



AGH UNIVERSITY OF KRAKOW

Numerical Simulation of Turbulent Flow in a Circular Pipe

CFD Modeling in Energy Engineering

Author:
Szymon Kuczek

Contents

1	Objective and Scope	2
2	Problem Statement	2
3	Analytical Solution	2
3.1	Reynolds Number	2
3.2	Friction Factor and Pressure Drop	2
4	Numerical Model	3
4.1	Discretization (Meshing)	3
4.2	Solver Settings	4
5	Simulation Results	4
5.1	Solution Convergence	4
5.2	y^+ Parameter Verification	5
5.3	Velocity and Pressure Fields	5
6	Conclusions and Validation	6

1 Objective and Scope

The objective of this report is the numerical analysis (CFD) of turbulent air flow in a straight pipe with a length of $L = 2$ m and a diameter of $D = 4$ mm. The simulation was conducted in the **Ansys Fluent 2025** environment using the $k - \omega$ SST turbulence model. The numerical results (pressure drop) were verified against an analytical solution based on the Darcy-Weisbach equation.

2 Problem Statement

The analysis concerns the steady-state flow of a compressible fluid (air) in a smooth pipe. The geometric parameters and physical properties of the fluid are presented in Table 1.

Geometry Parameter	Value	Fluid Property	Value
Pipe Length (L)	2.0 m	Density (ρ)	1.225 kg/m ³
Diameter (D)	0.004 m	Dyn. Viscosity (μ)	$1.7894 \cdot 10^{-5}$ Pa · s
Inlet Velocity (v)	50 m/s	Outlet Pressure	0 Pa

Table 1: Simulation input data

3 Analytical Solution

Theoretical calculations were performed to verify the correctness of the numerical model.

3.1 Reynolds Number

The flow regime was determined based on the Reynolds number:

$$Re = \frac{\rho \cdot v \cdot D}{\mu} = \frac{1.225 \cdot 50 \cdot 0.004}{1.7894 \cdot 10^{-5}} \approx \mathbf{13\,692} \quad (1)$$

A value of $Re > 4000$ clearly indicates **turbulent** flow.

3.2 Friction Factor and Pressure Drop

For hydraulically smooth pipes in the range of $4000 < Re < 10^5$, the Blasius formula was applied:

$$\lambda = \frac{0.3164}{\sqrt[4]{Re}} = \frac{0.3164}{\sqrt[4]{13\,692}} \approx 0.02927 \quad (2)$$

The expected pressure drop was calculated using the Darcy-Weisbach equation:

$$\Delta P_{analit} = \lambda \cdot \frac{L}{D} \cdot \frac{\rho v^2}{2} = 0.02927 \cdot \frac{2.0}{0.004} \cdot 1531.25 \approx \mathbf{22\,394.32} \text{ Pa} \quad (3)$$

4 Numerical Model

4.1 Discretization (Meshing)

The **MultiZone** method was used to discretize the computational domain, allowing for the generation of a high-quality hybrid mesh. The method was configured with *Mapped/Swept Type: Hexa* and *Free Mesh Type: Tetra/Pyramid*, ensuring the dominance of hexahedral elements while maintaining mesh flexibility.

To accurately represent the circular cross-section of the pipe, an **Edge Sizing** operation with 50 divisions was applied to the inlet and outlet edges. A key element was also the application of a boundary layer (*Inflation*), which is essential for meeting the requirements of the turbulence model.

Note: The mesh generation process was optimized regarding the limitations of the academic license (*Ansys Student Version*), which caps the number of elements/nodes. Thanks to the hybrid method, high resolution was achieved in key zones (near-wall) while keeping the total element count within the allowable limit without compromising solution accuracy.

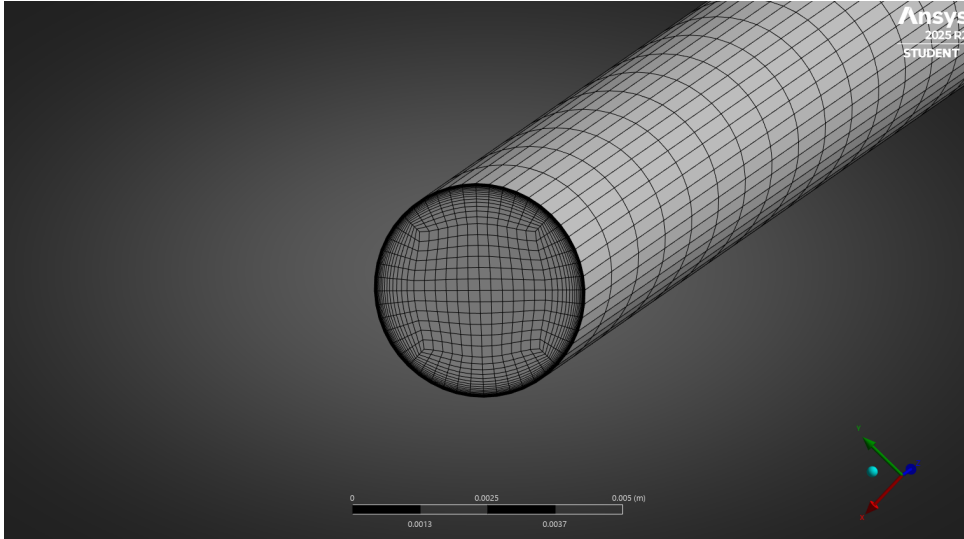


Figure 1: Computational mesh generated using the MultiZone method with visible inflation layer.

Mesh parameters:

- Method: MultiZone (Hexa Core / Tetra-Pyramid)
- First layer height at wall: $2.4 \cdot 10^{-6}$ m (for $y^+ \approx 1$)
- Element count: **963 792** (High-density mesh)

4.2 Solver Settings

The simulation was performed using a *Pressure-Based* solver in steady state.

- **Turbulence Model:** $k - \omega$ SST (Shear Stress Transport) – selected for accuracy in the near-wall region.
- **Scheme:** Coupled (pressure-velocity coupling).
- **Discretization:** Second Order Upwind for all variables.

5 Simulation Results

5.1 Solution Convergence

The computational process was stable. Monitored residuals, including the continuity residual, reached values below 10^{-3} , indicating the correct balancing of transport equations.

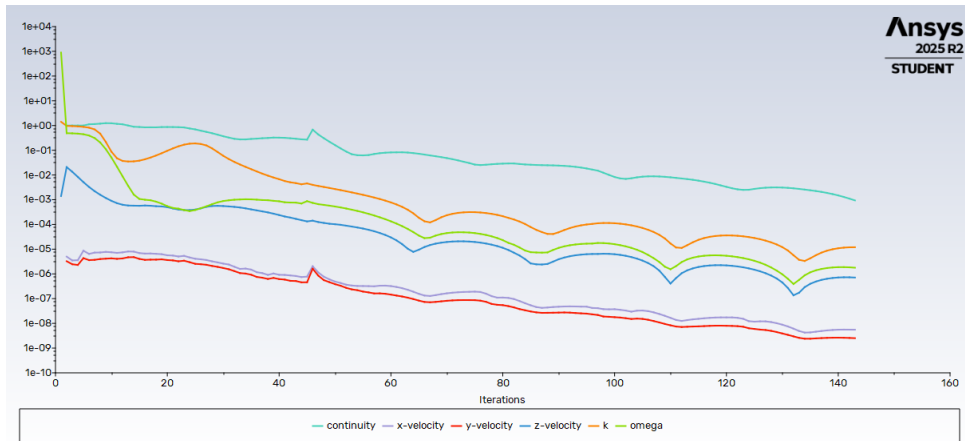


Figure 2: Residual convergence history.

5.2 y^+ Parameter Verification

The correctness of the boundary layer construction was verified by analyzing the dimensionless wall distance (y^+). A maximum value of 0.68 was obtained, which meets the rigorous criterion of $y^+ < 1$ for the SST model and guarantees the correct calculation of viscous friction.

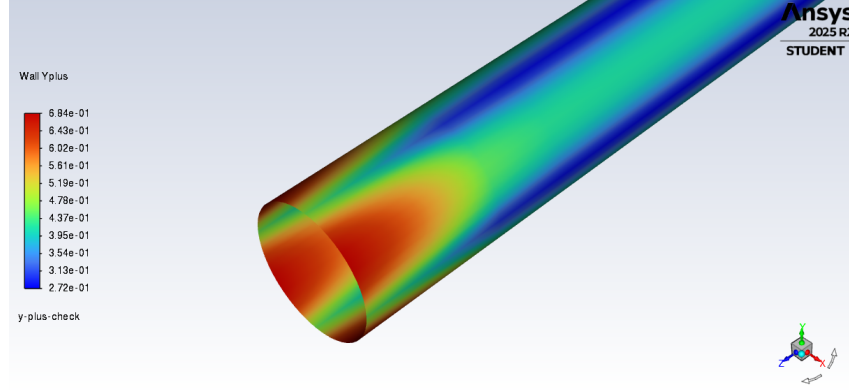
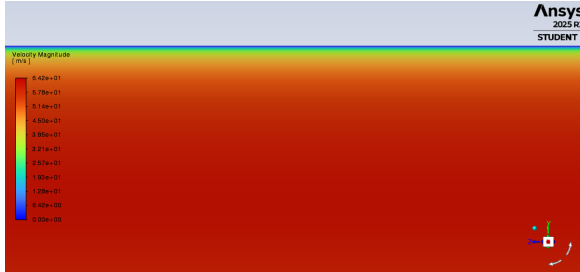


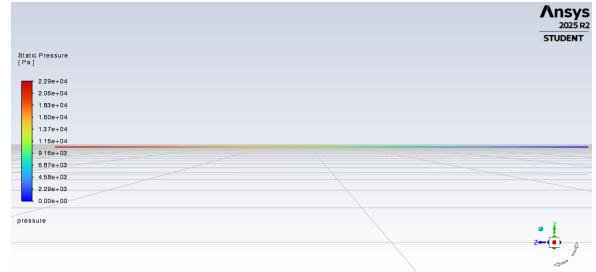
Figure 3: Contour of the y^+ parameter on the pipe wall.

5.3 Velocity and Pressure Fields

The obtained flow fields exhibit characteristics typical of developed turbulence. The velocity profile is flattened along the pipe axis, with high gradients occurring near the walls.



(a) Velocity [m/s]



(b) Pressure [Pa]

Figure 4: Field distributions inside the pipe (symmetry plane).

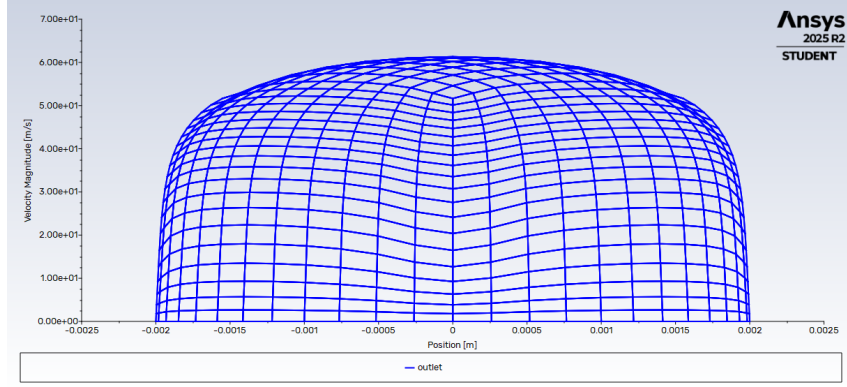


Figure 5: Velocity profile at the pipe outlet.

6 Conclusions and Validation

The total pressure drop in the CFD simulation was determined as the difference of area-weighted average static pressures at the inlet and outlet.

Method	Pressure Drop ΔP [Pa]	Relative Error
Analytical	22 394.32	-
Numerical (CFD)	22 891.83	2.22%

Table 2: Comparison of analytical and numerical results

Summary:

1. The numerical model showed very high agreement with theory (error in the order of 2%).
2. The higher result of the numerical simulation (22 891 Pa) compared to the analytical one (22 394 Pa) results from including the **hydrodynamic entrance region** in the CFD model. In this zone, the developing velocity profile generates higher local shear stresses, which the simplified Darcy-Weisbach formula does not account for.
3. The use of a dense mesh (nearly 1 million elements) and precise control of discretization parameters (Inflation, Edge Sizing) allowed for the elimination of numerical errors associated with mesh coarseness.