



**YEDİTEPE UNIVERSITY**

YEDİTEPE UNIVERSITY  
DEPARTMENT OF MECHANICAL ENGINEERING  
ME333 FLUID MECHANICS LABORATORY

## **CFD Analysis of a 2D Turbulent Flow Inside a 2D Pipe**

**Instructor:** Ali Bahadır Olcay

**Group Id:** 4G1

**Group Members:**

Ömer Dolaş	20200705001
İlker Kuzey Alkaşı	20210705076
Neşet Fatih Biçkin	20200705006
Artun Yağar	20200705055

**LAB DATE:** 08/11/2023

**DUE DATE:** 03/12/2023

## **Table of Contents**

<b>1)Abstract</b>	<b>3</b>
<b>2)Introduction</b>	<b>3</b>
<b>3)Materials and Methods</b>	<b>4</b>
<b>4)Results and Discussion</b>	<b>5</b>
<b>Additional Context:</b>	<b>8</b>
<b>A) Turbulence Model Comparison</b>	<b>8</b>
<b>B) Working Fluid Comparison</b>	<b>9</b>
<b>5)Discussion:</b>	<b>10</b>
<b>6)References</b>	<b>11</b>

# 1)Abstract

In this experiment, we examined how water flows in a stepped pipe. We focus on whether the flow is smooth or turbulent, and to understand this, we use different models to examine critical factors such as velocity/pressure contours, velocity vector and streamlines. When we calculated the Reynolds number, we realized that there was turbulent flow. We highlight the importance of mesh refinement in achieving precise numerical simulations, revealing notable differences in convergence, velocity distribution, pressure drop, and streamlines between coarse and fine mesh solutions. We also pay attention to how closely we look at the pipe walls in our simulation. As a result, we understand the importance of numerical precision in solving the complexities of fluid dynamics in confined geometries.

# 2)Introduction

This work focuses on the numerical analysis of a 2D turbulent flow inside a 2D pipe in the quest of a full knowledge of fluid mechanics within the field of computational fluid dynamics (CFD). This study digs into the complexities of fluid behavior, using ANSYS software to model and evaluate flow dynamics.

The fundamental goal of this analysis is to learn about the turbulent flow characteristics in a constrained pipe shape. The inquiry includes a thorough evaluation of critical factors such as velocity distribution, pressure fluctuations, and streamlines, with several mesh designs used for increased accuracy.

The numerical model is built using the ANSYS Standard Generate Mesh approach, with coarse and fine mesh solutions used to determine the influence on convergence and solution precision. The boundary conditions are depicted in Figure 1, with the entry as a velocity intake, the outlet as an opening to the atmosphere, and the walls as stiff with a no-slip boundary condition. The turbulence model used is k-omega, and the fluid used is air, which has features such as density and dynamic viscosity.

The Reynolds Number (Re), an important parameter in defining flow regimes, is calculated to be 20000, representing a turbulent flow regime. The CFD problem is initialized using the Standard Initialization Method, and solver parameters are adjusted for gradient, pressure, and momentum management.

In summary, our study of 2D turbulent flow inside a 2D pipe using CFD techniques revealed the critical importance of mesh improvement in obtaining accurate numerical simulations. The finer mesh appears as a critical feature in reproducing detailed flow phenomena inside confined geometries as the simulation results show differences in convergence, velocity distribution, pressure drop, and streamlines between coarse and fine mesh solutions. This sophisticated knowledge not only improves our understanding of turbulent flow features, but it also emphasizes the importance of precise numerical precision in unraveling the complexity of fluid dynamics.

### 3)Materials and Methods

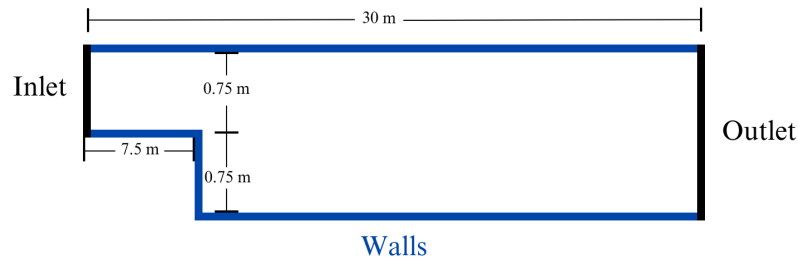


Figure 1: B.C. of numerical model

- The ANSYS Standard Generate Mesh technique broke down the fluid domain into 166 quadrilateral elements, showcased in Figure 2/A as the Coarse Mesh.
- The aspect ratio is maximum 1,338 and the skewness value is maximum 0,2773.
- The inlet conditions were assigned using a speed of 0.39 meters per second.
- The outlet was defined as an opening to the atmosphere, which set the gauge pressure to 0 pascals.
- The walls were considered rigid, and a no-slip boundary condition was enforced.
- The turbulence model is configured as k-omega sst.
- Gravity is configured at -9.81 meters per second in the y-direction.
- The working fluid is set as air, characterized by a density of 1.225 kilograms per cubic meter and a dynamic viscosity of 1.789e-5 pascal-seconds.
- The Reynolds Number (Re) was computed to be 20000 (from equation 1) which is greater than 4000.

$$Re = \frac{\rho u D}{\mu} \quad (1)$$

- Where:  
 $\rho$  is density,  
 $u$  is average velocity,  
 $D$  is Hydraulic Diameter of Pipe,  
 $\mu$  is dynamic viscosity.

- The CFD problem was initialized from the inlet using the Standard Initialization Method.
- The solver settings were adjusted to use "Least Squares Cell Based" for the gradient, "Second Order" for pressure, and "Second Order Upwind" for momentum handling.
- Residuals for continuity, x-velocity, and y-velocity were set at  $1e-5$ .
- The fluid domain has been divided into 14623 elements utilizing face sizing with minimum element size 100 and inflation mesh techniques as depicted in Figure 2/B. aiming for an estimated  $y^+$  value as 1, the first layer height is calculated as 1.2977-millimeter via online calculator [1].
- The max skewness value is 0,56127 and the max aspect ratio value is 89,817.
- These procedures were employed to achieve a more precise solution (Fine Mesh).

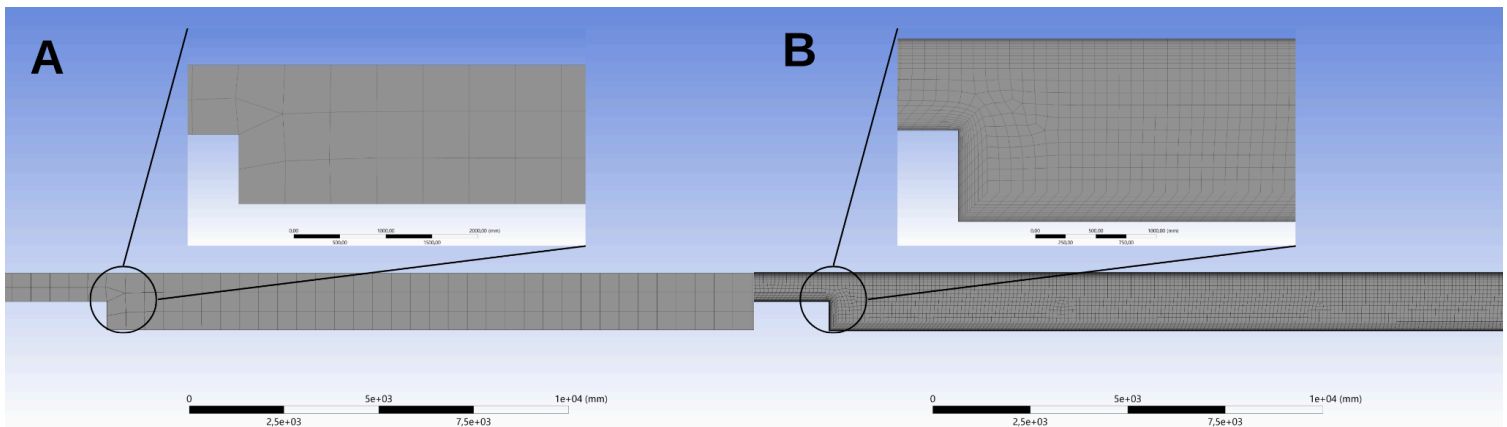


Figure 2: Coarse Mesh and Fine Mesh Solutions.

## 4) Results and Discussion

The solution using the coarse mesh converged by the 565th iteration, whereas the fine mesh solution converged by the 667th iteration, as shown in Figure 3

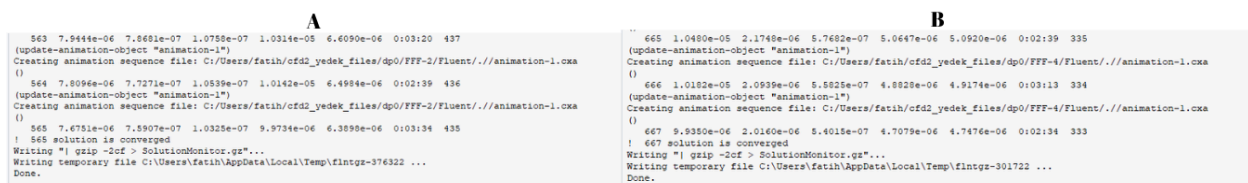


Figure 3: Iteration numbers of Coarse Mesh and Fine Mesh solutions.

As the fluid enters the pipe at a speed of 0.39 m/s, it reaches a maximum speed of 0.43 m/s before reaching the step region. In the time it reaches the step region, there is a vortex happens and the speed instantly decreases to zero. After a while the speed reached the mid value which was 0.21 m/s then it continues in that way to the output. In the fine mesh (Figure 4-5(B)), the vortex is visible in the specified region, whereas it's not distinctly noticeable in the coarse mesh (Figure 4-5(A)).

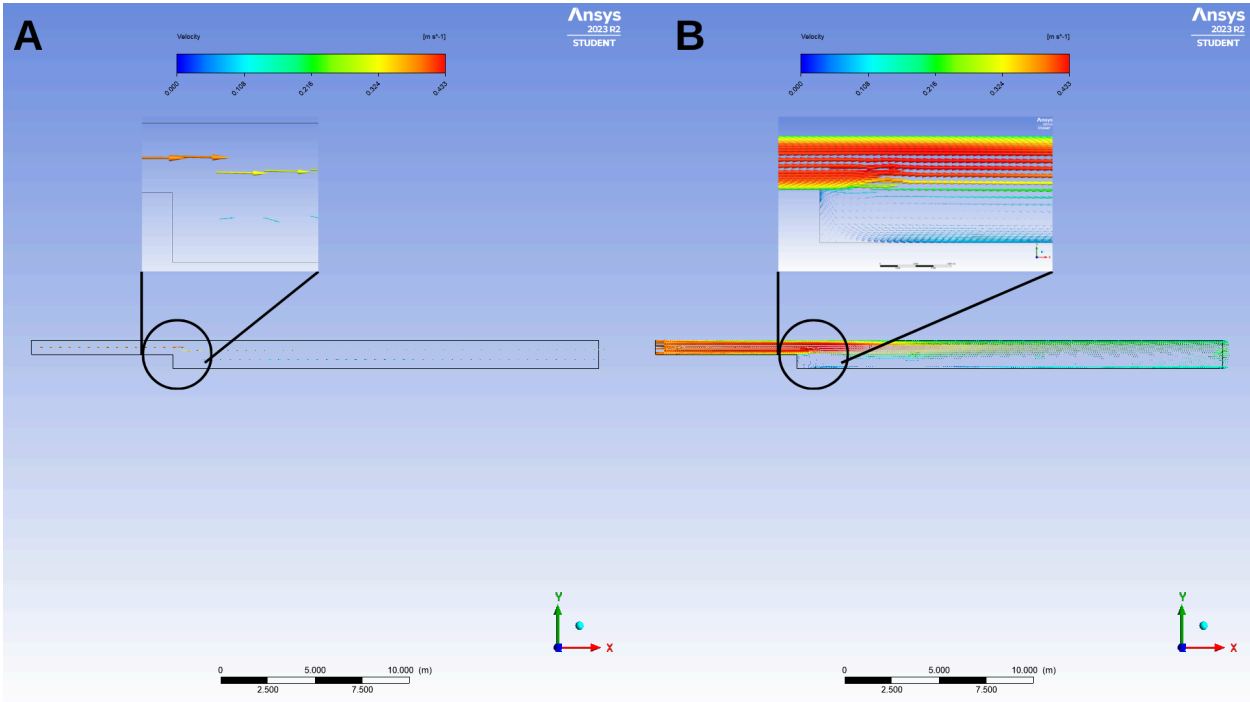


Figure 4: Velocity vector diagrams for Coarse and Fine Mesh solutions.

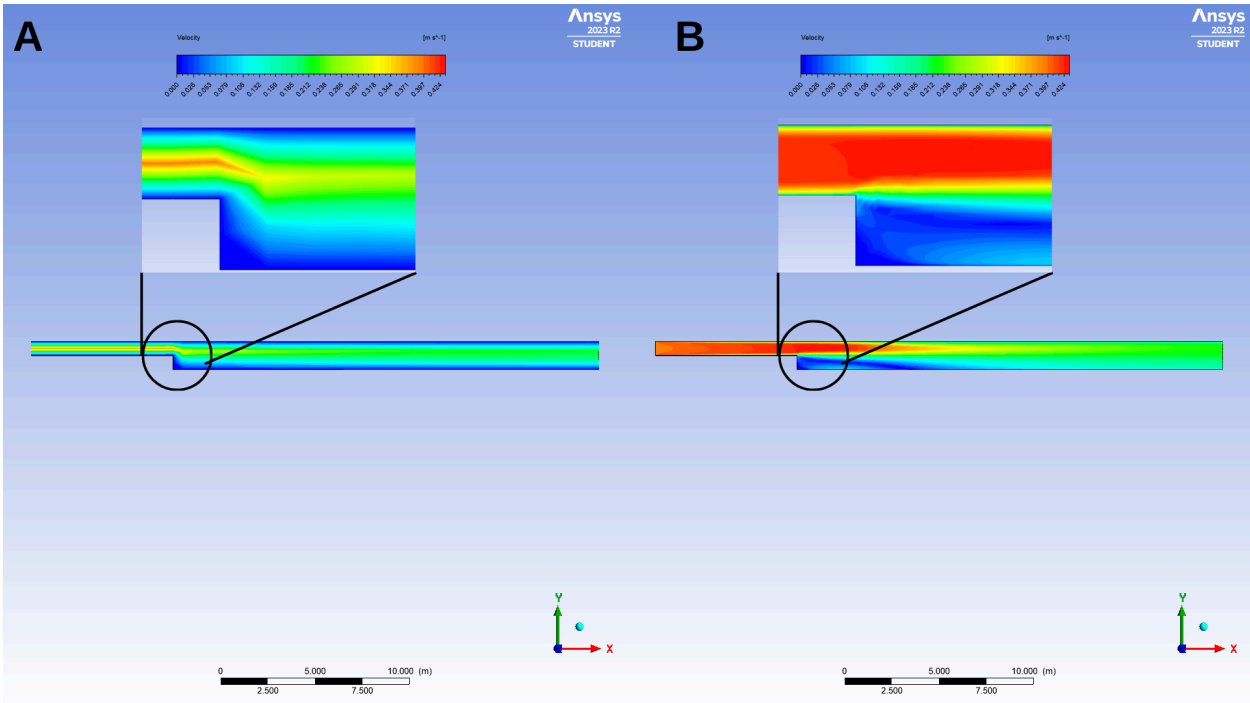


Figure 5: Velocity contour of Coarse and Fine Mesh Solutions.

kl

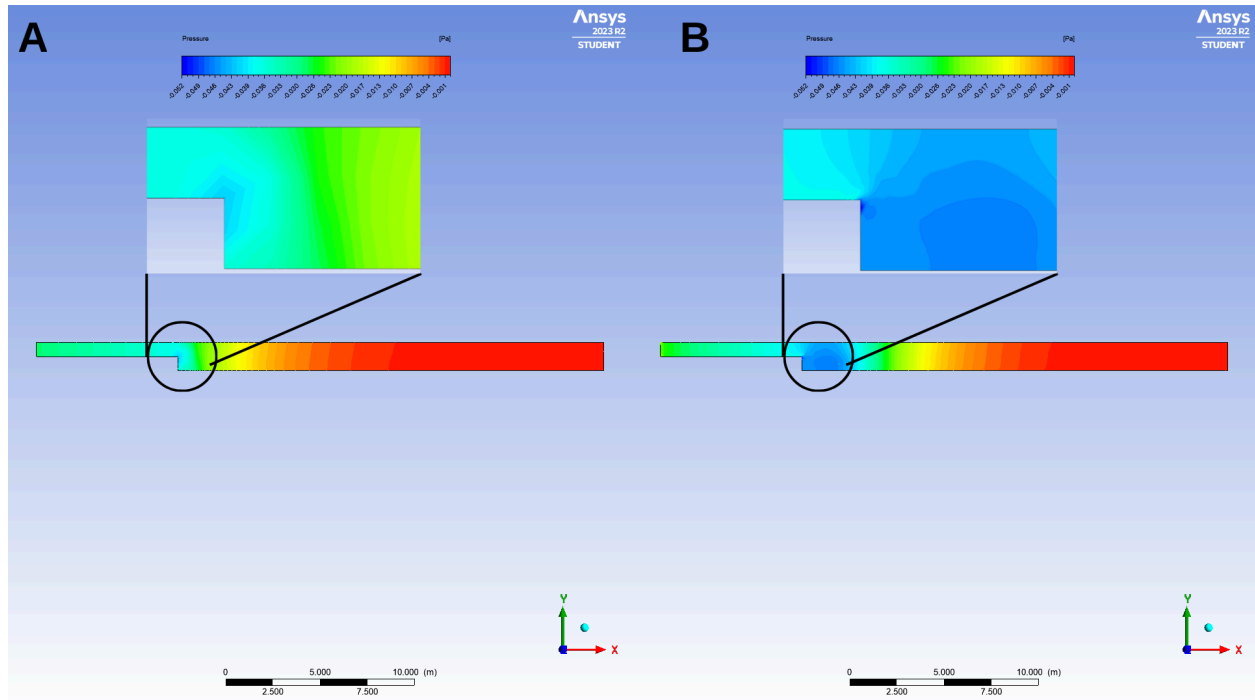


Figure 6: Pressure drop contour of Coarse and Fine Mesh Solutions.

Figure 7 shows the streamlines in both coarse and fine mesh solutions. The vortex can be clearly seen in the fine mesh solution, however, in the coarse mesh solution it is not possible to see it.

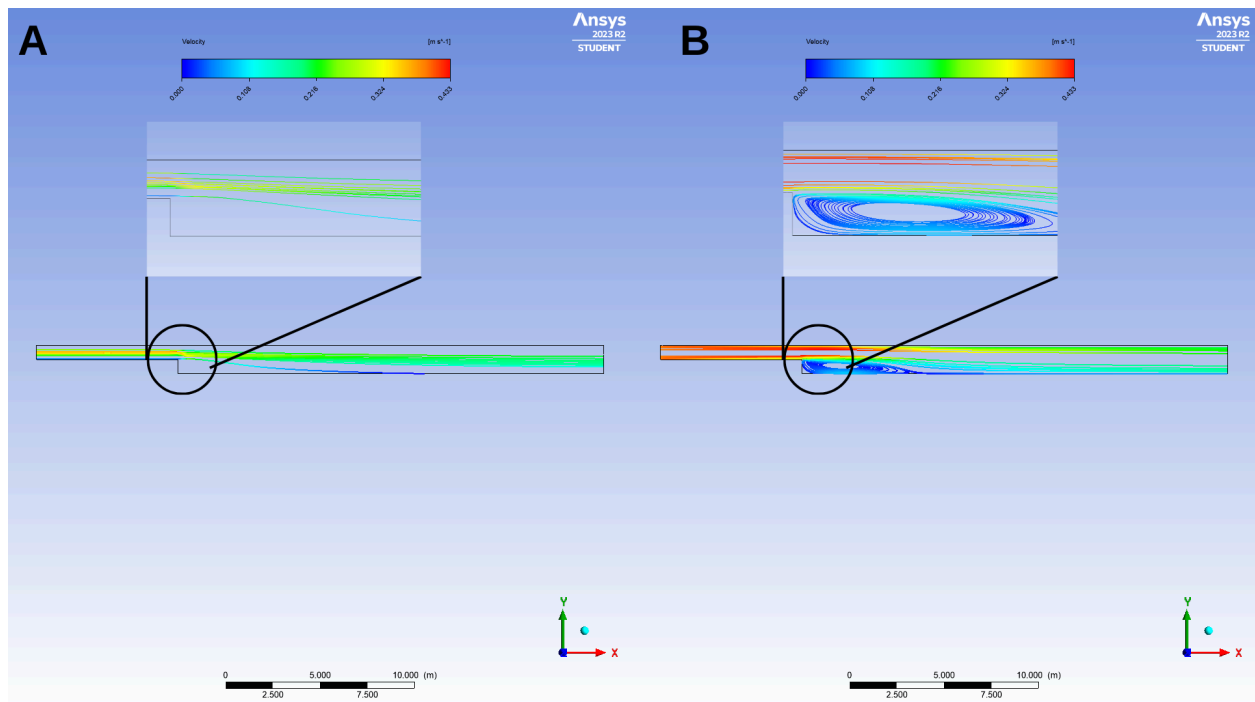


Figure 7: Streamlines of Coarse and Fine Mesh Solutions.

## Additional Context:

### A) Turbulence Model Comparison

In this part, the experiment was done with the k-epsilon sst turbulence method to observe the difference between the effect of the two methods on back-step flow. In each figure below, A is for the k-omega fine meshed and B is for the k-epsilon fine meshed solution.

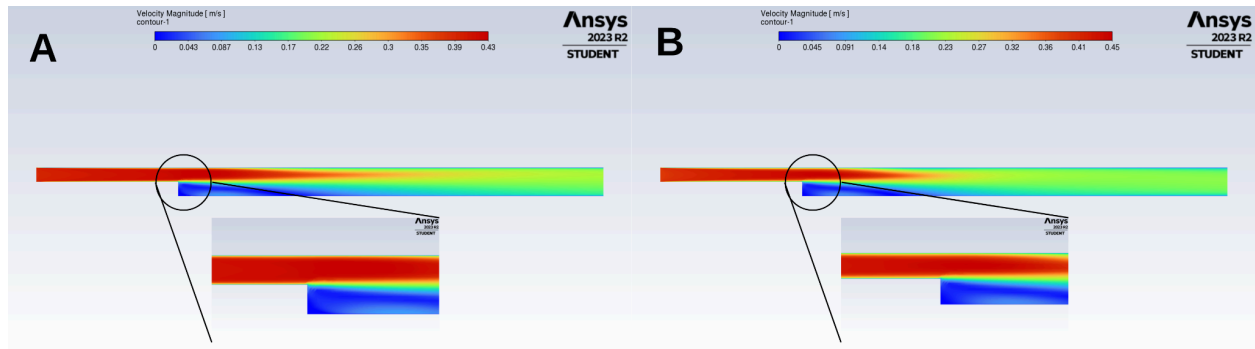


Figure 8: Velocity contour of k-omega and k-epsilon methods

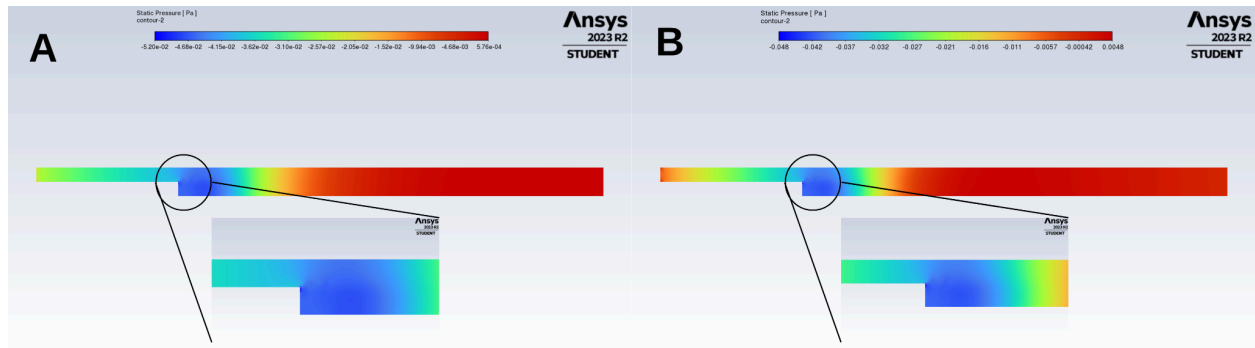


Figure 9: Pressure contour of k-omega and k-epsilon methods

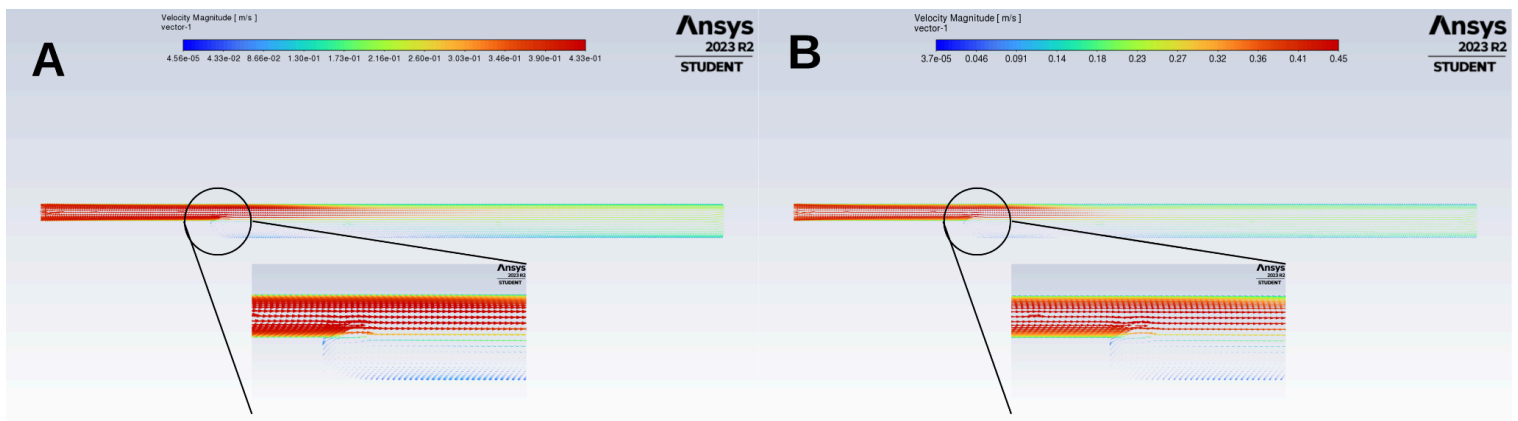


Figure 10: Velocity vector diagram of k-omega and k-epsilon methods



## B) Working Fluid Comparison

In this part, the experiment was done with different working fluids to observe the difference between the effect of different working fluids on back-step flow. The working fluid is set as water, characterized by a density of 998.2 kilograms per cubic meter and a dynamic viscosity of  $1003 \times 10^{-3}$  pascal-seconds.

The fluid domain has been divided into 168558 elements utilizing face sizing and inflation mesh techniques as depicted in Figure 11 aiming for an estimated  $y^+$  value of 1, the first layer height is calculated as 0.124956 millimeters via an online calculator [1]. These procedures were employed to achieve a more precise solution (Fine Mesh).

The aspect ratio is a maximum 187,18 and the skewness value is a maximum 0,65456

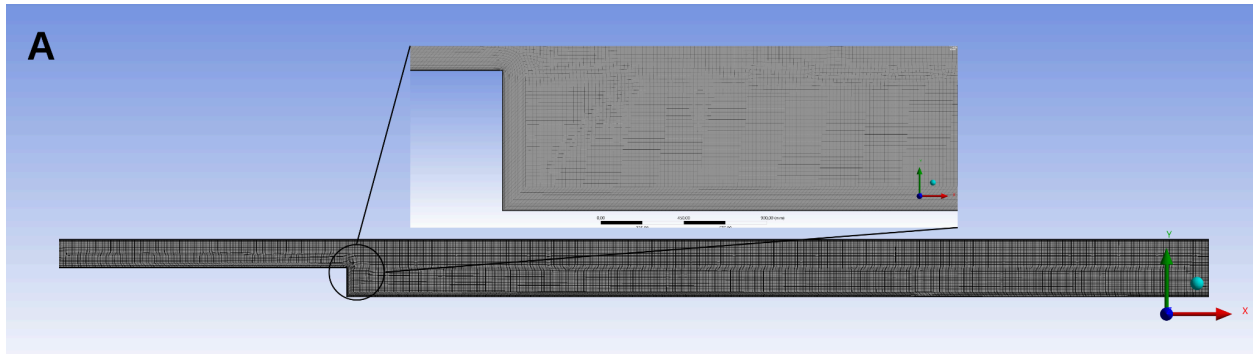


Figure 11: Water Working Fluid Fine Mesh Solution

In each figure below, A is for the Air fine meshed and B is for the Water fine meshed solution.

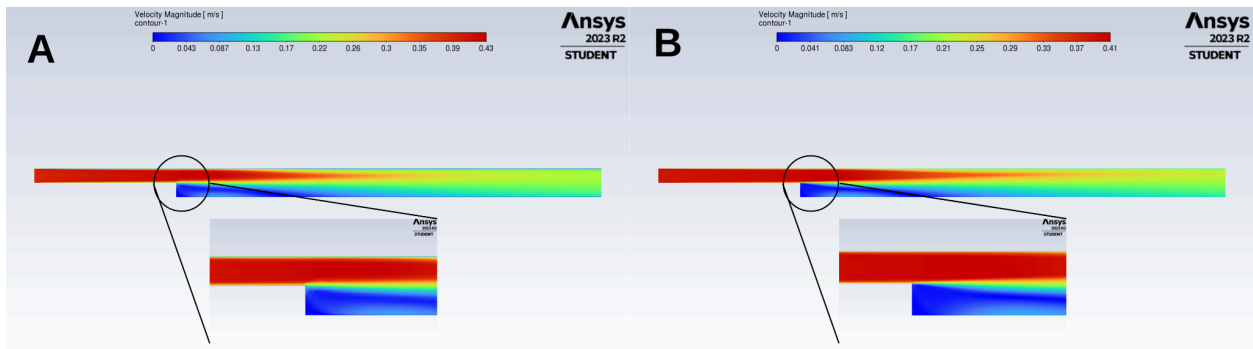


Figure 12: Velocity contour of air and water working fluids

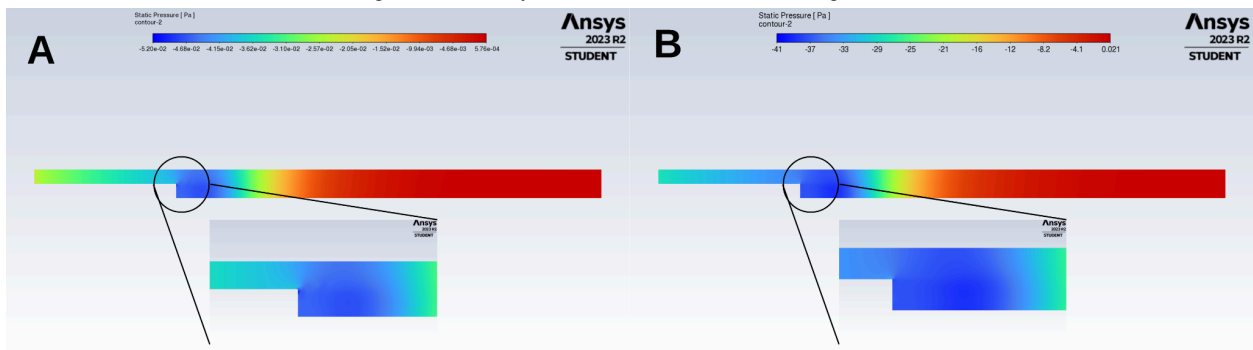


Figure 13: Pressure contour of air and water working fluids

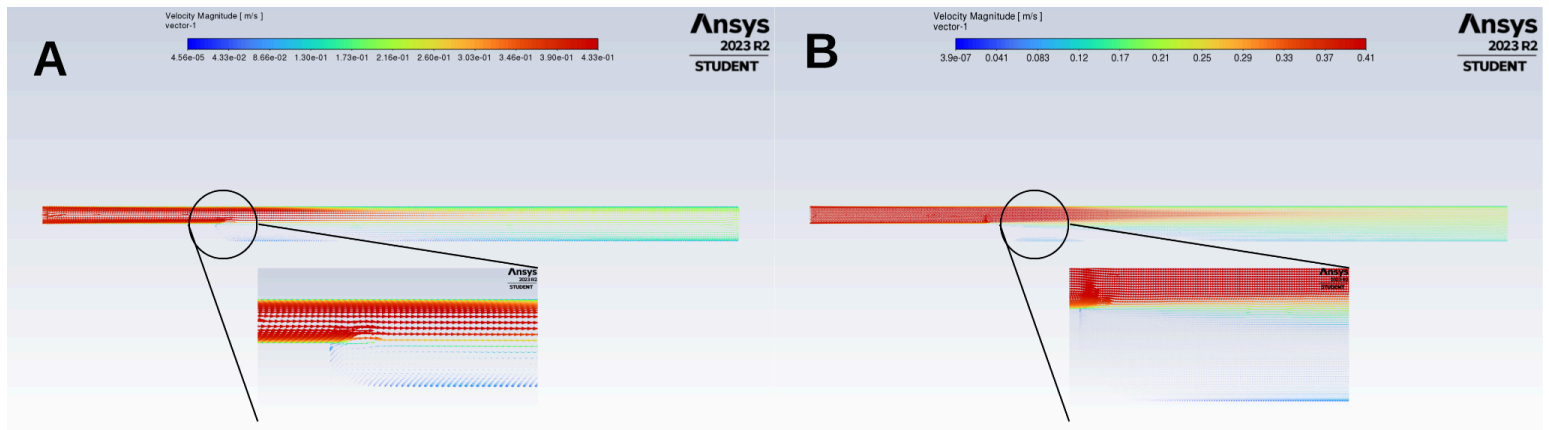


Figure 14: Velocity vector diagram of air and water working fluids

## 5)Discussion:

If there is a step region on the flow path, back-step flow occurs in this region and the flow separation occurs. An area that cannot be used in this region is formed and flow continuity is lost in this area and flow discontinuity exists. We can define this as a Vortex and we need to observe it in a well-defined analysis.

Y+ value refers to the boundary layer condition for the mesh we use for our geometry. It is a parameter that measures the boundary layer value. For an estimated  $y^+$  value of 1, The first layer height is calculated as 1.2977-Millimeter.

Two types of mesh called Coarse and Fine were used. Fine Mesh produced more detailed and more sensitive results. Coarse Mesh could not give good results in local points in the characteristics of the flow, while the general trends have caught well.

When Coarse Mesh is used, the formation of a vortex in the step region can be observed in a very difficult way and the characteristic features of this region cannot be observed in the speed/pressure contour graphics and speed vector graphs. ( $y^+$  value is not used because Coarse Mesh describes the situation that is not close to real)

In the case of the use of Fine Mesh, the formation of a vortex in the step region can be observed and the characteristic features of this region can be observed in the speed/pressure contour graphics and speed vector graph in reality. For example, sudden pressure losses occur in the pressure contour in the Vortex region, and back-step flow occurs as can be seen in the speed vector graph.

In summary, a good analysis can be made by observing the situations that should be by obtaining close to real results in the solution made using a Fine Mesh where the parameters are well adjusted.

When the fluid material was selected as water, a new first layer height was calculated for the Fine Mesh part and when a new mesh was created with this value, the highest Aspect Ratio value was calculated as 980. In order to correct this, we changed the minimum element size of the face sizing method to 20 and obtained a 187 Maximum Aspect Ratio value (which is better

than 900, we did not reduce the size of the element more due to computer limitation and did not change the mesh method because a triangular mesh was not used in the air material).

When the working fluid was air, the boundary layer value was 1.2977 mm, and when it is set as water, it is 0.124956. Thus, the first layer height of air is greater than water.

For the both working fluids, the pressure distribution shows similar behavior in both simulations, and the pressure value reaches the maximum value in the outlet section and it reaches the minimum in the step region.

When K-epsilon and K-Omega pressure contours are compared (Figure 9), the lowest pressure value for Step-Region is overlooked when K-epsilon is used. Because K-epsilon is not ideal for the use of No-Slip Condition. For this reason, the use of a K-omega is more suitable for achieving the right results for the back-step flow situation.

## 6)References

[1] <https://embed.plnkr.co/IWnRIQ8HCvnD4if901F9>