YEDİTEPE UNIVERSITY
DEPARTMENT OF MECHANICAL ENGINEERING
ME333 FLUID MECHANICS LABORATORY

CFD Analysis of a 2D Laminar Pipe Flow in Fluent

Instructor: Ali Bahadır Olcay
Group Id: 4G1
Group Members:

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

LAB DATE: 25/10/2023
DUE DATE: 12/11/2023

Table of Contents

1)Abstract

In this experiment, we examined the flow in a pipe with a length of 8 meters and a diameter of 0.2 meters with the help of Ansys. The first thing we do is draw the geometry of the pipe in 2D and define our surfaces. (Inlet, Outlet, PipeWall, CenterLine)Then, we apply the requested mesh values to our system. Finally, we entered the numerical values and ran our system. As a result, we were able to examine the movements of velocity, pressure, and viscosity in the pipe and plot them on graphs.

2)Introduction

Fluid mechanics is a branch of mechanical engineering that studies the behavior of fluids. Fluid flow is critical in the design and analysis of engineering systems that span pipelines to aircraft. Computational Fluid Dynamics (CFD) has become an effective tool for simulating and analyzing fluid flows in this context, providing insights that would be difficult to obtain through experimental means alone.

The focus of this experiment is the CFD analysis of a 2D laminar pipe flow. The laminar flow is a flow regime defined by smooth and orderly fluid motion at relatively low velocity. Analysis we have conducted is by using ANSYS Fluent which is a widely used CFD software. In Fluent we used some programs that construct the experiment, those are; SpaceClaim which is the pre-processing stage of Fluent involves tasks such as defining the geometry and generating the grid. In this phase, we defined the pipe's geometry (0.2X8 m pipe in the 2D model). Meshing is the process of creating a computational grid suitable for the defined geometry, this step is crucial for a solution. For example, we did two meshes, the first one had 8000 cells, and 1000 for the second mesh. By doing this we analyzed that the more meshes the more accurate the solution. The next step is the solution setup. In this step, we customized the flow conditions such as the type of the flow, fluids properties, pressures, and velocity. We obtained contours, graphs, mesh solutions, vectorial representation of velocity on pipe, etc.
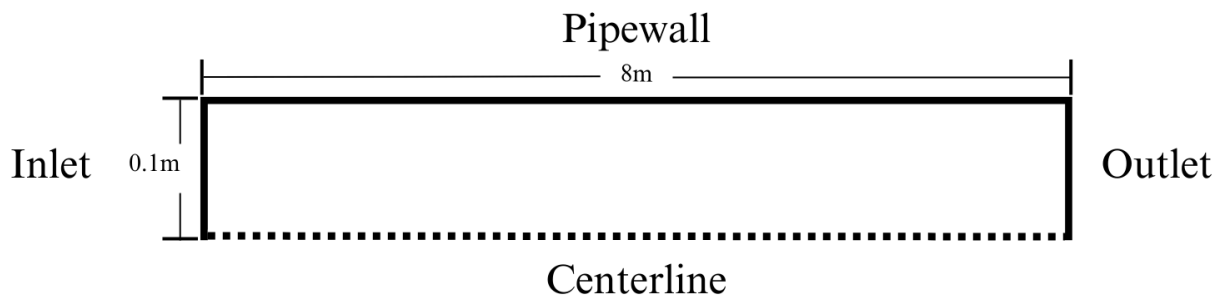
3)Materials and Methods

Figure 1: B.C. of numerical model

- The fluid domain has been segmented into 1000 quadrilateral elements using the sizing mesh method, as illustrated in Figure 2/A (Coarse Mesh).
- Inlet boundary conditions were specified as a velocity inlet with a uniform velocity magnitude of 1 m/s.
- The outlet was designated as an opening to the atmosphere, resulting in the relative pressure being set at 0 Pa.
- The pipewall was treated as rigid, and a no-slip boundary condition was applied.
- Recognizing the symmetry, the bottom line of the fluid domain was identified as the centerline.
- The working fluid was created randomly, with a density of 1 kg/m^3 and a dynamic viscosity of 2e-3 Pa. The average velocity was set at 1 m/s, and the hydraulic diameter of the pipe was designated as 0.2 m.

- Re (Reynolds Number) was calculated as 100 (from equation 1)

$$Re = \frac{\rho u D}{\mu}$$

(1)

- Where:
  ρ is density,
  u is average velocity,

D is Hydraulic Diameter of Pipe,
μ is dynamic viscosity.

- The CFD problem was addressed using a COUPLED solution method. The gradient was configured to "Least Squares Cell Based," while pressure was specified as "Second Order," and the momentum was handled using the "Second Order Upwind" method.
- Residuals for continuity, x-velocity, and y-velocity were established at 1e-6.
- The solution was solved as steady.
- The fluid domain has been segmented into 8000 quadrilateral elements using the sizing mesh method, as illustrated in Figure 2/B. The steps above were used to solve for a more accurate solution (Fine Mesh).
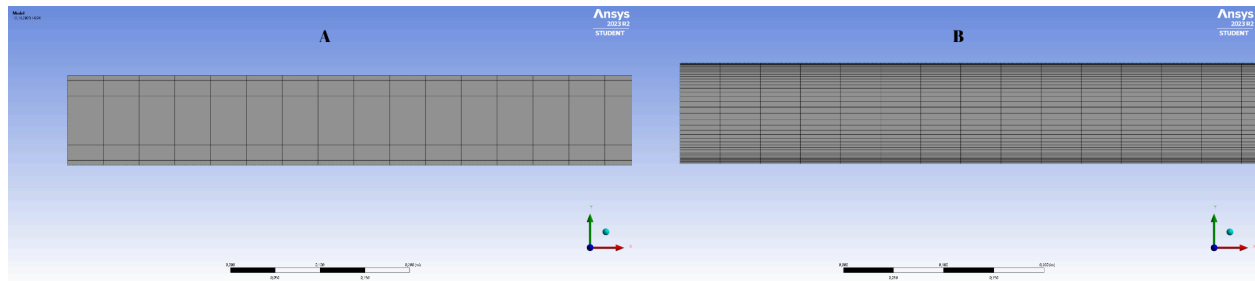

Figure 2: Coarse Mesh and Fine Mesh Solutions.

4)Results and Discussion

The coarse-meshed solution converged at the 63rd iteration, while the fine-meshed solution converged at the 44th iteration (Figure 3).

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

The velocity reaches 1.71 m/s with a coarse mesh, while it reaches approximately 2.0 m/s with a fine mesh (Figure 4).
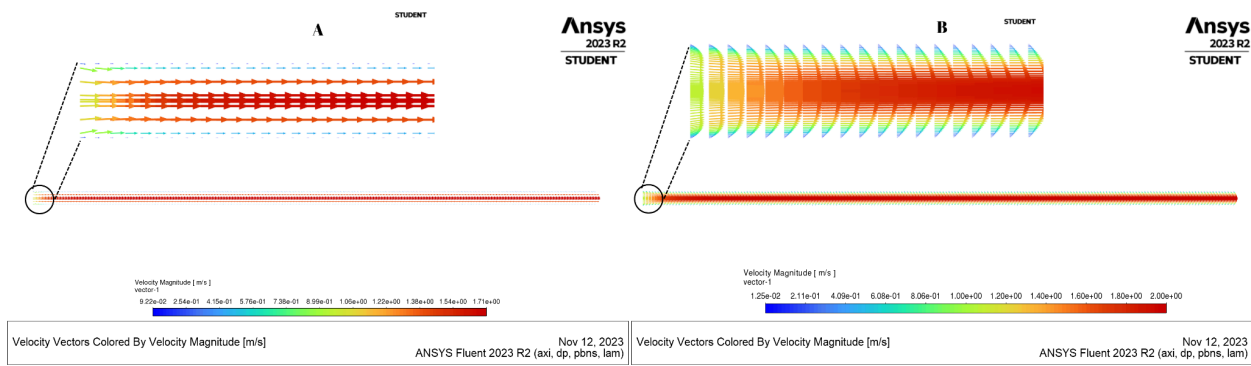
Figure 4: Velocity vector diagrams for Coarse and Fine Mesh solutions.

The coarse-meshed configuration attains full development at approximately 1.2 m from the inlet, whereas the fine-meshed one achieves full development at a distance of 1.8 m from the inlet.
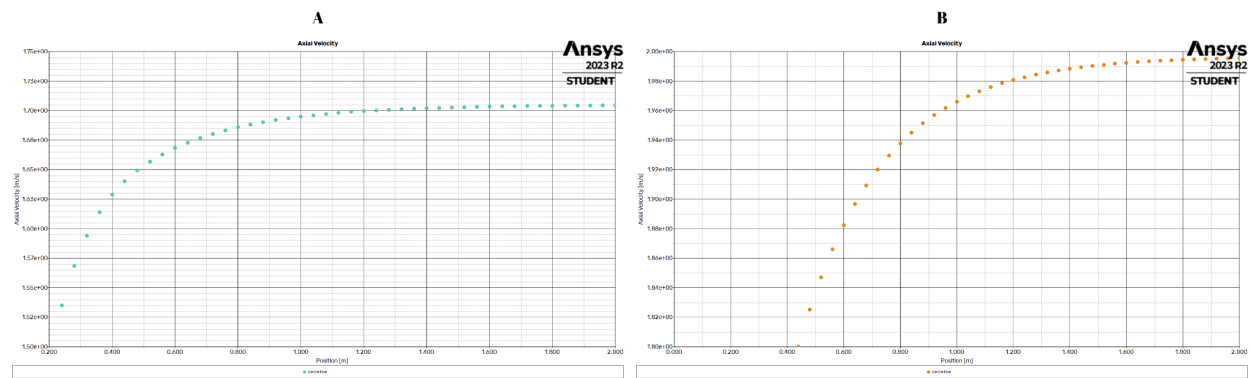

Figure 5: Centerline axial velocity of Coarse and Fine Mesh Solutions.

The pressure drop is 11.6 Pa with a coarse mesh, whereas it increases to 13.7 Pa with a fine mesh (refer to Figure 6).
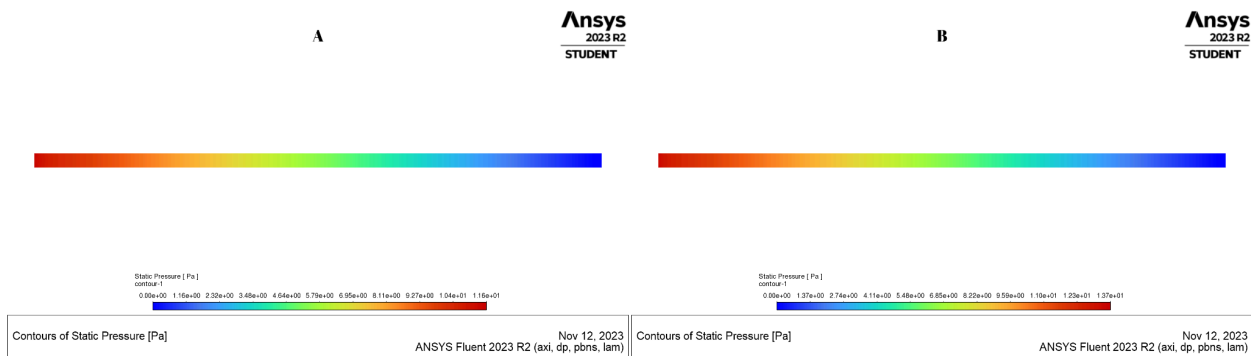

Figure 6: Pressure drop contour of Coarse and Fine Mesh Solutions.

Discussion:

Laminar flow analysis was performed in a long pipe (very long compared to its length), while two different types of mesh namely Coarse and Fine were applied to the geometry. In addition to Coarse, the Fine Mesh aims to better describe the flow simulation in the Ansys Fluent and to simulate it closer to real life. As the number of elements in the mesh increased, a simulation closer to real life was obtained. As a result, the relevant values converge more consistently to the desired range (Speed 2.0 m/s). The results obtained were by the entered data parameters, according to the randomly generated liquid, the Reynolds Number was 100 and the flow was compatible with laminar.

As seen in Figure 3, calculations could be made with fewer iterations when Fine Mesh was used. A calculation was made logically without tiring the computer.

As seen in Figure 4, when Fine Mesh was used, the results were as expected. The flow rate is slower in the parts close to the wall and higher in the middle part of the pipe. In the case of using Coarse Mesh, a similar result was obtained, but a result close to real life was not obtained here.

As seen in Figure 6, the pressure started to decrease slowly after reaching a certain value. There is no difference in behavior other than values between Fine Mesh and Coarse Mesh. The pressure has the lowest value at the exit point.

5)Conclusion

In this report, laminar flow movement in a pipe was examined using Ansys fluent. Our main purpose here is to analyze and report how the laminar flow in the pipe is, the speed changes, pressure changes, and temperature changes by using the basic principles of fluid mechanics thanks to the Ansys Fluent program. Pipe design, mesh, and installation part were done in Ansys, the flow rate, pressure changes were examined, then the number of meshes increased from 1000 to 8000 and laminar flow analysis was performed again. The result was analyzed again and these results were compared. Some results were obtained. First of all, the Reynolds number was calculated as 100 since the Reynolds number is less than 2300, the flow in the pipe is compatible with laminar flow. Velocities in a section within the pipe are slower near the wall and faster toward the center of the pipe. The pressure in the pipe initially increases until it reaches a certain value and then decreases in a near-linear behavior. In addition, as the number of meshes increases, the Ansys Fluent program measures the solutions more precisely.

In addition, when performing laminar flow analysis, it is important to increase the number of meshes to get better results, and choosing the features that define the flow correctly in the setup section will greatly affect the analysis results. In this regard, it is necessary to learn computer-supported software (Ansys Fluent) well.