Technical Report

ME 412

Numerical Thermo-Fluid Mechs

Fifth Ansys Project

Author:
Arturo Machado Burgos

Submitted to:
Dr. Surya Pratap Vanka

April 21, 2022

1 Problem Description

The objective of this project is to solve the Turbulent Mixing of Hot and Cold Water. It consists in the following problem: simulate a 2D inlet flow over the geometry.

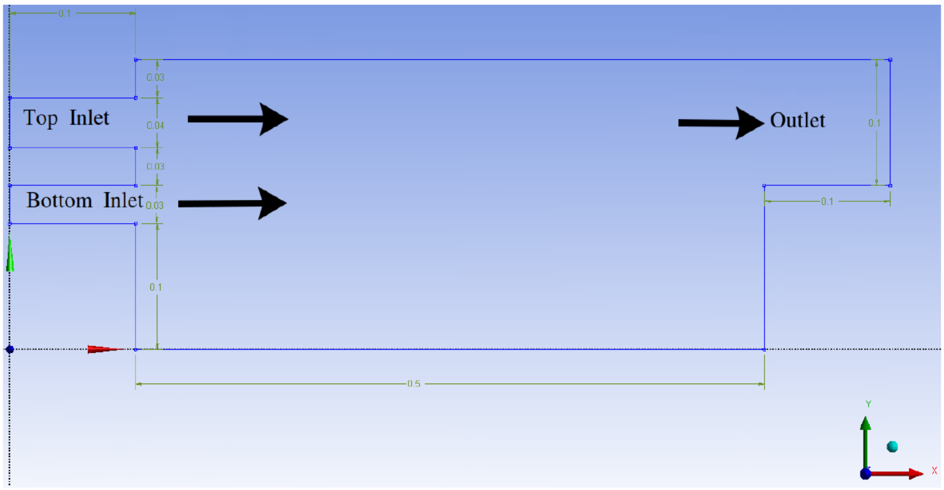


Figure 1: Turbulent mixing of hot and cold water

Unlike previous reports, this one did not require a specific Reynolds number to be determined. Solve this problem and include the following plots:

1. Mesh
2. All residuals as a function of iterations
3. Streamlines
4. X velocity contours
5. Y velocity contours
6. Velocity magnitude contours
7. Turbulent Eddy dissipation contours
8. Turbulent kinetic energy contours
9. Pressure
10. Temperature

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. 1:

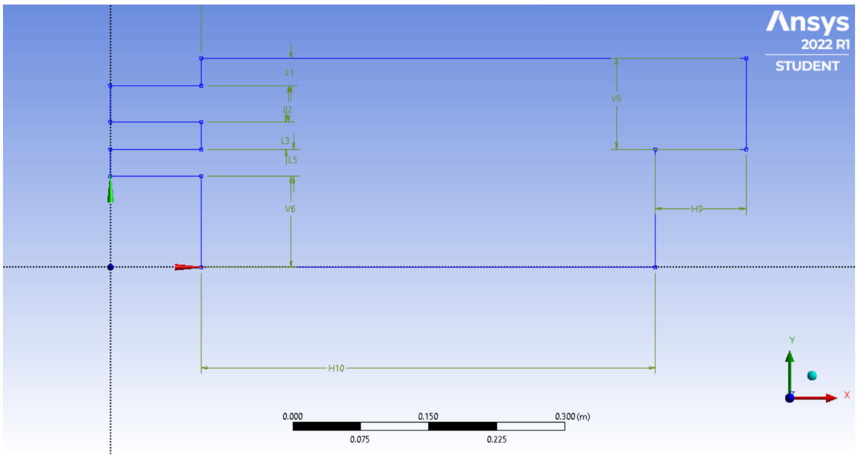


Figure 2: Geometry's constrains

2.2 Mesh

As the next step, it started to create the mesh for the body to be simulated. As seen in Figs. 3 to 5, using Face Mesh conformation and face sizing with 0.005 m as element size.

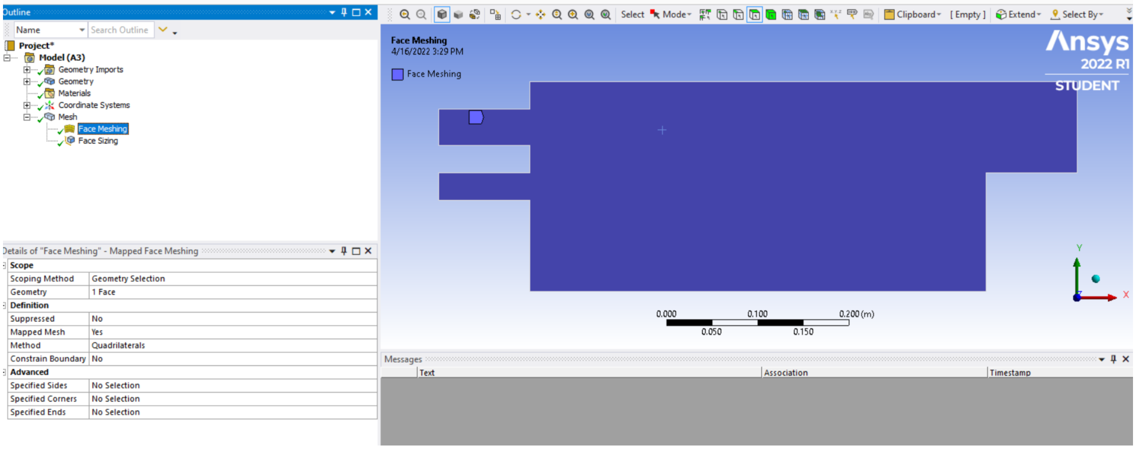


Figure 3: Face Mesh

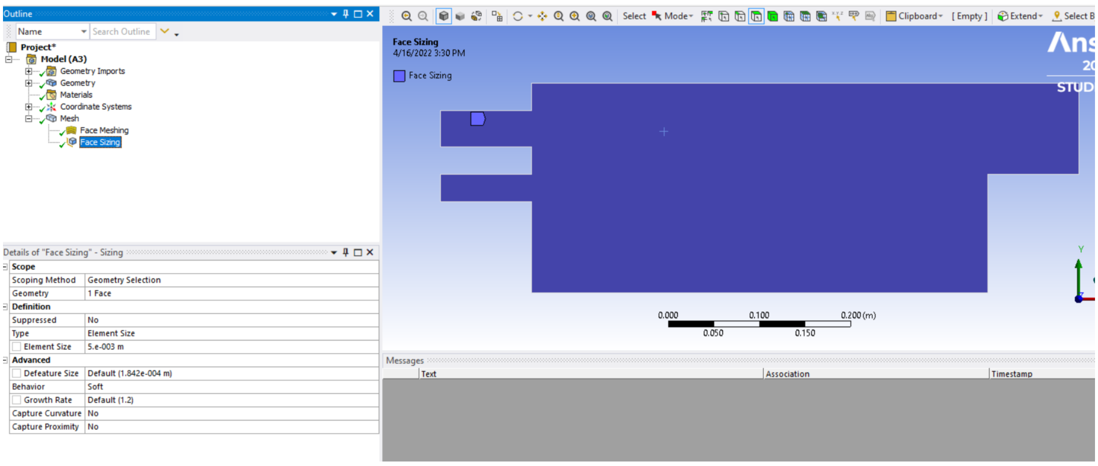


Figure 4: Face sizing

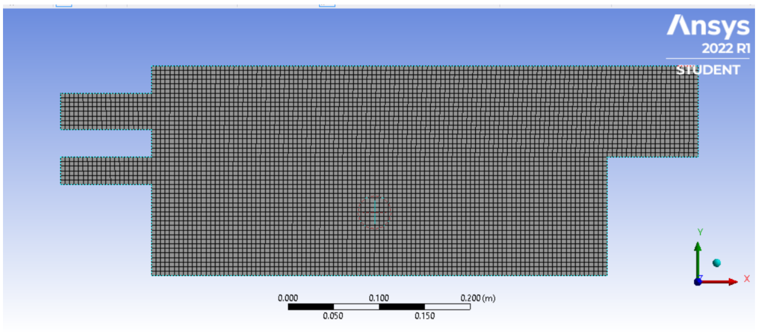


Figure 5: Final mesh

Then it was defined the named selections accordingly.

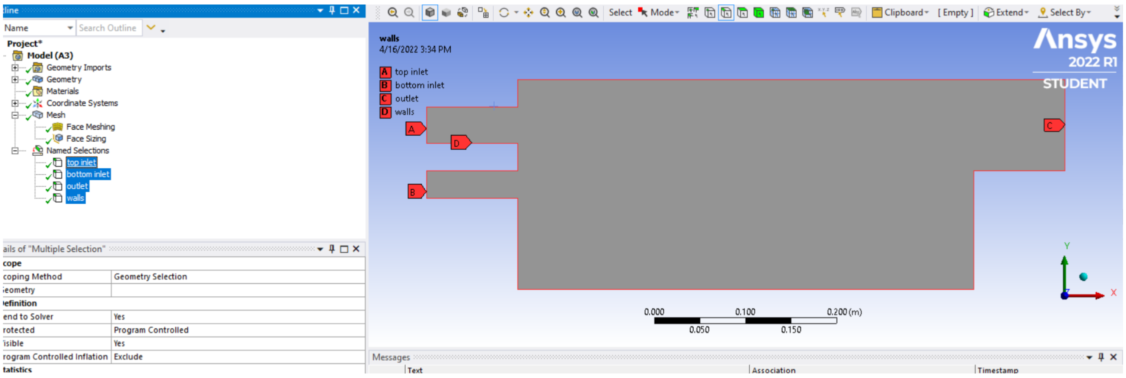


Figure 6: Named selections

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. 7. Apart from that, on models only the Viscous option was set as k-epsilon, Energy was also set to on, while the others were off. This can be seen in Fig. 8

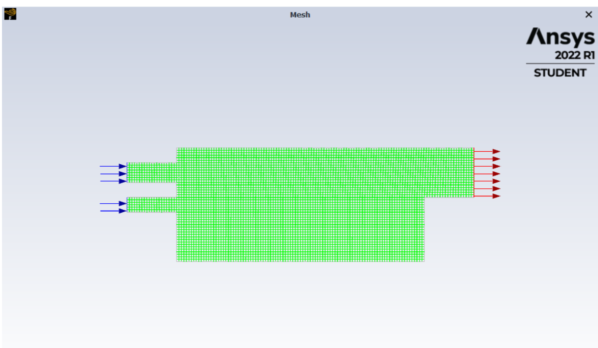


Figure 7: Initial Fluent configuration

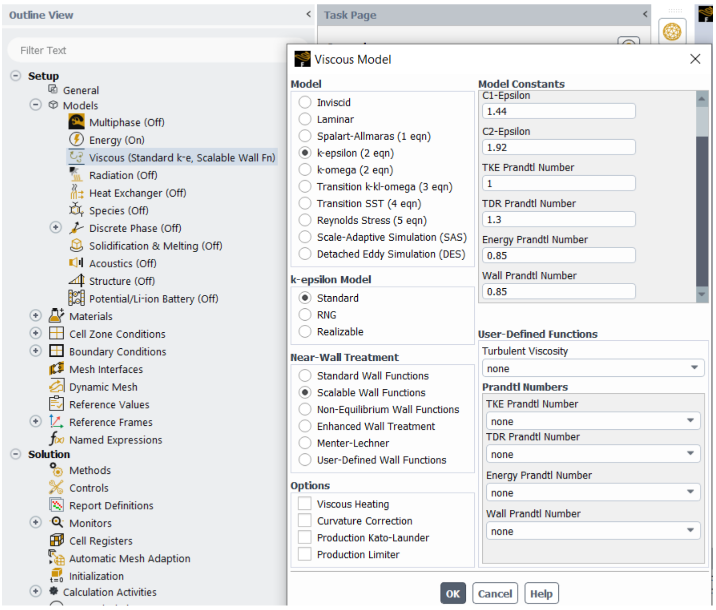


Figure 8: Models setup

As for the fluid selected, it was chosen liquid water with the following properties as show in Fig. 9

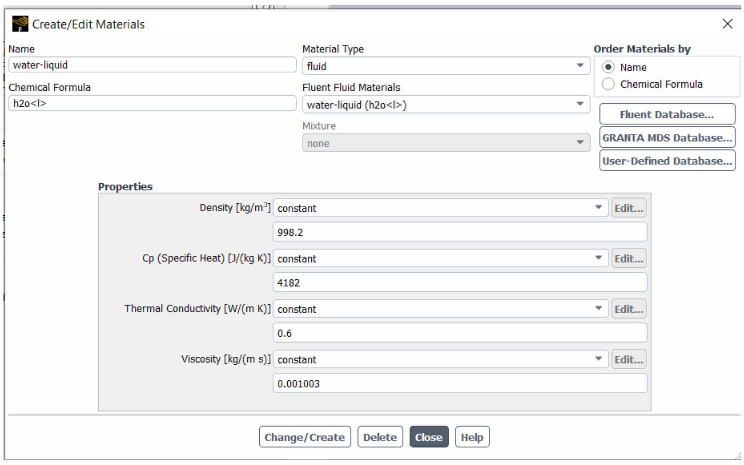
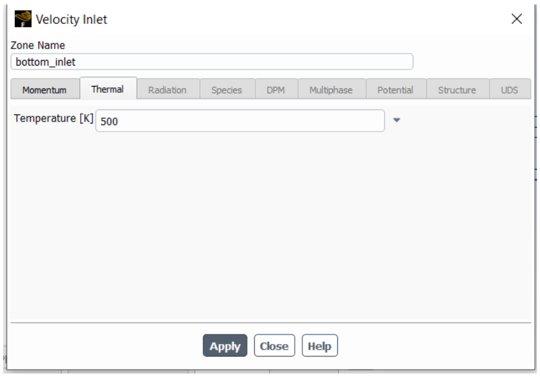
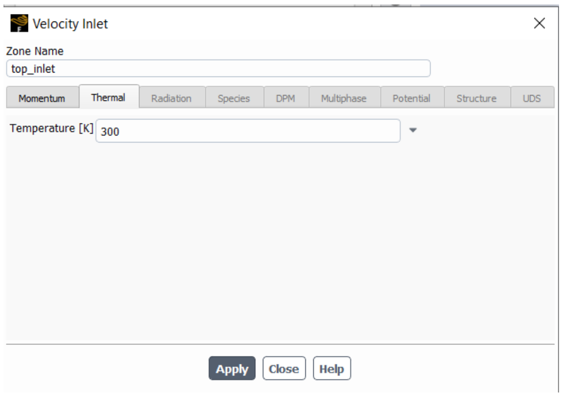


Figure 9: Liquid water properties

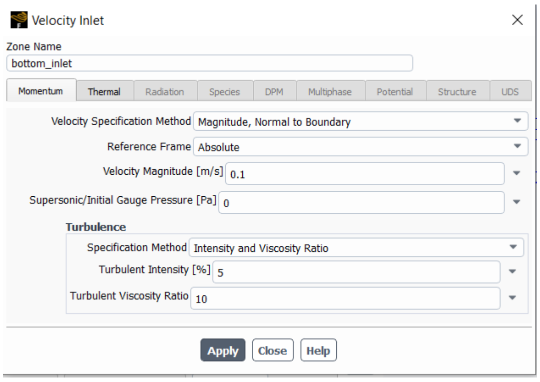
Figure 10 show the Boundary Conditions used:



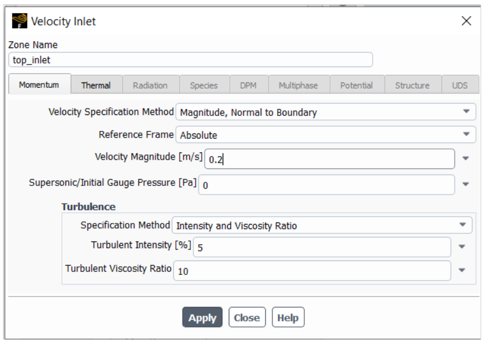
(a) Temperature bottom inlet



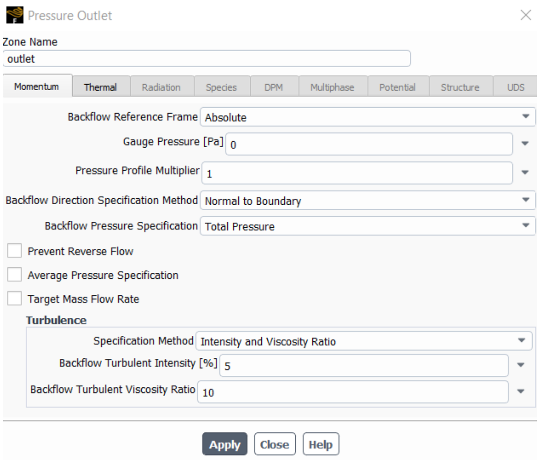
(b) Temperature top inlet



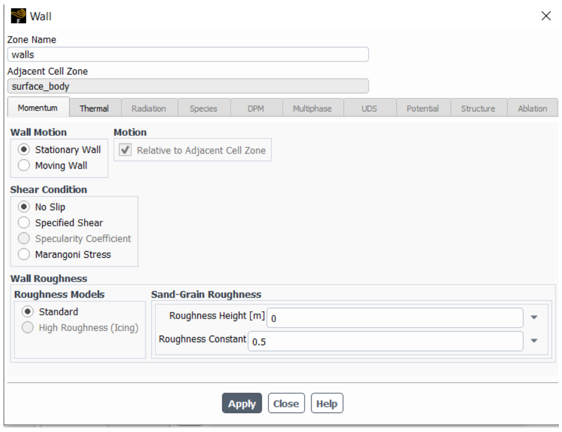
(c) Velocity bottom inlet



(d) Velocity top inlet



(e) Outlet

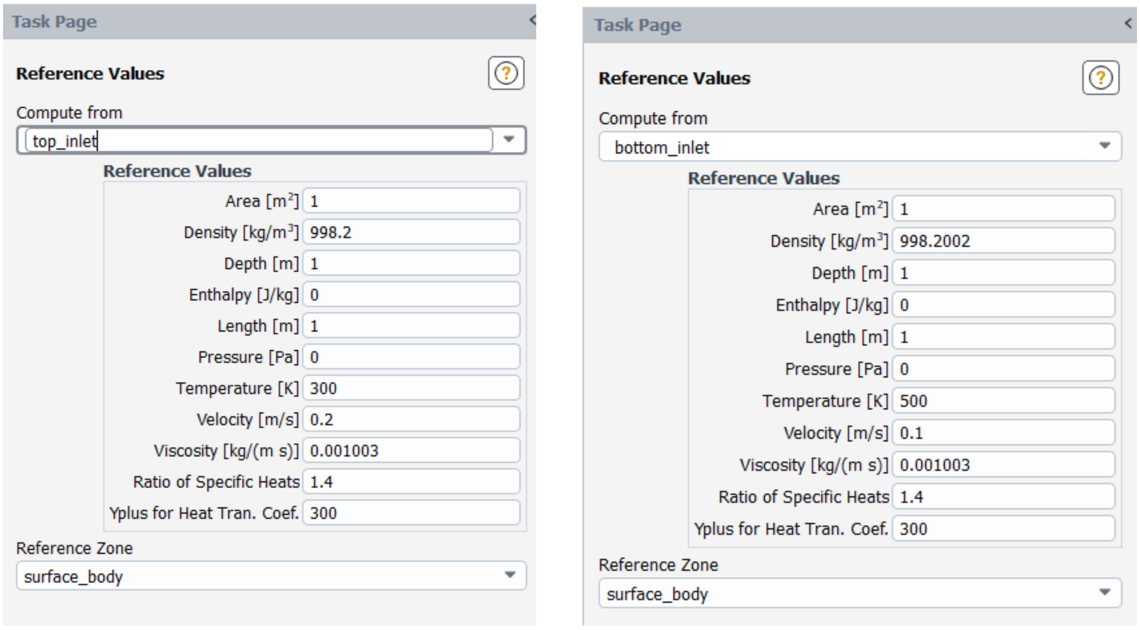


(f) Walls

Figure 10: Boundary Conditions

2.4 Set Solution Initialization and Methods

Figures 11 and 12 show the reference values and the methods used during the procedure. The solution initialization was hybrid.



(a) Reference values top (b) Reference values bottom

Figure 11: Reference values

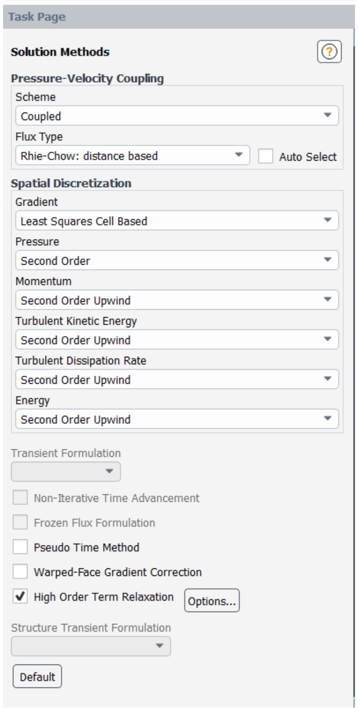


Figure 12: Solution methods

Regarding the "Run Calculations" it was set the maximum number of iterations as 1000, what can be seen in Fig. 13.

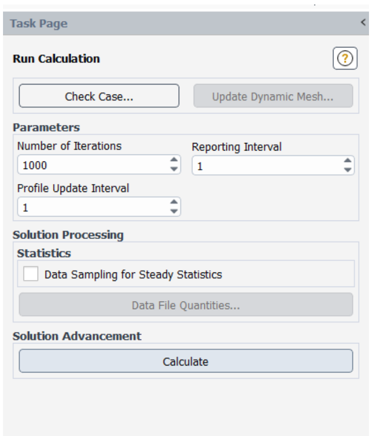


Figure 13: Run Calculations

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

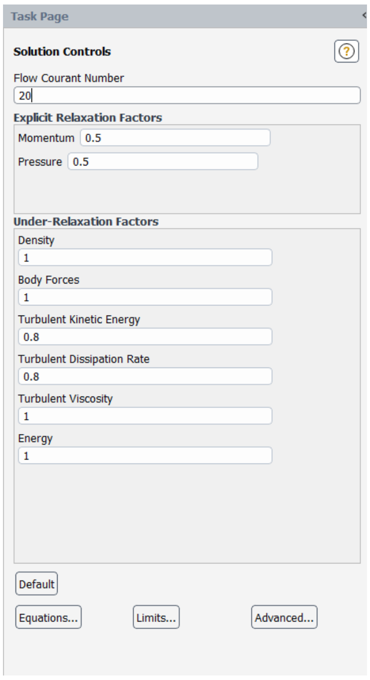


Figure 14: Solution controls

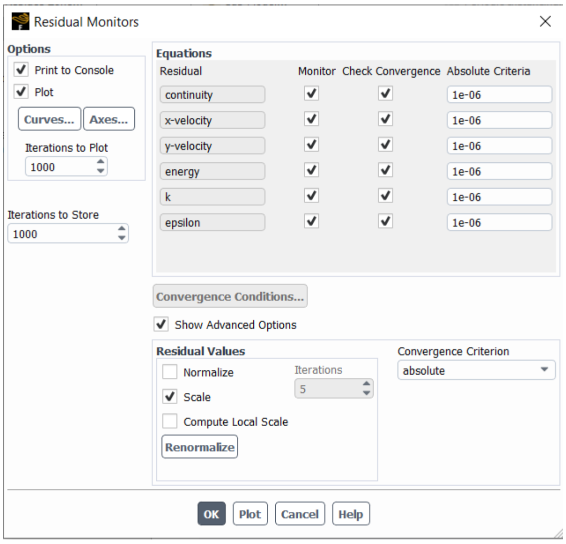


Figure 15: Residuals criteria

3 Results and Discussion

One single simulation was performed.

3.1 Residuals

Figure 16 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 continuity did not converge to the expected tolerance since 1000 iterations seems to be not sufficient. Moreover it seems that the residuals have converged to a value that would not be possible to decrease even further, apparently they reach a stationary value despite the increasing number of iterations.

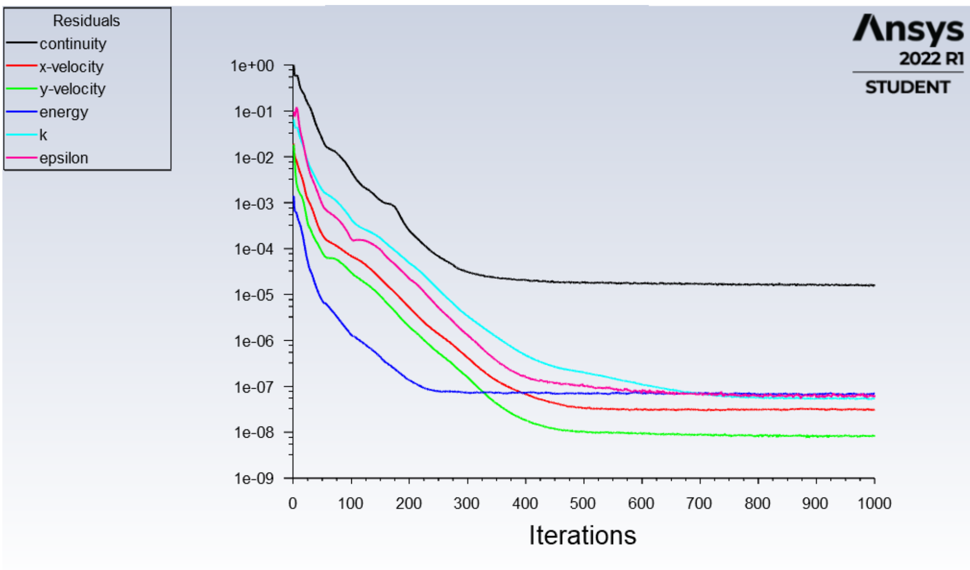


Figure 16: Residuals

iter	continuity	x-velocity	y-velocity	energy	k	epsilon	pressure-c	time/iter	
991	1.5988e-05	3.0615e-08	8.3865e-09	6.7946e-08	5.2225e-08	6.2349e-08	-2.9653e-01	0:00:00	9
992	1.6258e-05	3.0901e-08	8.2573e-09	6.6972e-08	5.3438e-08	6.4222e-08	-2.9653e-01	0:00:02	8
993	1.6120e-05	3.0808e-08	8.3945e-09	7.0242e-08	5.2311e-08	6.4707e-08	-2.9653e-01	0:00:01	7
994	1.6069e-05	3.0834e-08	7.9677e-09	6.7891e-08	5.2493e-08	5.9660e-08	-2.9653e-01	0:00:01	6
995	1.6194e-05	3.1030e-08	8.4083e-09	6.9275e-08	5.5631e-08	6.2854e-08	-2.9653e-01	0:00:01	5
996	1.6073e-05	3.0793e-08	8.3642e-09	6.7166e-08	5.3767e-08	6.1298e-08	-2.9653e-01	0:00:00	4
997	1.5850e-05	3.0882e-08	8.5830e-09	6.8585e-08	5.5740e-08	6.3292e-08	-2.9653e-01	0:00:00	3
998	1.5593e-05	3.0938e-08	8.5930e-09	7.0744e-08	5.2495e-08	6.2016e-08	-2.9653e-01	0:00:00	2
999	1.5495e-05	3.0908e-08	8.1438e-09	6.9529e-08	5.4462e-08	5.8477e-08	-2.9653e-01	0:00:00	1
1000	1.6613e-05	3.1522e-08	8.5176e-09	6.9018e-08	5.4065e-08	6.4178e-08	-2.9653e-01	0:00:00	0

Figure 17: Console iterations

3.2 Streamlines

Figures 18 and 19 show the streamlines simulation. The plots seem correct accordingly to the problem physics. It is possible to notice three regions where re-circulation flow shows up due to the adverse pressure gradient. One is right after the top inlet, one between the inlets and one next to the step of the outlet.

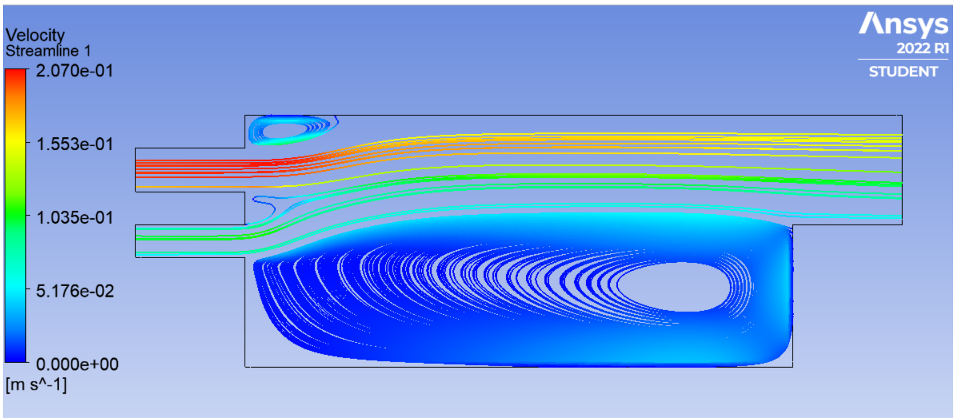


Figure 18: Streamlines - 38 lines configuration

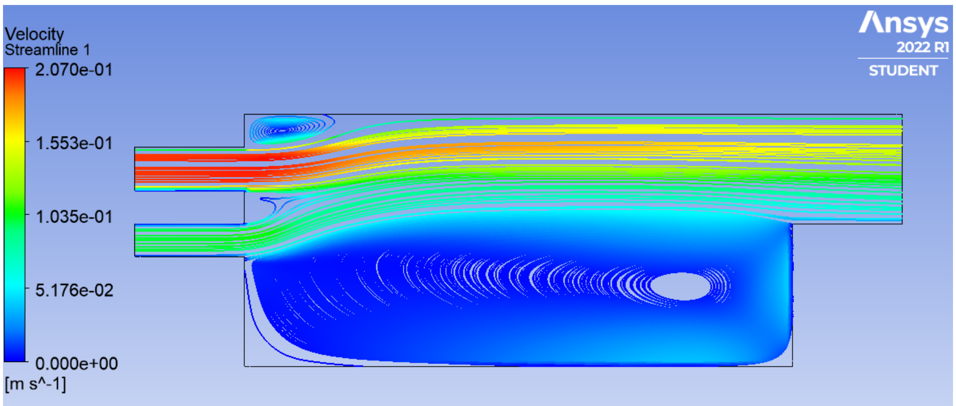


Figure 19: Streamlines - 100 lines configuration

3.3 Velocity Contour

Figures 20 to 22 show the velocity contour of the previous simulation. The plots seem correct accordingly to the problem physics.

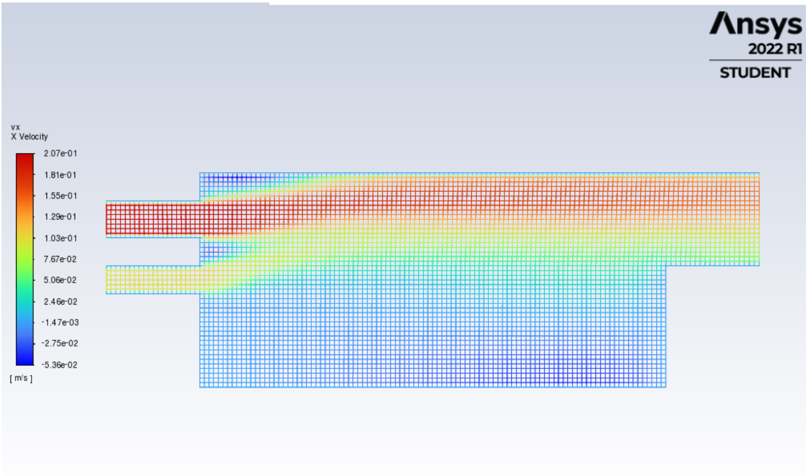


Figure 20: X Velocity contour

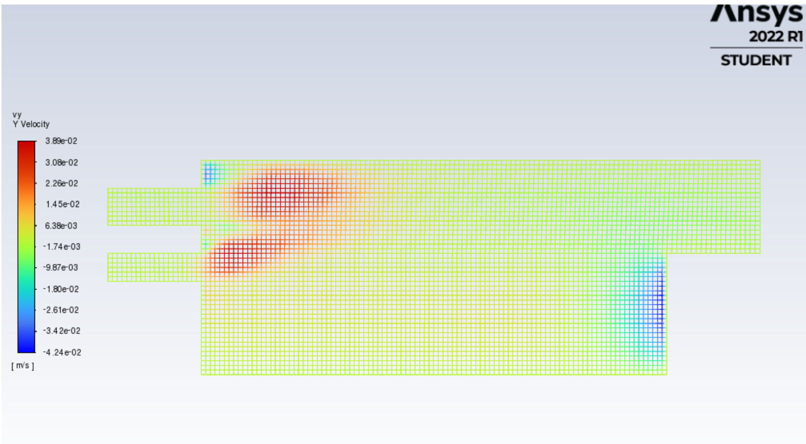


Figure 21: Y Velocity contour

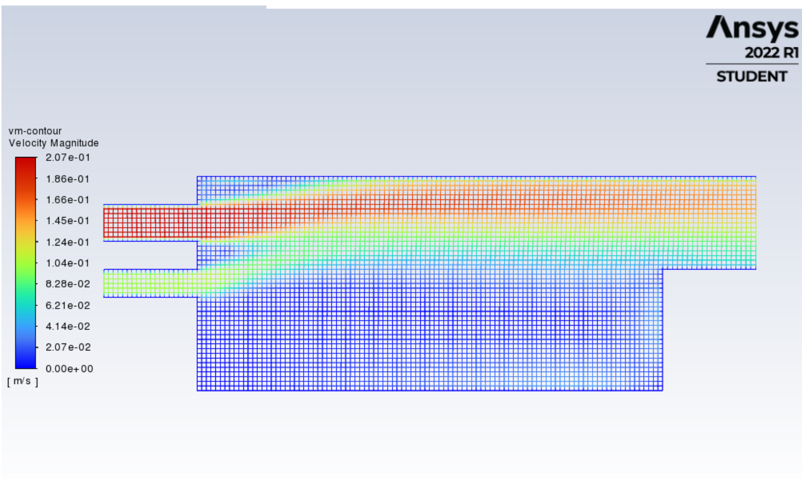


Figure 22: Magnitude Velocity contour

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) for both inlet's. Also, as expected, the top inlet show a higher entrance velocity than the bottom inlet.

3.4 Turbulence

Figures 23 to 24 show the turbulent aspects of the previous simulation. The plots seem correct accordingly to the problem physics.

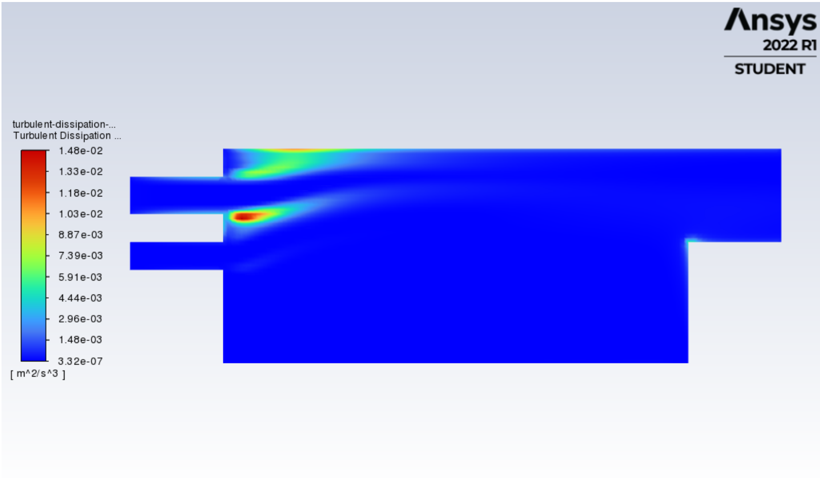


Figure 23: Turbulent dissipation contour

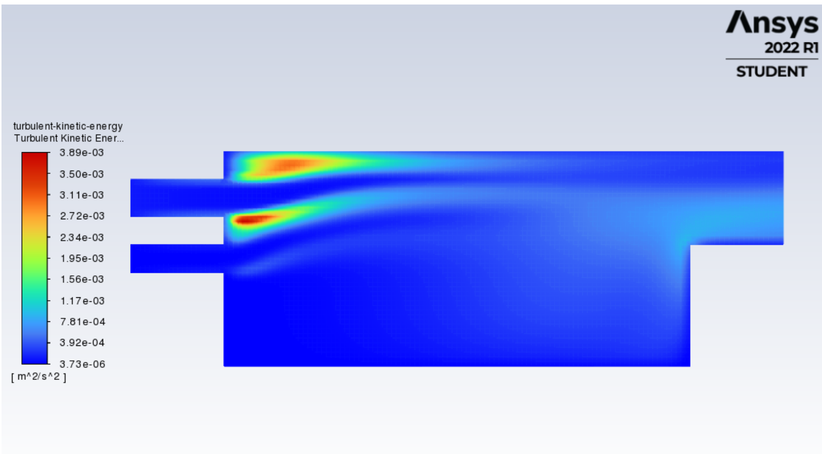


Figure 24: Turbulent kinetic contour

Since the turbulent dissipation is more concentrated between the inlet chambers, and also due to the difference in velocity and temperature, just above the top inlet chamber. Since at these points the velocity is visibly higher as seen from streamline plots, therefore, there is not a large rate of dissipation in the region where the lower vortex is concentrated (where there is also the lowest flow velocity). Similarly it is possible to state the same for the turbulent kinetic energy.

3.5 Pressure

Figures 25 to 27 show the pressure aspects of the previous simulation. The plots seem correct accordingly to the problem physics.

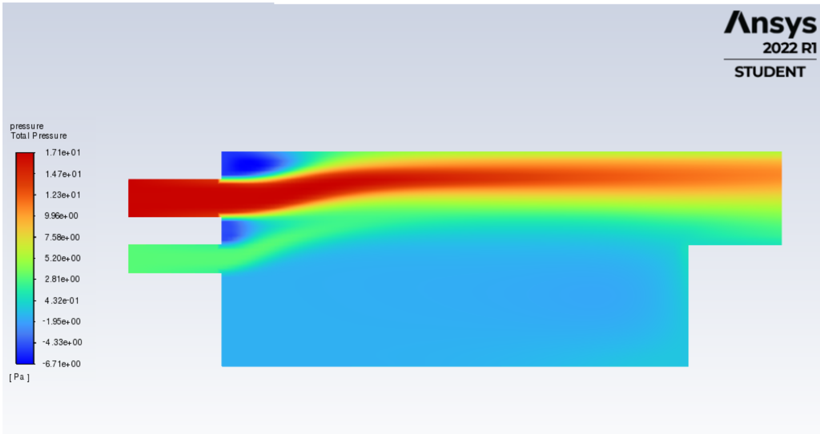


Figure 25: Total Pressure

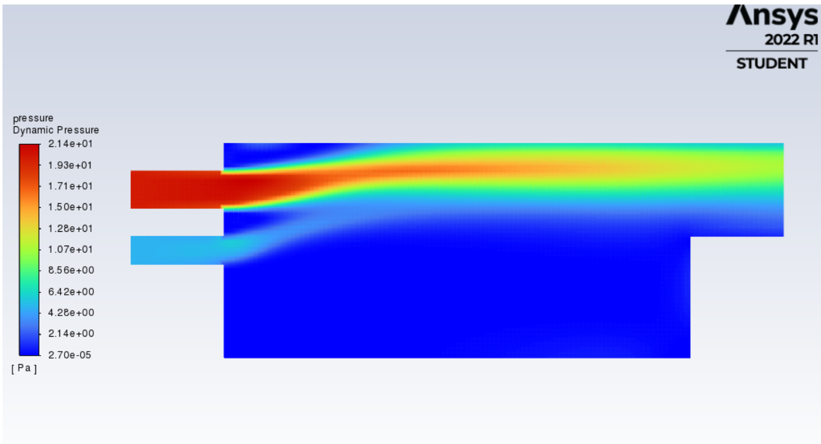


Figure 26: Dynamic Pressure

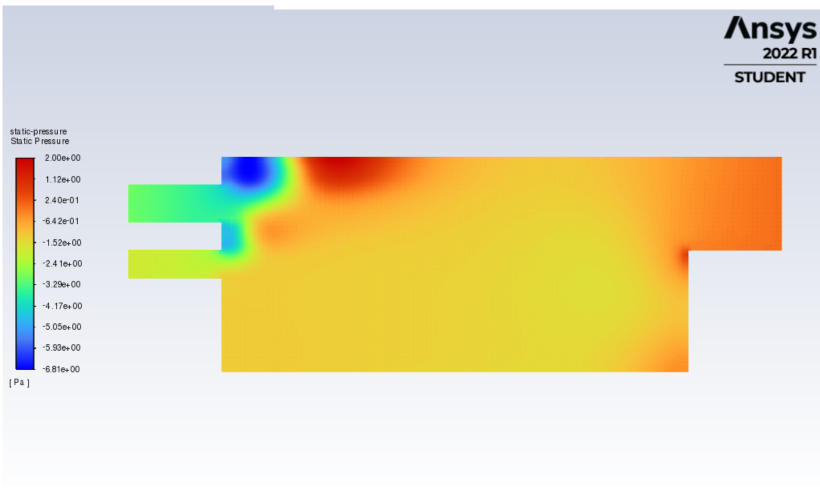


Figure 27: Static Pressure

Since the pressure is closely related to the fluid velocity, we have that such contours are in agreement with the streamline and velocity plots, after all, as it is possible to observe in the dynamic pressure plot, we have a correspondence to the vortex regions. Furthermore, it is possible to verify that the highest pressure gradient comes from the top inlet, since this inlet velocity is higher than the bottom one.

3.6 Temperature

Figure 28 shows the Temperature contour of the previous simulation. The plots seem correct accordingly to the problem physics.

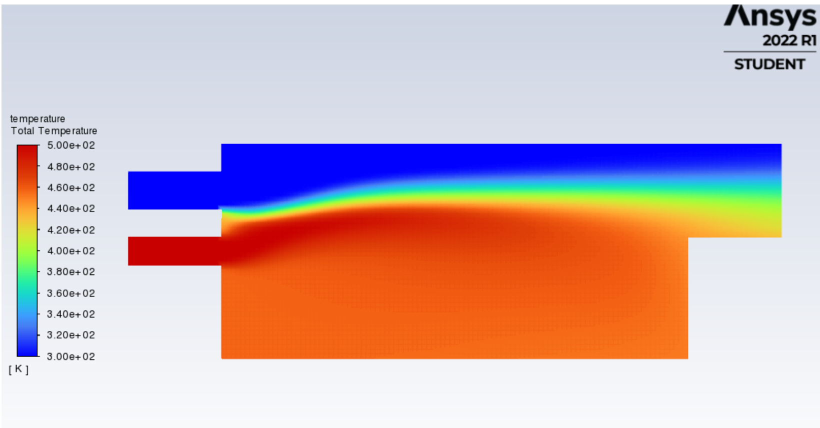


Figure 28: Total Temperature

It is possible to notice that:

Here is an interesting aspect, which is how the same fluid behaves in the midst of different inlet temperatures. There is an interesting behavior of natural convection that happens in this system: the bottom inlet chamber has an extremely higher temperature than the top inlet chamber. For this reason, it is seen that the fluid with the highest temperature tends to rise and therefore "pushes" the upper fluid with the lowest temperature. However, there is no emergence of a re-circulation since the fluid in the upper chamber is at a much higher velocity and therefore in the given geometry there is no space for it. Note that at the entrance the temperature distinction is much more pronounced than at the outlet, since there the colors that represent the system are much softer and there is a smoother temperature transition.