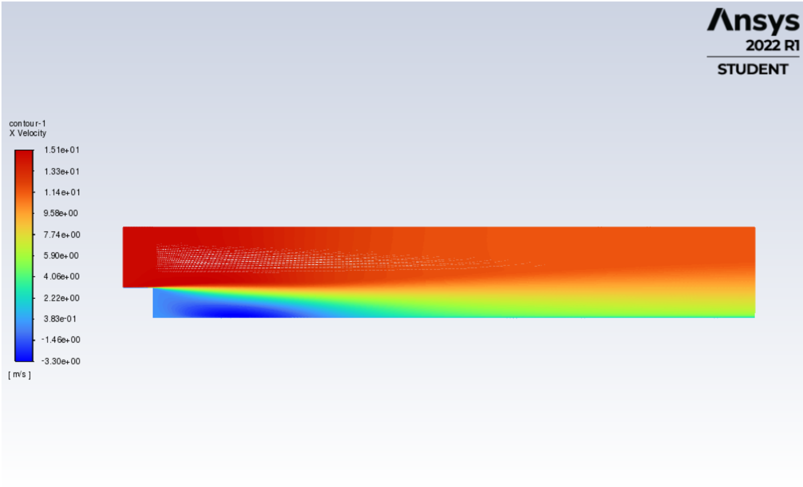

Figure 23: X Velocity contour

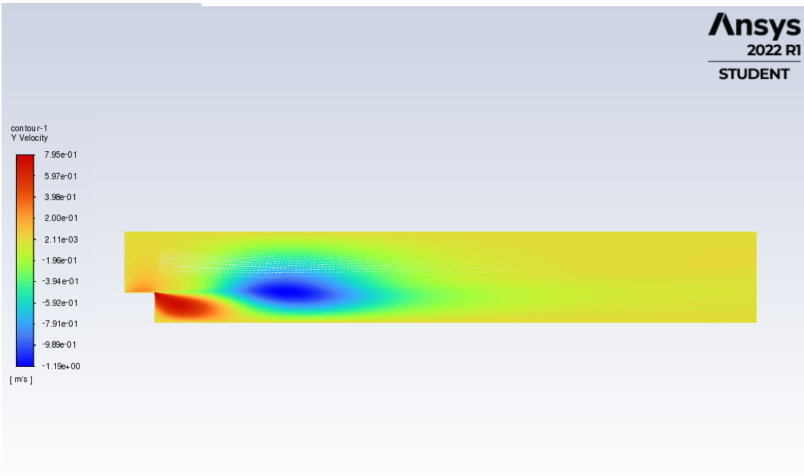

Figure 24: Y Velocity contour

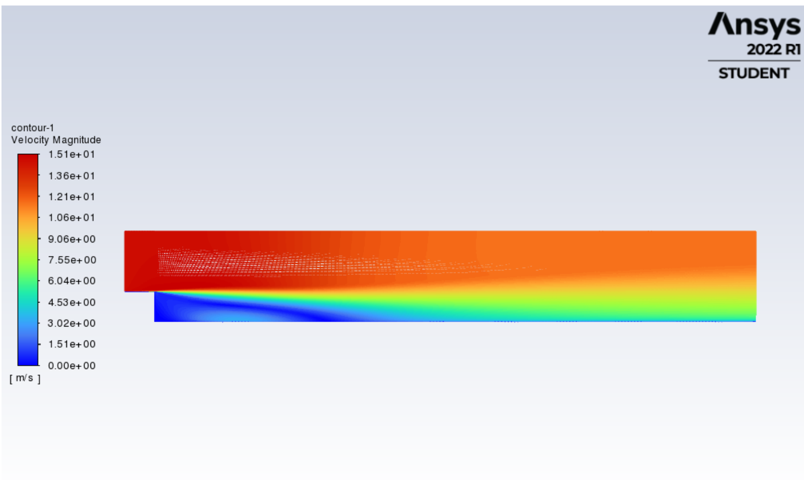

Figure 25: Magnitude Velocity contour

Looking at the figures it is possible to infer that since the magnitude velocity contour plot seems very similar to the x velocity contour, u component is way more relevant than the v (y component).

3.5 Turbulence

Figures 26 to 27 show the turbulent aspects of all previous simulations. The plots seem correct accordingly to the problem physics.

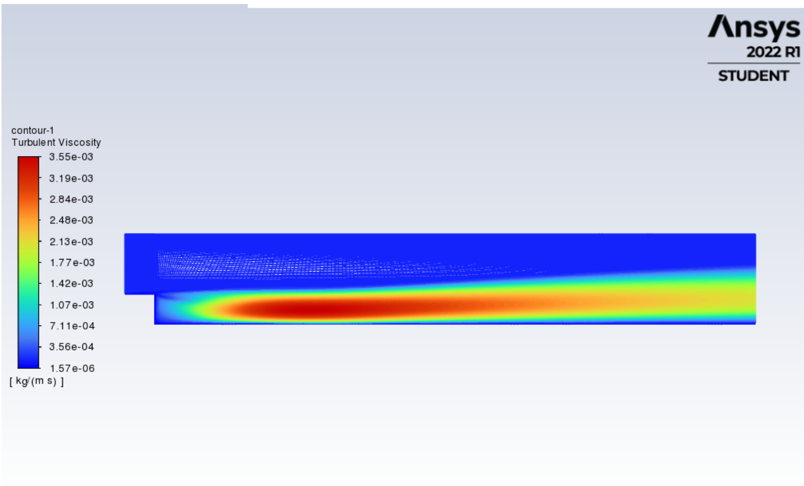

Figure 26: Turbulent viscosity contour

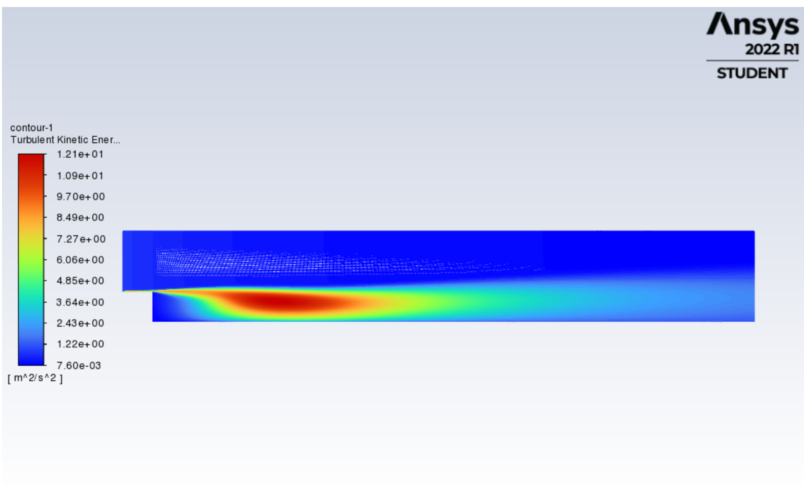

Figure 27: Turbulent kinetic contour

3.6 Pressure and Skin Friction coefficients of X on lower wall

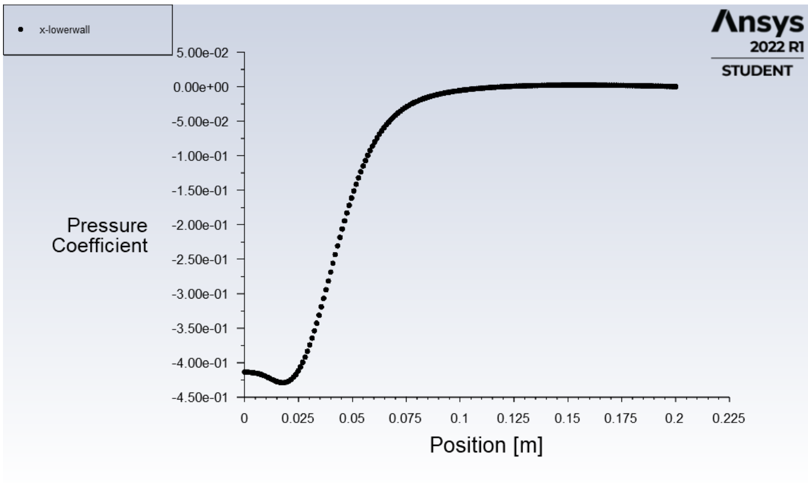

Figure 28: Pressure Coefficient at lower wall

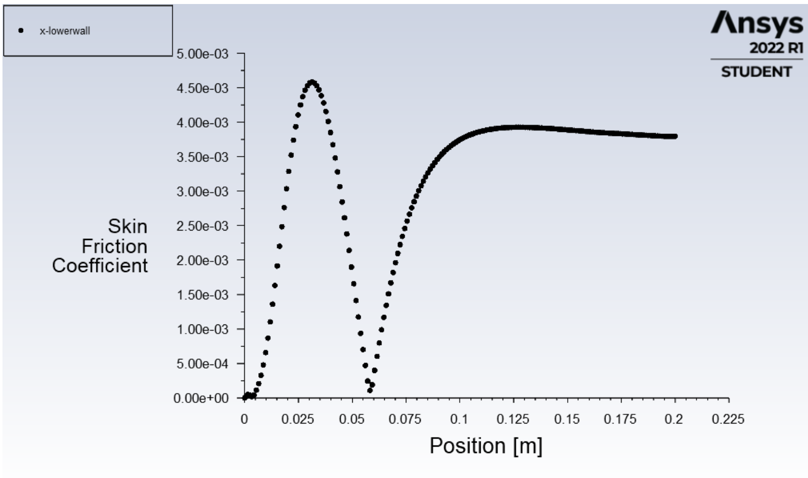

Figure 29: Skin Friction Coefficient at lower wall

It is possible to notice that the Pressure Coefficient reaches its lower value near the step. On the other hand the Skin Friction coefficient reaches its higher value close to the step.

3.7 XY plot

Figures 19 to 20 show the XY of all previous simulations. The plots seem correct accordingly to the problem physics.

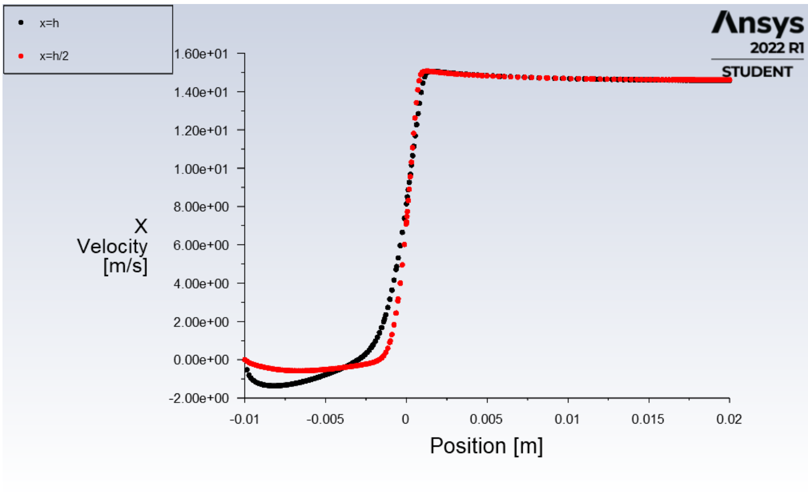

Figure 30: X velocity plots

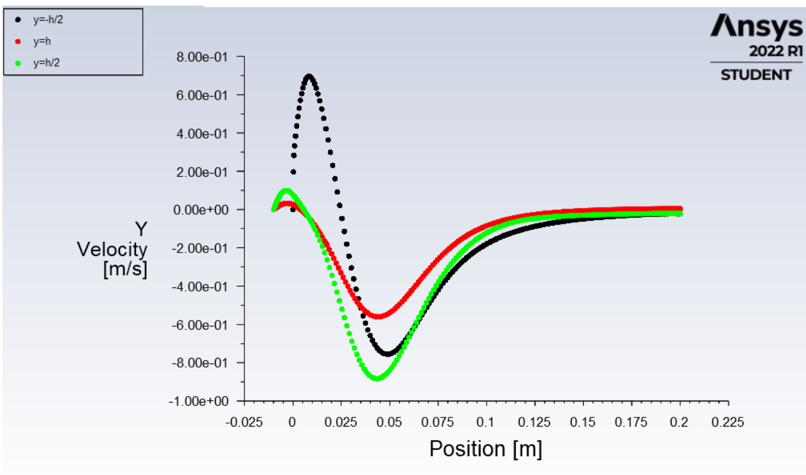

Figure 31: Y velocity plots

It is possible to notice that:

In relation to X we have that for the two set values the value remains stable for values in Y above 0. Note that for negative values of Y for a distance of H in x we have negative values for this speed which should cause the fluid re-circulation. In relation to Y we have that the black line that represents a point close to the re-circulation state the velocity in this component as expected is greater than the others present.