Technical Report

ME 412

Numerical Thermo-Fluid Mechs

Second Ansys Project

Author:
Arturo Machado Burgos

Submitted to:
Dr. Surya Pratap Vanka

February 28, 2022

0.1 Problem Description

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

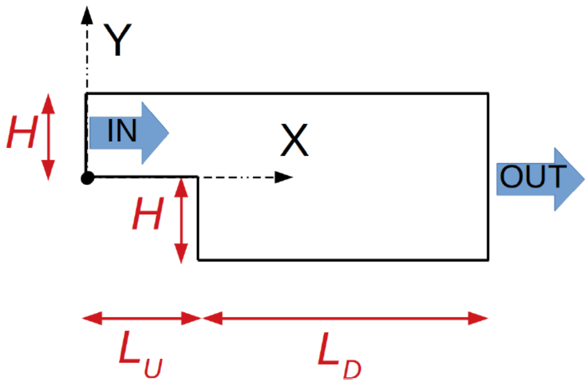


Figure 1: Backward facing step

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

Table 1: Geometry elements and its magnitude

Geometry Element	**Magnitude**
H	1 $[m]$
L_u	1 $[m]$
L_d	5 $[m]$

It is required to solve the steady state solution at two values of Reynolds number: 50 and 100, by setting appropriate values of U.

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

$$Re = \frac{2\rho U H}{\mu} \tag{1}$$

Where:

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

U is the velocity that is set in the following code provided by the Professor Vanka in order to calculate the velocity inlet profile.

```
#include "udf.h"
#define U 50 //unit m/sec
#define H 1 //unit m
DEFINE_PROFILE(inlet_x_velocity, thread, position)
{
real x[ND_ND]; /* this will hold the position vector */
real y;
face_t f;
begin_f_loop(f, thread)
{
F_CENTROID(x,f,thread);
y = x[1];/* Y co-ordinate*/
F_PROFILE(f, thread, position) = 6*U*y*(H-y)/(H*H); /*velocity
profile*/
}
end_f_loop(f, thread)
}
```

The only observation considering the code is that aiming to reach 50 and 100 as the Reynolds number, on line "2" the U variable had to be changed according to the relation: $U = 50[m/s]$ corresponds to $Re = 100$ and $U = 25[m/s]$ corresponds to $Re = 50$.

Solve for two values of Reynolds numbers: 50 and 100 for each case. Repeat the following for each of the TWO problems:

1. Plot all 3 residuals
2. Plot contours of velocity magnitude
3. Plot Streamlines
4. Five XY plots of X component of velocity at inlet, outlet, X = 1.5, X = 2 and X = 3 in a single window

For each problem, please include the above four plots along with a single common mesh plot. Also, include a discussion of the effects of Reynolds number on the flow.

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:

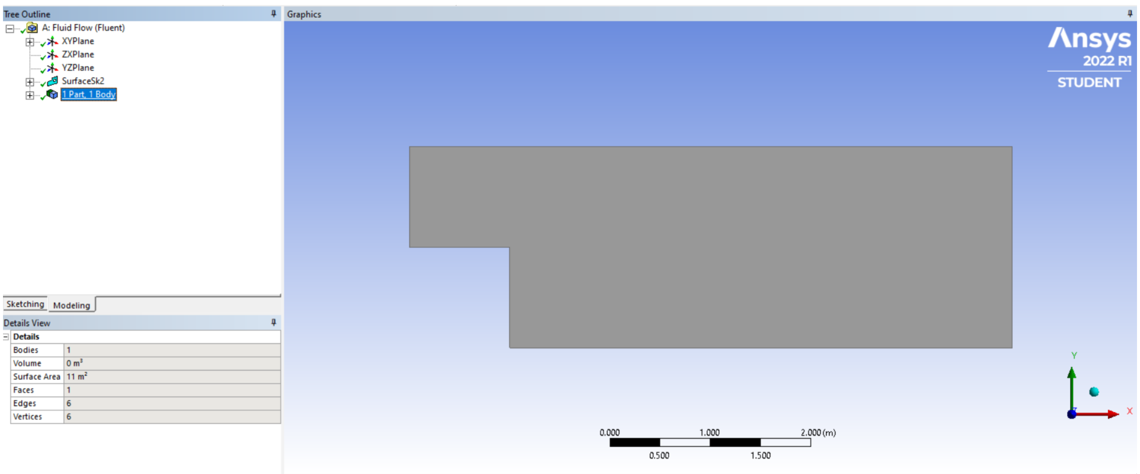


Figure 2: Geometry's constrains

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, using a quadrilateral mesh conformation, each mesh element had a size of 2e-002 m.

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	2.e-002 m
Advanced	
Defeature Size	Default (1.5811e-003 m)
Behavior	Soft
Growth Rate	Default (1.2)
Capture Curvature	No
Capture Proximity	No

Figure 3: Mesh properties

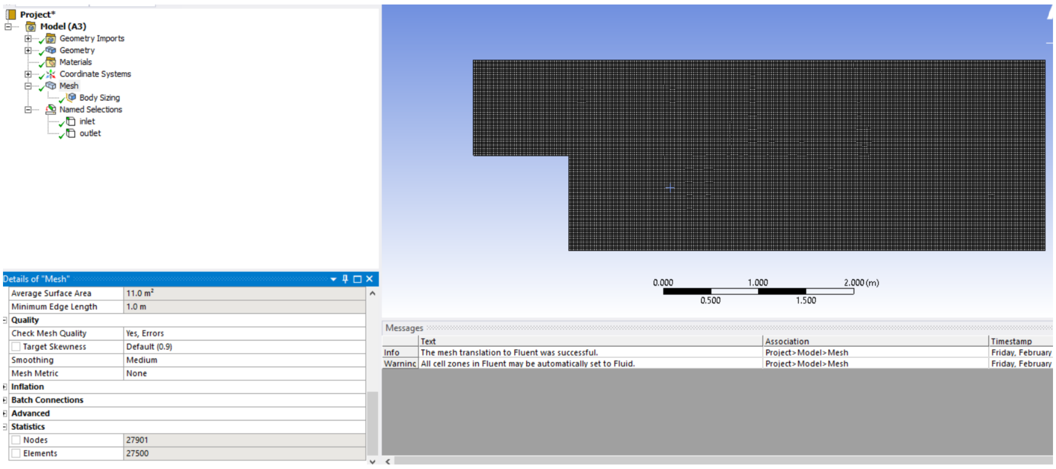


Figure 4: Geometry's mesh

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 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

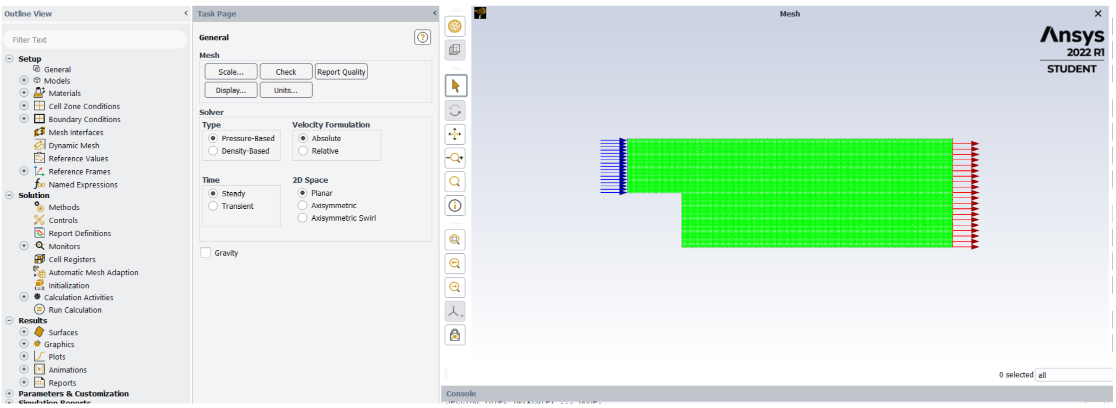


Figure 5: Initial Fluent configuration

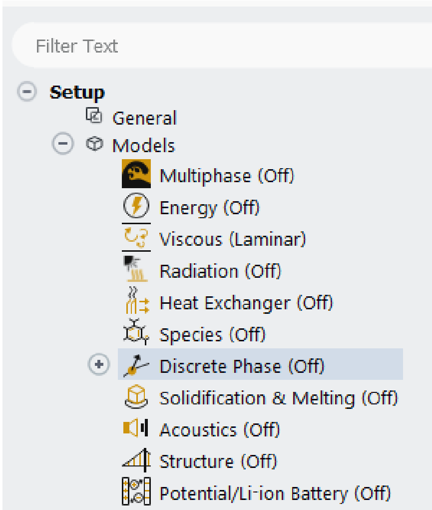


Figure 6: Models setup

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

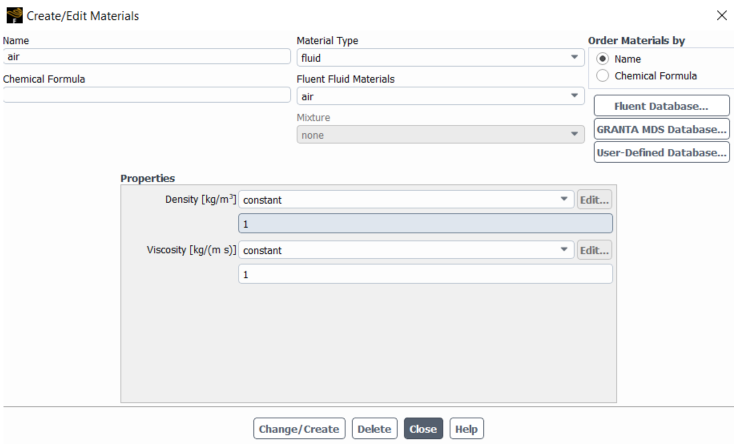


Figure 7: Air properties

For the Momentum Boundary conditions the walls apart form the inlet and outlet 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 the c code provided above. for the outlet wall the condition was set as outflow.

0.2.4 Set Solution Initialization and Methods

Figures 8 and 9 show the solution initialization and the methods used during the procedure. For the Methods it was used the SIMPLE Scheme and the other parameters were set as the default ones.

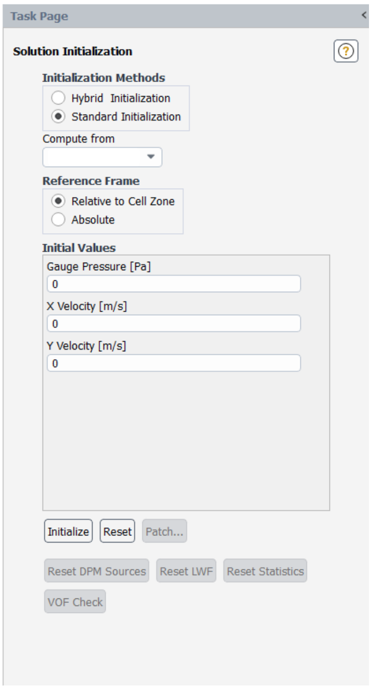


Figure 8: Solution Initialization

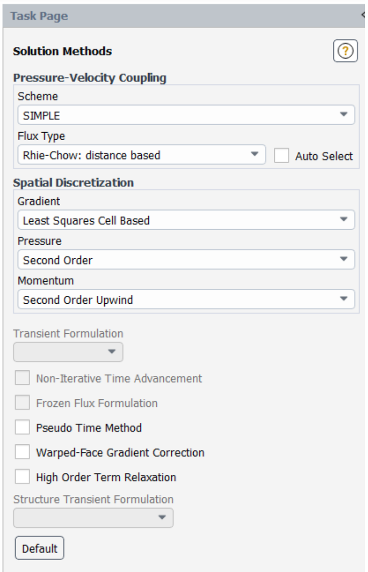


Figure 9: Solution Methods

Regarding the "Run Calculations" it was set the maximum number of iterations as 5000, what can be seen in Fig. 10.

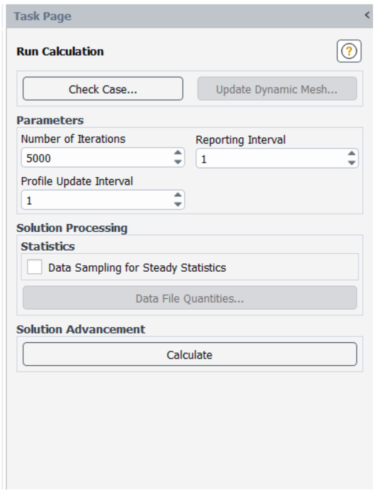


Figure 10: Run Calculations

0.3 Results and Discussion

Two simulations were performed. The first one with 1000 elements and Re = 100. The second with Ra = 50.

0.3.1 Residuals

Figures 11 to 12 show the residuals for continuity, and momentum (represented by x-velocity and y-velocity) over the number of iterations required to converge the solution. The convergence criteria chosen was 1e-4. It is possible to verify two main aspects: first, all the residuals converged to small values and second during all the simulations despite 5000 as the maximum number of iterations no simulation required such amount of cycles as shown in the Tab. 2.

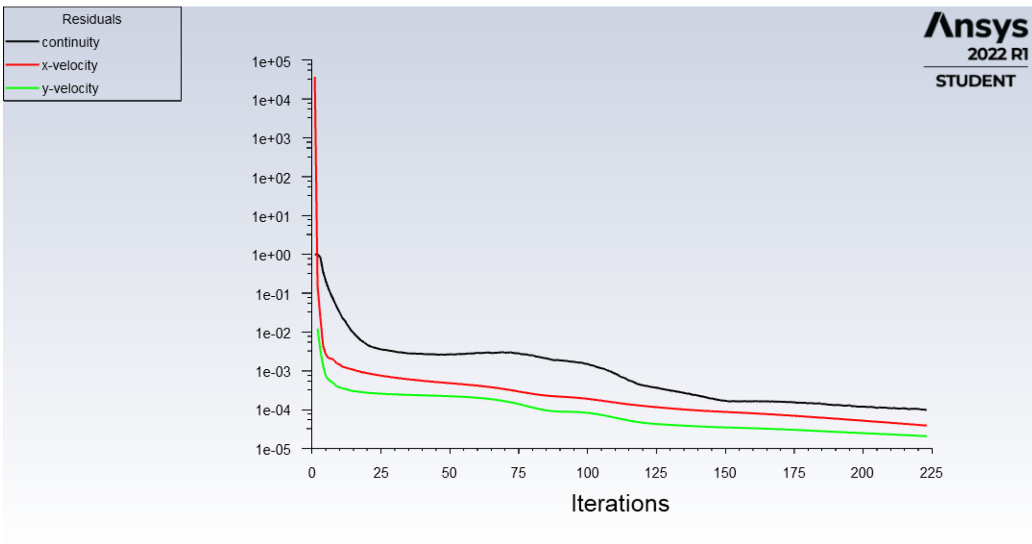


Figure 11: Residuals for Ra = 100

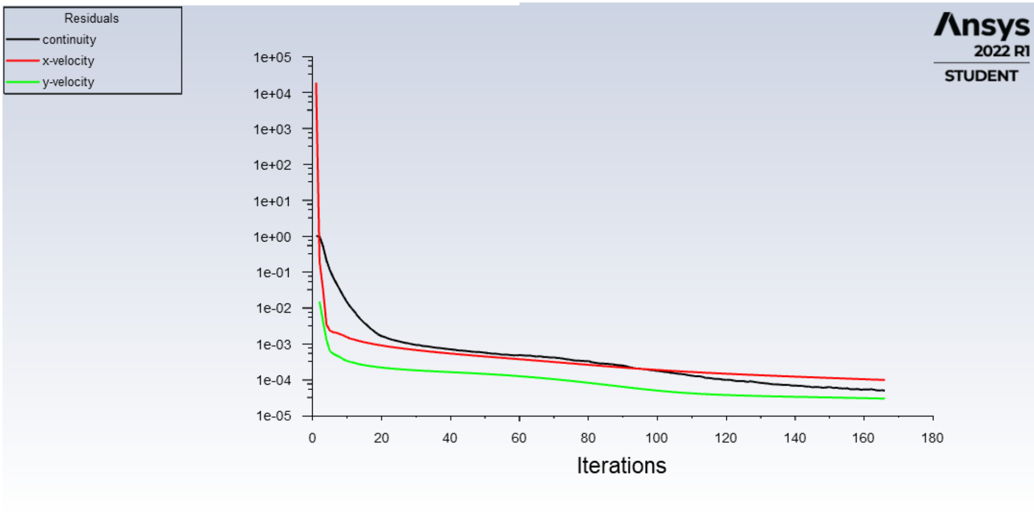


Figure 12: Residuals for Ra = 50

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 Iterations**
Ra = 100	223
Ra = 50	166

0.3.2 Velocity Contour

Figures 13 to 14 show the velocity contour of all previous simulations. The plots seem correct accordingly to the problem physics.

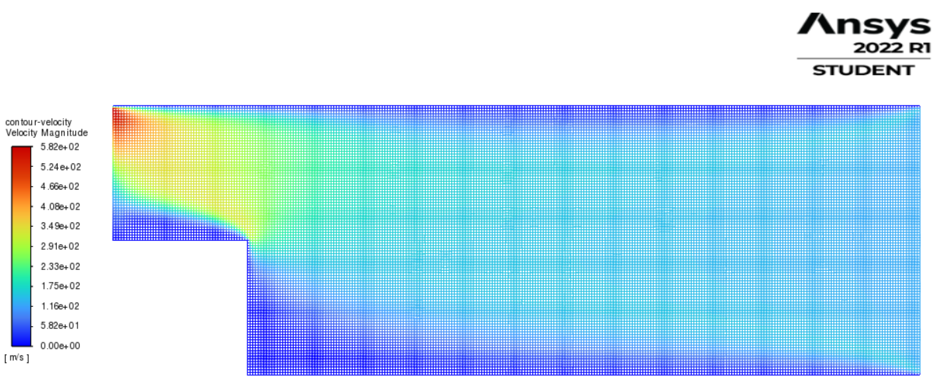


Figure 13: Velocity contour for Ra = 100

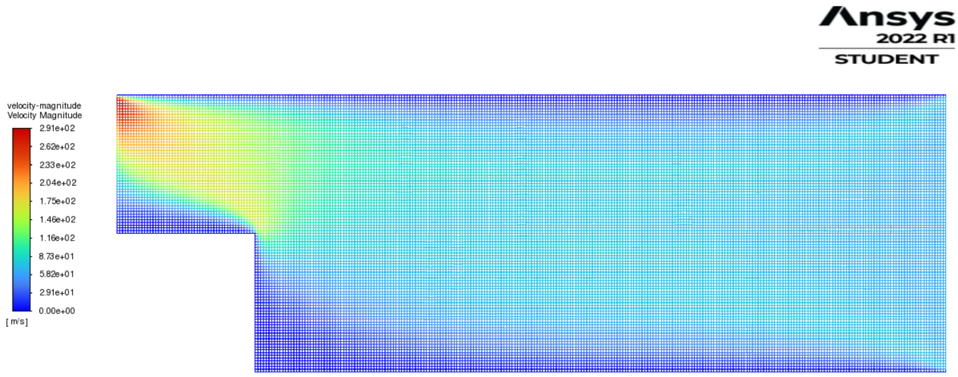


Figure 14: Velocity contour for Ra = 50

0.3.3 Streamlines

Figures 15 to 16 show the streamlines of all previous simulations. The plots seem correct accordingly to the problem physics.

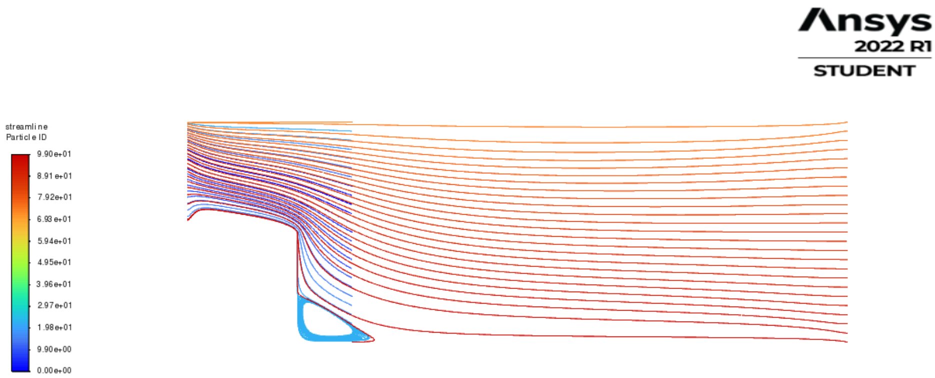


Figure 15: Streamline for Ra = 100

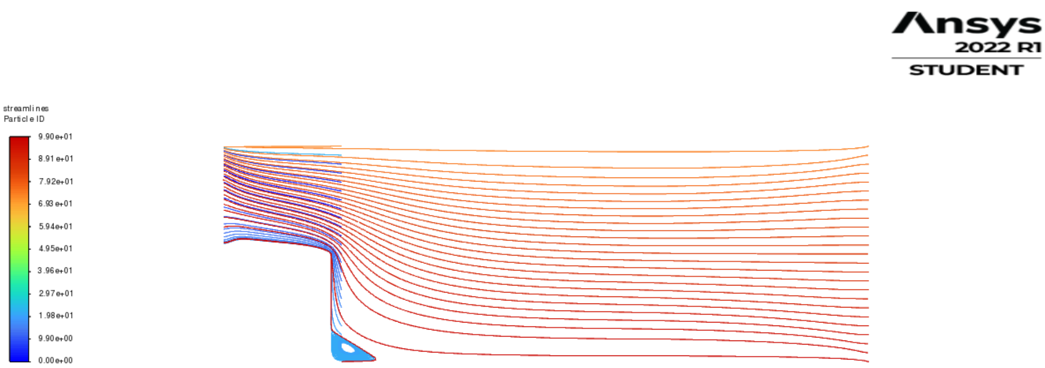
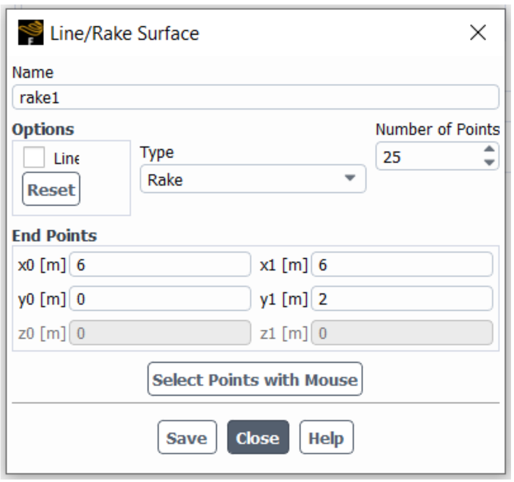
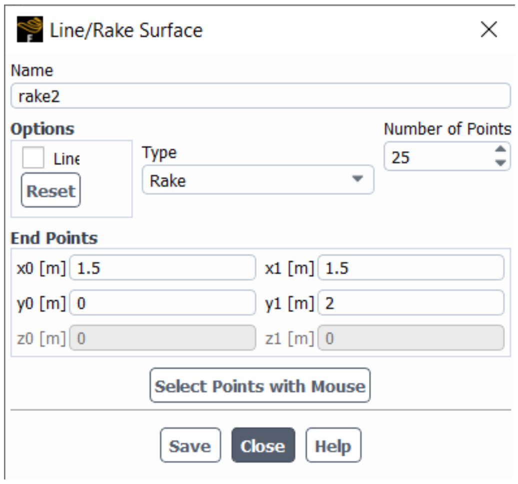


Figure 16: Streamline for Ra = 50

Important to mention that in this step many difficulties were found. In the beginning the streamline plot was not showing the the re-circulation flow. To fix that it was created two rake for each Reynolds's number simulation. Figures 21 and 18 show the two rake for each simulation.

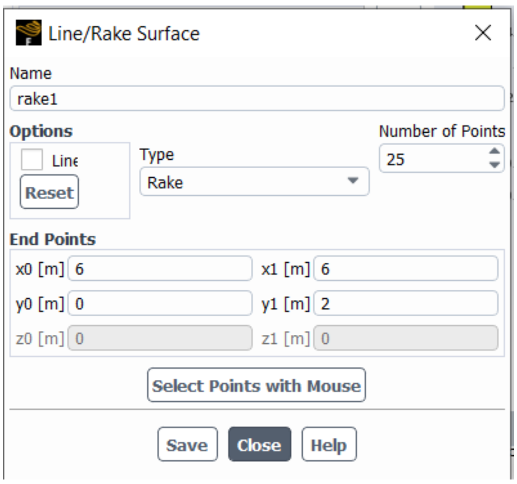


(a) Rake 1

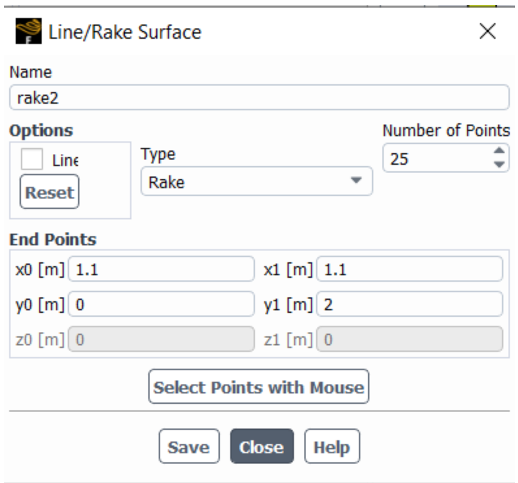


(b) Rake 2

Figure 17: Rake's for Re = 100



(a) Rake 1



(b) Rake 2

Figure 18: Rake's for Re = 50

0.3.4 XY plot

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

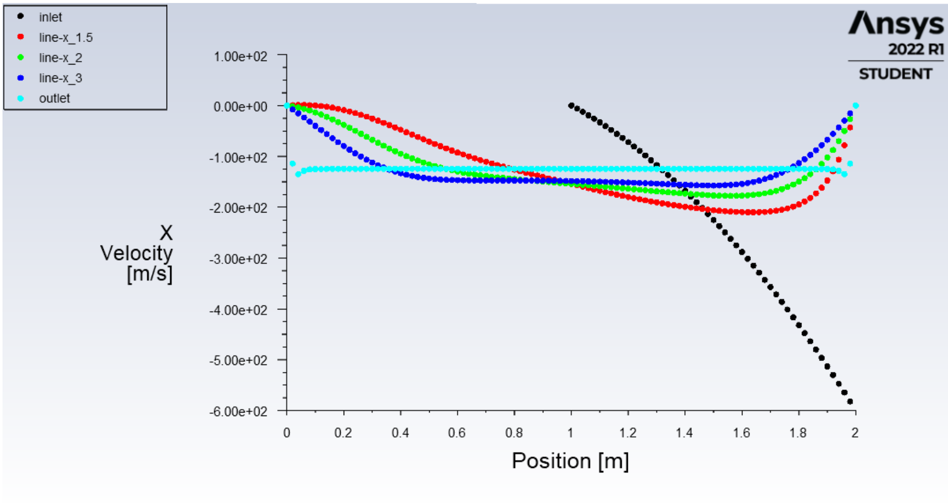


Figure 19: XY for Ra = 100

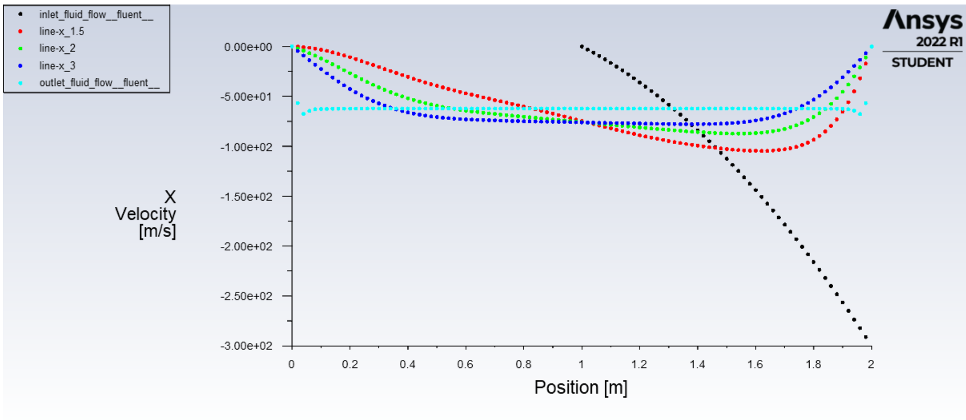
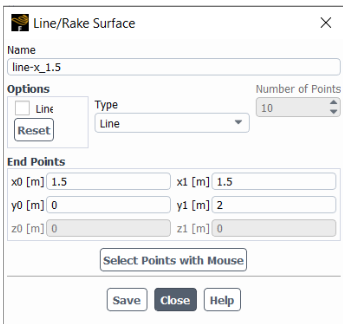
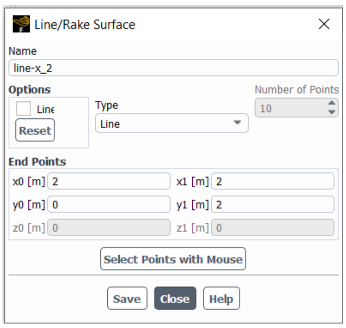


Figure 20: XY for Ra = 50

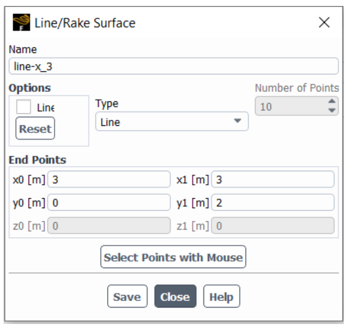
Figures 21a, 21b and 21c show the lines



(a) Line 1



(b) Line 2



(c) Line 2

Figure 21: Rake's for Re = 100

0.3.5 Discussion

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

- With higher values of Reynolds as demonstrated in the simulation of 100 higher values are seen for the velocity contour obtained

- In relation to the 100 Reynolds's number it is possible to note a higher fluid re circulation pattern which is justified with the higher inlet velocity. Everything is in accordance with physics of the problem