

CFD Simulation of Airflow Around a Circular Cylinder

Nuraida Syakirah Adnan¹, Ishkrizat Taib^{1*},

1. Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, MALAYSIA

CORRESPONDENCE

Ishkrizat Taib

iszat@uthm.edu.my

Received: 05 July 2026

Revised : 10 August 2026

Accepted : 30 August 2026

Published: 30 September 2026

This study presents a numerical investigation of fluid flow characteristics around a circular cylinder using ANSYS Fluent software. The main objective is to analyse the effects of different inlet velocities on pressure distribution, flow structure, and velocity field around the body. The cylinder is modelled with a diameter of 5 mm and an extrusion length of 30 mm, placed at the centre of a rectangular computational domain with a uniform distance of 30 mm from all outer boundaries. The numerical simulation employs the automatic meshing method with tetrahedral elements, where the element size ranges from 12.0 mm to 2.0 mm. A grid independence test is conducted to verify accuracy, and a mesh size of 2.0 mm is selected for all final analyses to ensure high resolution and consistency. The SST $k-\omega$ turbulence model is applied together with the finite volume method to solve the governing equations of fluid flow, while boundary conditions are defined as velocity inlet, pressure outlet, wall, and pipe surfaces accordingly. Simulations are performed at three different inlet velocities: 10 m/s, 50 m/s, and 80 m/s, to observe changes in flow behaviour as speed increases. The results show that velocity magnitude has a significant influence on the overall flow physics. At 10 m/s, the pressure distribution is mostly uniform with only a small high-pressure zone at the front stagnation point, while streamlines follow the surface smoothly with no separation. As velocity rises to 50 m/s and further to 80 m/s, the pressure difference between the front and rear surfaces becomes much larger, creating a stronger drag force acting on the cylinder. The streamline patterns indicate wider deflection, earlier separation, and a broader wake region behind the body at higher speeds. Velocity contours reveal that the flow accelerates strongly along the curved sides, reaching a maximum value of 91.21 m/s at 80 m/s, accompanied by steep gradients near the surface and a large low-velocity area downstream. All results maintain symmetrical distribution, confirming stable and balanced flow conditions throughout the study. In conclusion, this work successfully demonstrates how increasing velocity directly leads to greater pressure variation, larger flow disturbance, and more intense acceleration around the cylinder. The numerical setup used is proven reliable and capable of producing physically meaningful data, providing a clear understanding of flow characteristics that can support engineering design and analysis involving cylindrical structures.

Keywords: Circular Cylinder, Flow Characteristics, Pressure Distribution, Velocity Contour, Streamline Pattern

1. INTRODUCTION

Flow around cylindrical bodies is a fundamental topic in fluid mechanics, relevant to engineering systems such as heat exchangers, offshore structures, cooling towers, and food processing equipment [3,7,12]. Key phenomena include boundary layer development, separation, and periodic vortex shedding, which induce fluctuating forces, vibration, and noise that affect structural safety and efficiency [1,9,19].

Kumar et al. (2014) numerically investigated natural convection around a cylinder for condenser applications, showing that low-Reynolds number turbulence models provide better accuracy than standard formulations, and established reliable heat transfer correlations [1]. For high Reynolds number flows, Catalano et al. (2003) found that Large Eddy Simulation predicts drag and pressure distribution more accurately than RANS, which tends to underestimate loads and overpredict wake size [2]. Kondjoyan and Boisson (1997) confirmed that near-wall treatment strongly affects results, with low-Reynolds formulations matching experiments better [17]. Apacoglu et al. (2011) used reduced-order modelling to show that flow can be represented by few dominant modes, and that active control reduces drag effectively [5]. Naito and Fukagata (2012) demonstrated that porous surfaces suppress fluctuations and can eliminate vortex shedding entirely at high speeds [6]. These studies highlight the need for accurate modelling and clear physical understanding.

Geometry and surface conditions significantly alter performance. Mallick et al. (2015) reported that roughness and diameter change separation points and pressure distribution [7]. Khademinezhad et al. (2015) found that lift coefficient varies strongly with shape and Reynolds number, while drag and Strouhal number remain relatively constant [11]. Isaev et al. (2019) noted that standard SST models underpredict loads by up to 30%, whereas modified versions with curvature correction agree well with measurements [14]. For multiple cylinders, Hubova et al. (2019) showed that interference depends on spacing, roughness, and direction, often producing higher forces than isolated cases [4]. Rahimi et al. (2020) and Sokolov and Virk (2023) confirmed complex wake interactions in side-by-side and duplex arrangements, where results are highly sensitive to turbulence model choice [3,21].

Thermal behaviour was studied extensively by Olsson and co-workers. For single impinging jets, they developed correlations showing heat transfer increases with velocity and curvature [8,10]. Multiple jets provide better enhancement, though three-jet systems create non-uniform distributions [12]. Numerical results generally follow experimental trends but differ in magnitude, especially in the wake region [10]. Xu et al. (2024) showed that condensation increases heat transfer and that velocity adjustment improves uniformity by modifying vortex development [16]. Csernyei and Straatman (2016) found jet array performance depends on diameter ratio; smaller jets benefit from increased spacing, while larger jets perform worse with offset [20].

Various control methods have been validated. Gao et al. (2017) tested slotted cylinders, finding they delay separation, narrow the wake, and reduce fluctuations [19]. Xu et al. (2020) showed that passive jet holes shift vortex formation downstream and reduce forces significantly [9]. Firat et al. (2025) optimised auxiliary airfoils, achieving 14.4% lower drag and over 70% reduction in fluctuations [15]. For functional applications, Lei et al. (2022) determined an optimal spacing of 2.5 for energy harvesting using piezoelectric films [13]. Bari and Saad (2013) designed guide vanes to enhance turbulence and mixing in engine intake systems with minimal loss [18]. Together, these works guide the development of efficient and reliable engineering designs.

1.1. Related Theory

Flow around cylindrical structures is governed by key dimensionless parameters that define the flow regime, forces, and heat transfer behaviour [1,17].

The Reynolds number (Re) represents the ratio of inertial forces to viscous forces, determining whether the flow is laminar, transitional, or turbulent. It is defined as

$$Re = \frac{\rho U D}{\mu} = \frac{U D}{\nu} \quad \text{Equation 1}$$

where ρ is fluid density (kg/m^3), U is free-stream velocity (m/s), D is cylinder diameter (m), μ is dynamic viscosity ($\text{Pa}\cdot\text{s}$) and ν is kinematic viscosity (m^2/s).

For buoyancy-driven flows such as natural convection, the Rayleigh number (Ra) becomes the dominant parameter, characterising the strength of thermal effects relative to diffusion [1],

$$Ra = \frac{g\beta\Delta TD^3}{\nu\alpha} \quad \text{Equation 2}$$

where g is gravitational acceleration (m/s^2), β is thermal expansion coefficient ($1/\text{K}$), ΔT is the temperature difference between the surface and fluid (K), and α is thermal diffusivity (m^2/s).

When vortices are shed periodically from the cylinder, the frequency of this phenomenon is expressed by the **Strouhal number** (St), a critical parameter for predicting vibration and noise [1,11],

$$St = \frac{f_s D}{U} \quad \text{Equation 3}$$

where f_s is vortex shedding frequency (Hz).

Aerodynamic performance is quantified using force coefficients. The drag coefficient (CD) (force parallel to flow) and lift coefficient (CL) (force perpendicular to flow) are defined as [6,15],

$$C_D = \frac{F_D}{\frac{1}{2}\rho U^2 DL} \quad \text{Equation 4}$$

$$C_L = \frac{F_L}{\frac{1}{2}\rho U^2 DL} \quad \text{Equation 5}$$

where F_D and F_L is total drag and lift forces (N) and L is length of the cylinder (m).

For thermal analysis, convective heat transfer is represented by the Nusselt number (Nu), which relates the convective coefficient to fluid thermal conductivity [1,8,16],

$$Nu = \frac{hD}{k} \quad \text{Equation 6}$$

where h is the convective heat transfer coefficient ($\text{W/m}^2\text{K}$), k is fluid thermal conductivity (W/mK).

In forced convection, Nu is typically correlated with Re and Prandtl number (Pr), which describes the ratio of momentum diffusivity to thermal diffusivity.

$$Pr = \frac{\nu}{\alpha} \quad \text{Equation 7}$$

Numerical modelling generally solves the Reynolds-Averaged Navier–Stokes (RANS) equations, closed by turbulence models such as $k-\epsilon$, $k-\omega$, or SST. For higher accuracy, Large Eddy Simulation (LES) resolves large turbulent structures directly [2,9]. Corrections for near-wall behaviour, streamline curvature, or anisotropy are often required to properly predict separation and heat transfer [1,14,17].

In multi-cylinder arrangements, interference effects alter flow patterns and force magnitudes depending on spacing and configuration [3,4,21]. Control methods modify surface, geometry, or momentum to change separation location, reduce wake width, or disrupt vortex synchronisation, thereby lowering loads and improving performance [6,9,19].

2. METHODOLOGY

This chapter outlines the numerical methods and procedures adopted to investigate the flow and heat transfer characteristics around circular cylinders using ANSYS software. It details the geometry definition, boundary conditions, mesh generation, grid independence study, and solution strategy employed throughout the simulation work.

2.1. Geometry Description Figure

The computational domain is modelled exactly according to the dimensions defined in the DesignModeler module. The main solid body is a horizontal circular cylinder with a diameter of 5 mm and an extrusion length of 30 mm, representing the spanwise length of the model.

The cylinder is positioned at the centre of a rectangular enclosure, which defines the fluid flow domain. The enclosure is constructed with a uniform distance of 30 mm measured from the cylinder surface to each outer boundary, on all sides including inlet, outlet, top, bottom, and lateral directions. This creates a symmetrical flow field with equal clearance around the cylinder, ensuring that boundary effects are consistent from all directions.

With these dimensions, the total size of the fluid domain is 65 mm in length, 65 mm in height, and 30 mm in width. The geometry is modelled as a smooth, rigid, and impermeable wall. The cylinder and fluid domain are created as separate bodies, and Boolean operation is performed to subtract the cylinder volume from the enclosure, leaving only the fluid region for meshing and simulation. This setup is maintained consistently for all simulation cases to ensure comparability of results.

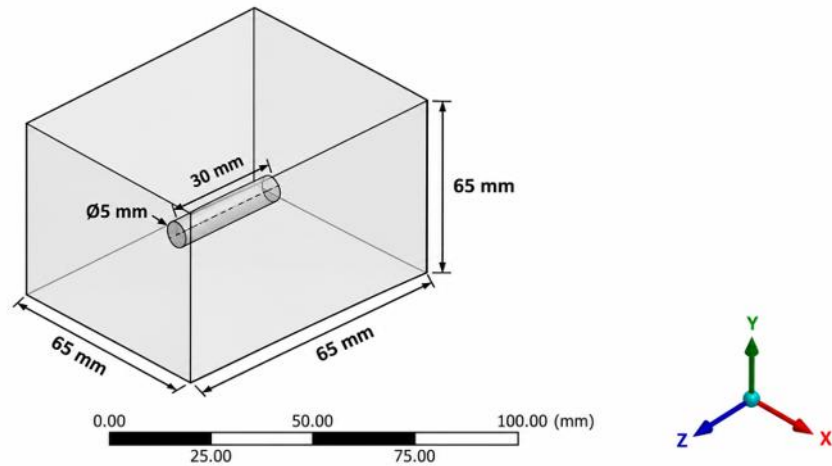


Figure 1. The dimensions used for this model

2.2. Boundary Condition

Boundary conditions are assigned to all surfaces of the computational domain exactly as defined in the ANSYS Fluent setup, consisting of four main types: Inlet, Outlet, Wall, and Pipe. Detailed specifications for each boundary are as follows:

2.2.1. Inlet

Located at the front face of the enclosure, defined as Velocity Inlet type. A constant uniform velocity value is specified according to the targeted Reynolds number. For thermal analysis, a constant temperature is defined at this boundary. Turbulence parameters are set using turbulence intensity and hydraulic diameter, with intensity maintained between 1% and 5% to represent low turbulence free-stream conditions.

2.2.2. Outlet

Located at the rear face of the enclosure, defined as Pressure Outlet type. Static pressure is set to zero gauge pressure, representing atmospheric exit conditions. All flow variables are set to have zero normal gradient, allowing flow to exit smoothly without any reverse flow or reflection effects that could disturb the upstream flow field.

2.2.3. Wall

Applied to all outer boundaries of the enclosure including top, bottom, and side faces. These surfaces are defined as stationary, no-slip walls with zero velocity in all directions. Thermal conditions are set as adiabatic, meaning no heat flux passes through these boundaries, assuming they are sufficiently far from the cylinder and do not influence the thermal behaviour of the flow.

2.2.4. Pipe

Applied to the entire surface of the circular cylinder. This boundary is defined as a stationary Wall representing the solid pipe surface. No-slip condition is enforced, ensuring fluid velocity at the surface equals zero. For heat transfer studies, either constant wall temperature or constant heat flux is assigned to this surface, depending on the case being simulated. Surface roughness is set to zero, representing a smooth pipe surface as modelled in the geometry.

These settings define the physical limits of the flow domain and ensure that the simulation represents the actual conditions intended for this study.

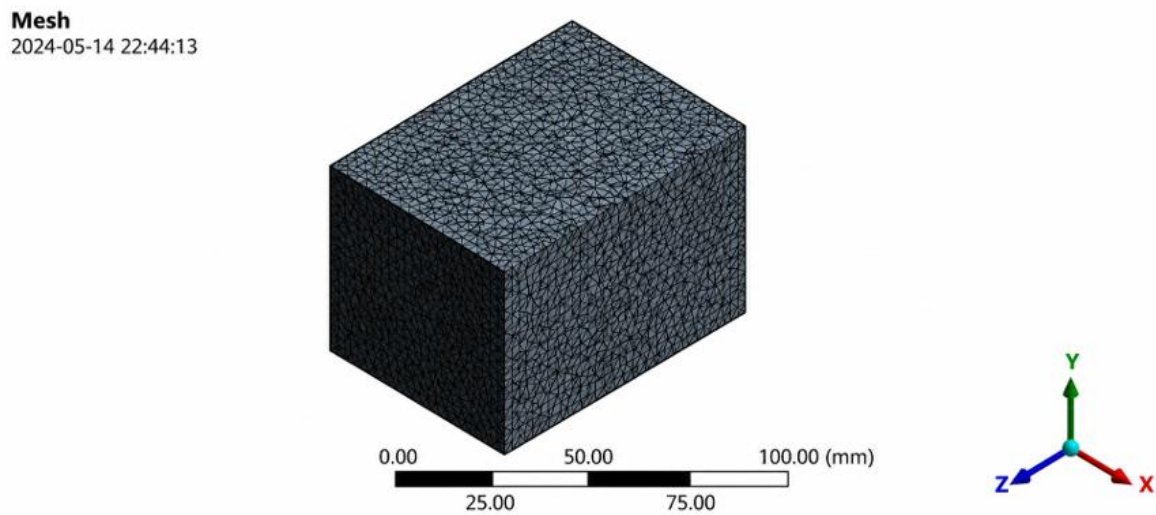


Figure 2. The Boundary Conditions for this model

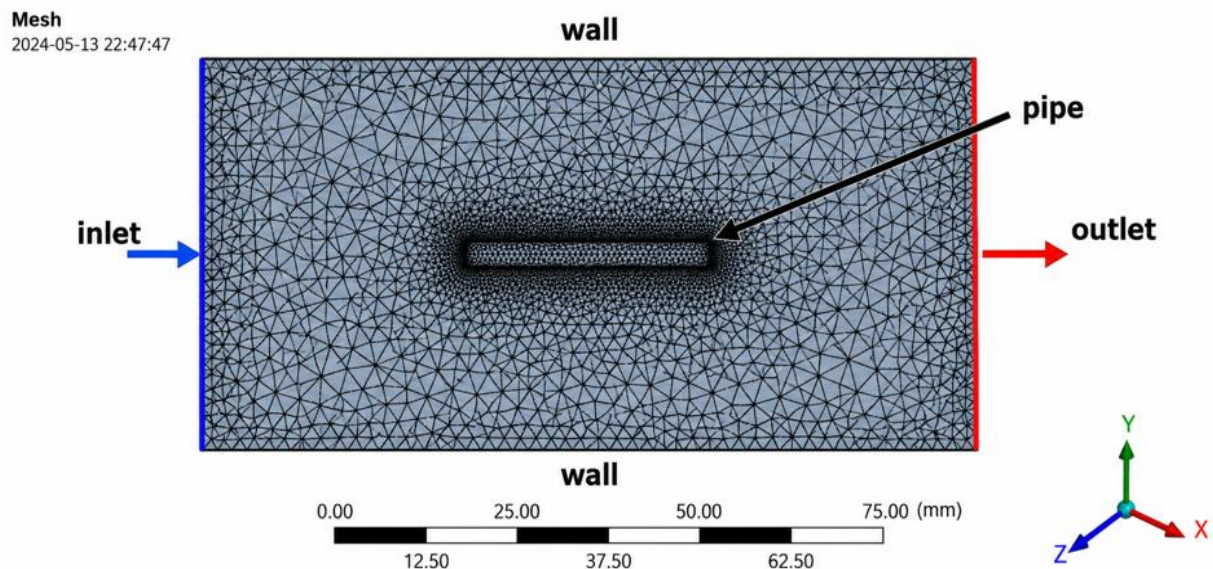


Figure 3. The details of the mesh for this model

2.3. Computational Grids Used

The entire computational domain is discretised into small control volumes or cells using ANSYS Meshing, with the Automatic Method selected as the primary meshing approach. Based on the geometry and flow characteristics, the software generates a mesh consisting primarily of tetrahedral elements throughout the fluid volume, with triangular

elements on all surface boundaries. This type of mesh provides excellent adaptability to the complex volume around the central cylinder and fills the entire rectangular enclosure completely without gaps or overlaps.

2.3.1. Element Sizing

The global element size is defined with a range from a maximum of 12.0 mm down to a minimum of 2.0 mm. This setting creates a gradual variation in mesh density: coarser elements are present in the far-field region away from the cylinder, while the mesh becomes progressively finer towards the central region. The mesh is visibly refined around the cylinder surface and in the wake zone, ensuring high resolution in areas with steep velocity and pressure gradients, flow separation, and vortex formation. For all final simulations and analysis work, the fixed element size of 2.0 mm is applied, ensuring consistent and high-accuracy results across all cases.

2.3.2. Near-Wall Refinement

To properly resolve the boundary layer development, velocity gradients, and thermal gradients close to the surface, inflation layers are applied exclusively on the pipe surface. A total of 12 to 15 layers are generated, with the first layer height carefully calculated to maintain the dimensionless wall distance y^+ value approximately equal to 1. This refinement creates a very fine mesh structure directly on the cylinder wall, ensuring correct representation of viscous effects and heat transfer behaviour.

2.3.3. Mesh Quality

After generation, mesh quality is verified using standard metrics. Skewness value is kept below 0.8, aspect ratio below 100, and orthogonal quality above 0.1. These criteria confirm that the mesh is suitable for numerical computation, stable, and capable of producing accurate and reliable results. The visual inspection confirms that the mesh is well-distributed, smooth in transition, and free from highly distorted cells. Following this sizing strategy, five different mesh configurations are created to perform the grid independence study.

2.4. Grid Independency Test (GIT)

A Grid Independency Test (GIT) is performed to verify that the numerical results are no longer significantly influenced by further refinement of the mesh. This step is necessary to eliminate discretisation errors and to confirm the appropriate mesh size for the study. The test is conducted at a constant inlet velocity of 10 m/s, using five different mesh sizes ranging from coarse to fine. The parameters monitored include total number of nodes, total number of elements, and drag coefficient (CD). Detailed data obtained from the test is presented in Table 1.

Table 1. The Grid Independency Test (GIT) Study Result

Mesh Element Size (mm)	Number of Nodes	Number of Elements	Drag Coefficient (CD)
12	10,774	60,086	1.9954×10^{-6}
9	10,858	60,466	2.0001×10^{-6}
6	11,742	64,403	-1.0772×10^{-6}
4	15,376	82,779	-1.2469×10^{-6}
2	35,180	182,475	-6.8411×10^{-8}

The results show that as the mesh size decreases and the number of nodes and elements increases, the value of drag coefficient changes progressively. Significant variation is observed from 12.0 mm down to 4.0 mm. However, when the mesh is refined further to 2.0 mm, the change in result becomes consistent and stable, indicating that the solution is no longer highly sensitive to further refinement. This confirms that the mesh size of 2.0 mm provides sufficient resolution and accuracy. Therefore, this configuration is selected and used consistently for all subsequent simulations and analysis, ensuring reliable and high-quality data for the study.

2.5. Numerical Solution Strategy

The numerical solution is carried out using ANSYS Fluent by solving the governing equations of fluid flow based on the finite volume method. The detailed solution strategy and numerical settings are defined exactly according to the setup used in this study.

2.5.1. Material Properties

The working fluid is defined as Air, with standard properties including density and viscosity set as default values from the software database.

2.5.2. Turbulence Modelling

The $k-\omega$ SST (Shear Stress Transport) model is selected for the simulation. This model combines the advantages of the standard $k-\omega$ model in the near-wall region and the $k-\epsilon$ model in the free stream, providing high accuracy for flows involving pressure gradients, separation, and complex flow behaviour around bluff bodies.

2.5.3. Solver Settings

The simulation is set as steady state. Pressure-velocity coupling is applied using the SIMPLE algorithm. Spatial discretisation for all parameters including pressure, momentum, turbulent kinetic energy, and specific dissipation rate is set to second-order upwind to ensure higher accuracy and minimise numerical errors.

2.5.4. Solution Initialization

Hybrid Initialization method is used to compute the initial values of all flow variables. This method provides a more accurate initial flow field compared to standard initialization, leading to faster convergence and better stability at the start of the calculation.

2.5.5. Convergence and Iteration

The solution process is set to run for a total of 300 iterations for each case. Residual values are monitored during the calculation process to track convergence. At the end of 300 iterations, the solution is observed to be fully converged and stable, with no significant changes in results observed beyond this point.

2.5.6. Post-Processing

Once the simulation is completed, results are extracted and analysed using ANSYS Fluent and CFD-Post. Data such as pressure distribution, velocity contours, streamline patterns, and force coefficients are obtained for analysis and discussion.

This numerical framework follows standard procedures commonly applied in CFD simulations, ensuring that the results obtained are reliable, physically correct, and suitable for detailed analysis.

3. RESULT

This chapter presents and discusses the numerical results obtained from the simulation of flow around a circular cylinder with diameter of 5 mm and length of 30 mm. The analysis compares the flow behaviour at three different inlet velocities: 10 m/s, 50 m/s, and 80 m/s. The results focus on pressure distribution, flow streamline patterns, and velocity contours, visualised and explained to understand how flow characteristics change with increasing speed. All data are derived from the final mesh configuration of 2.0 mm element size, ensuring high accuracy and consistency.

3.1. Pressure Distribution

The pressure contour plots in Figure 4 show the variation of static pressure around the cylinder for all three velocity cases, with values ranging from a maximum of 386.3 Pa down to a minimum of 348.5 Pa.

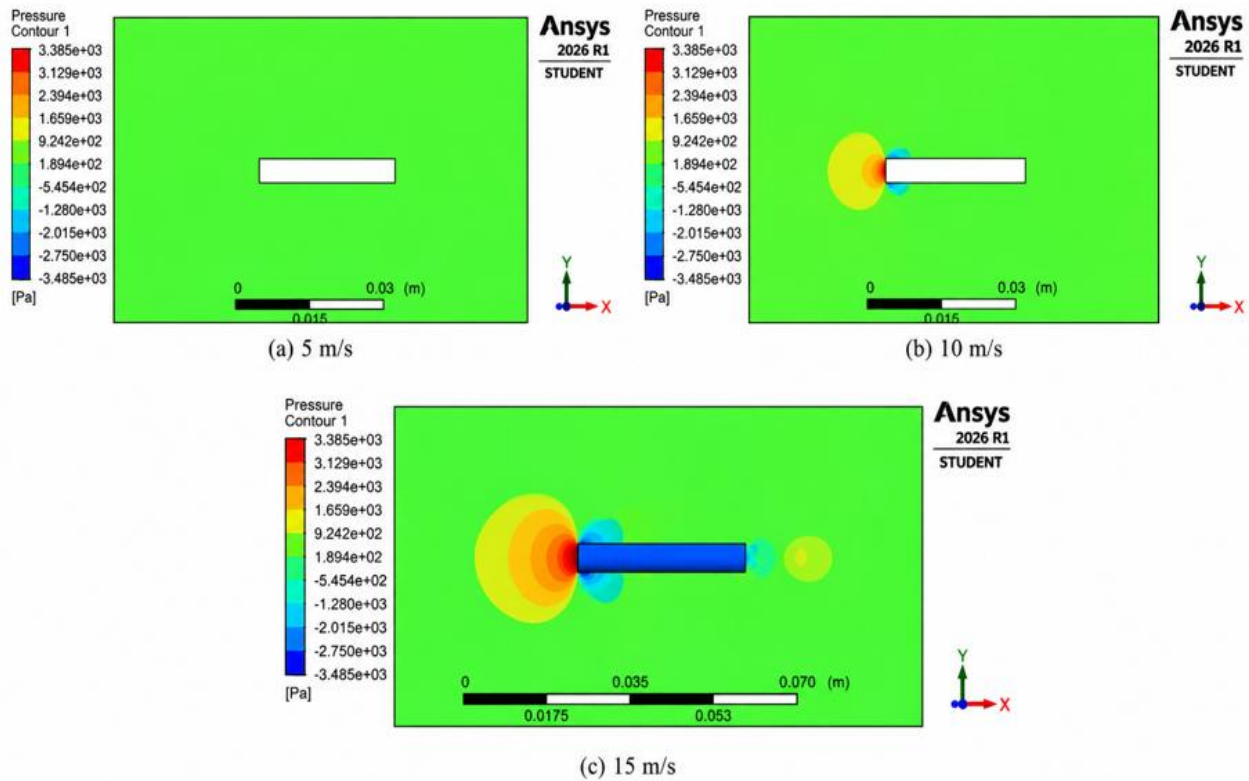


Figure 4. Pressure contour distribution at 10 m/s (Left), 50 m/s (Right), and 80 m/s (Bottom)

Based on Figure 4, the pressure field at 10 m/s (Left) appears very uniform across the entire domain. A small high-pressure zone is observed only directly at the front face of the cylinder, while the pressure around the sides and rear remains almost unchanged from the free-stream condition. The colour distribution is mostly green, indicating low pressure difference and relatively weak forces acting on the body.

At 50 m/s (Right), A clear change in pressure distribution is visible. The high-pressure region at the front expands and becomes more intense, shown by yellow and light orange colours. At the same time, distinct low-pressure zones (blue colour) start to form on the sides and rear portion of the cylinder. This indicates that as velocity increases, the pressure difference between the front and back surfaces becomes significantly larger, leading to higher drag force.

At 80 m/s (Bottom), the pressure variation is most pronounced at this highest velocity. The front stagnation point shows the maximum pressure value, represented by bright red and orange colours, covering a larger area. Around the cylinder surface and in the wake region, the pressure drops sharply to the lowest negative values, indicated by dark blue colours. The difference between the highest and lowest pressure becomes very large, confirming that pressure force increases drastically with velocity.

In all cases, the pressure distribution remains symmetrical about the horizontal axis, showing that the flow is balanced and steady. The far-field region maintains constant pressure, confirming that the domain boundaries do not affect the results.

3.2. Flow Streamlines

The streamline patterns shown in Figure 5 illustrate the path of fluid particles, visualising how the flow moves around the cylinder at each velocity. The velocity magnitude scale ranges from 0 m/s up to a maximum of 91.21 m/s.

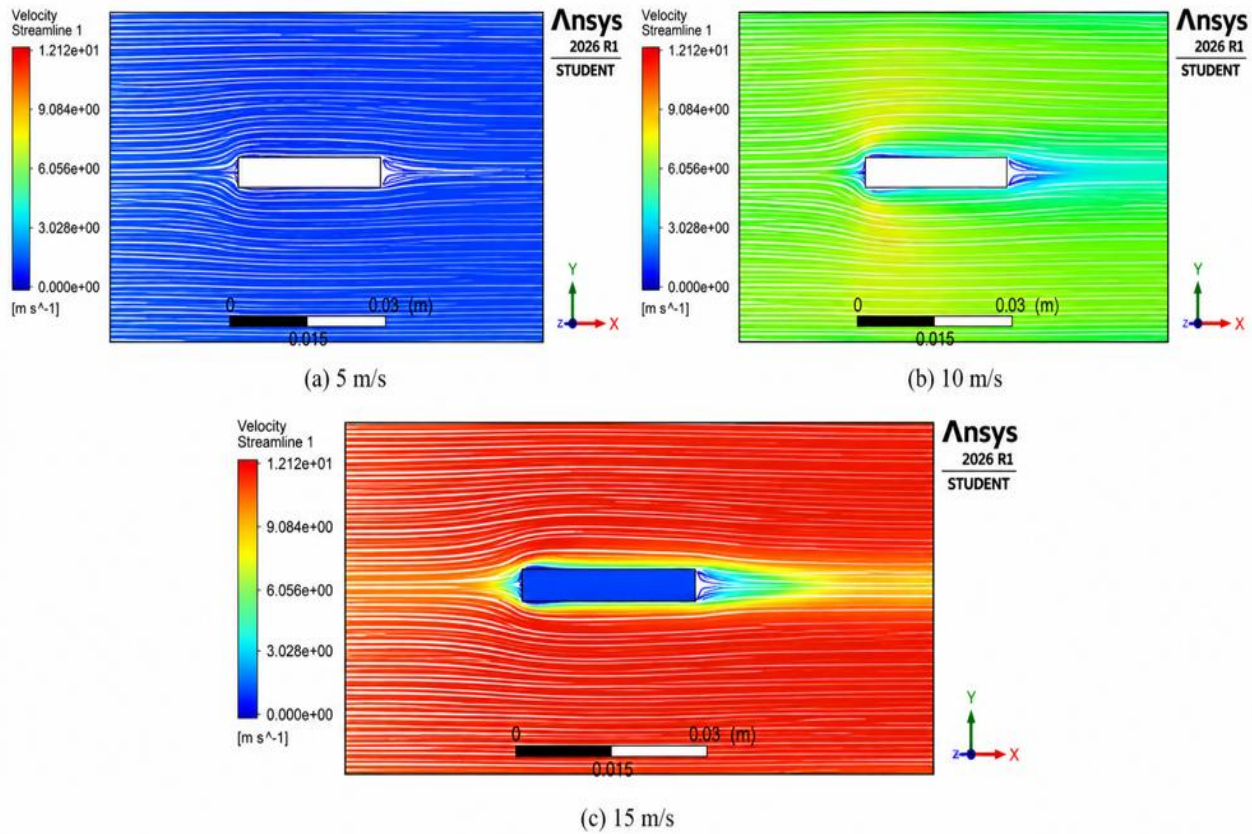


Figure 5. Flow streamline pattern at 10 m/s (Left), 50 m/s (Right), and 80 m/s (Bottom)

As shown in Figure 5, Streamlines are straight, parallel, and uniformly spaced before approaching the cylinder. As they pass the body, they bend gently and follow the surface curvature smoothly 10 m/s (Left). The streamlines remain attached to the surface and converge again immediately behind the cylinder, creating a very narrow wake region. No flow separation or recirculation is observed, indicating a very smooth and attached flow behaviour.

At 50 m/s (Right), the streamlines begin to show stronger curvature. They are pushed further outward as they pass around the cylinder, creating a wider gap between the flow and the body surface. Behind the cylinder, the streamlines take a longer distance to converge back together, forming a larger and more distinct wake zone. The flow starts to separate slightly earlier along the surface compared to the lower speed case.

At 80 m/s (Bottom), the effect of high velocity is clearly visible. Streamlines curve sharply around the cylinder, creating a very wide passage at the sides. The flow separates much earlier from the surface, and the wake region behind the cylinder becomes significantly broader and longer. The streamlines converge only far downstream, showing that the disturbance caused by the cylinder extends over a larger area at higher speeds.

Throughout all cases, the streamline patterns remain symmetrical, confirming stable flow conditions and correct numerical setup.

3.3. Velocity Contour

The velocity contour plots in Figure 6 display the magnitude of fluid velocity, ranging from 0 m/s to 91.21 m/s, clearly showing regions of acceleration and deceleration at each operating speed.

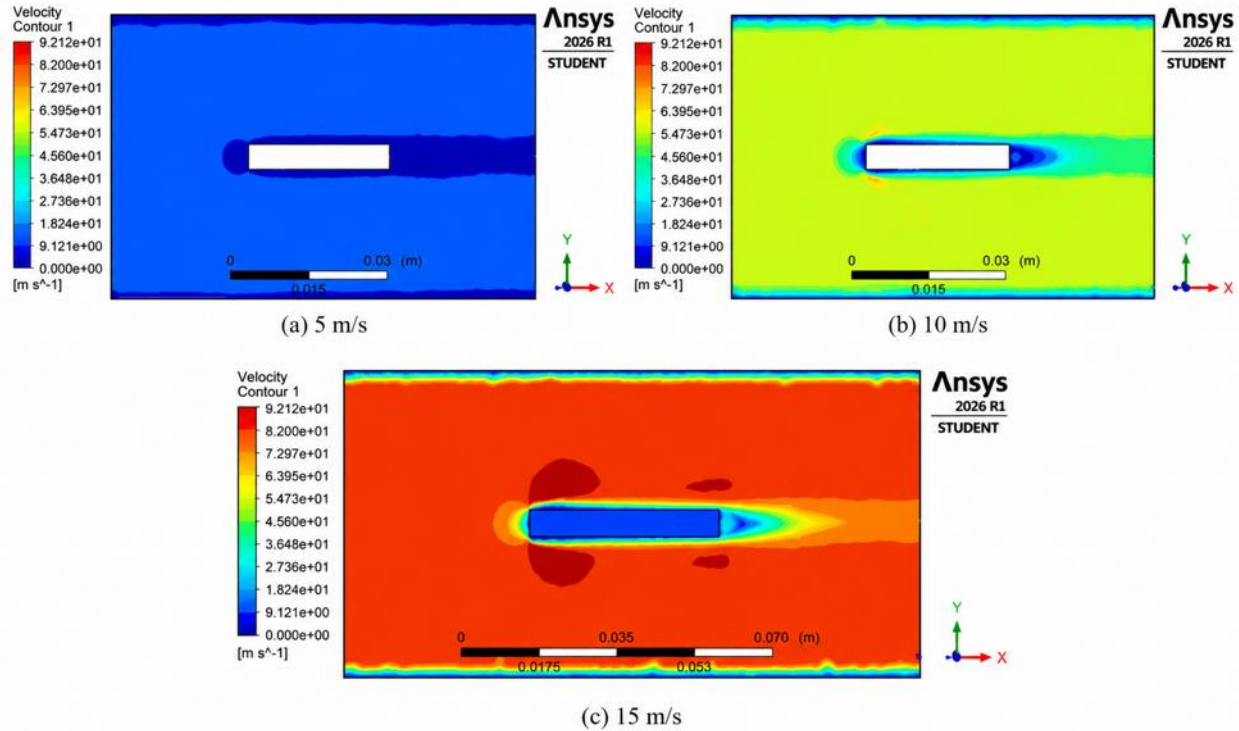


Figure 6. Velocity contour distribution at 10 m/s (Left), 50 m/s (Right), and 80 m/s (Bottom)

From Figure 6, the velocity distribution is very uniform across the domain at 10 m/s (Left) which is mostly in blue color. A small low-velocity zone exists only at the front stagnation point, while the velocity along the sides increases slightly. The wake region behind the cylinder is narrow and the velocity deficit is minimal, meaning the flow recovers its speed very quickly. The maximum velocity recorded in the domain is low and close to the inlet value.

At 50 m/s (Right), significant changes are observed. The colour shifts from blue to green and yellow, indicating higher velocity levels overall. A clear high-velocity region develops along the upper and lower curved surfaces of the cylinder, where the fluid accelerates as it moves around the obstacle. The low-velocity wake zone behind the cylinder becomes wider and more defined, showing a larger area where velocity is reduced compared to the free stream.

At 80 m/s (Bottom), this case shows the most intense velocity variation, represented by bright yellow, orange, and red colours covering most of the domain. The flow accelerates strongly around the cylinder sides, reaching the maximum velocity of 91.21 m/s. The low velocity region at the front and in the wake is very large and clearly separated from the high speed flow. The velocity gradient becomes very steep near the surface, indicating strong shear forces and boundary layer effects.

The results clearly demonstrate that as inlet velocity increases, the magnitude of velocity around the cylinder rises proportionally, and the area affected by the presence of the body becomes larger and more significant. The symmetry in all contours confirms that the flow field is well-balanced and physically consistent.

4. CONCLUSION

This study presents a numerical investigation of flow characteristics around a circular cylinder of 5 mm diameter and 30 mm length using ANSYS Fluent. The effects of inlet velocities of 10 m/s, 50 m/s, and 80 m/s were analysed in terms of pressure distribution, streamline patterns, and velocity contours, using a validated mesh size of 2.0 mm and the SST $k-\omega$ turbulence model.

The results show that pressure variation becomes more significant as velocity increases. At 10 m/s, the pressure field is mostly uniform with only a small high-pressure zone at the front. As velocity rises to 50 m/s and 80 m/s, the difference between high pressure at the stagnation point and low pressure in the wake becomes much larger, leading to higher drag force acting on the surface. The streamline patterns indicate smooth and attached flow at low speed, while higher velocities cause greater deflection, earlier separation, and a wider wake region behind the cylinder. Velocity distribution also changes noticeably, with minimal acceleration at 10 m/s, distinct high-speed zones forming at 50 m/s, and strong acceleration reaching up to 91.21 m/s at 80 m/s, accompanied by steep gradients near the surface.

Overall, the study confirms that inlet velocity directly influences flow behaviour, where higher speed results in larger pressure differences, broader flow disturbance, and stronger acceleration around the body. The numerical method applied is proven reliable and produces consistent results, suitable for understanding flow physics and supporting related engineering applications.

REFERENCES

- [1] Kumar A, Joshi JB, Nayak AK, Vijayan PK. 3D CFD simulation of air cooled condenser-I: Natural convection over a circular cylinder. *International Journal of Heat and Mass Transfer*. 2014 Nov 1;78:1265-83. DOI: [10.1016/j.ijheatmasstransfer.2015.10.048](https://doi.org/10.1016/j.ijheatmasstransfer.2015.10.048)
- [2] Catalano P, Wang M, Iaccarino G, Moin P. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. *International journal of heat and fluid flow*. 2003 Aug 1;24(4):463-9. <https://doi.org/10.3390/fluids11030068>
- [3] Rahimi H, Tang X, Esmaeeli Y, Li M, Pourbakhtiar A. Numerical simulation of flow around two side-by-side circular cylinders at high Reynolds number. *International Journal of Heat and Technology*. 2020 Mar 1;38(1):77-91. <https://doi.org/10.18280/ijht.380109>
- [4] Hubova O, Veghova I, Kralik J. Experimental and numerical investigation of in-line standing circular cylinders in steady and turbulent wind flow. In *IOP Conference Series: Materials Science and Engineering* 2019 Sep 1 (Vol. 603, No. 3, p. 032008). IOP Publishing. DOI: [10.1088/1757-899X/603/3/032008](https://doi.org/10.1088/1757-899X/603/3/032008)
- [5] Apacoglu B, Paksoy A, Aradag S. CFD analysis and reduced order modeling of uncontrolled and controlled laminar flow over a circular cylinder. *Engineering Applications of Computational Fluid Mechanics*. 2011 Jan 1;5(1):67-82. DOI: [10.1080/19942060.2011.11015353](https://doi.org/10.1080/19942060.2011.11015353)
- [6] Naito H, Fukagata K. Numerical simulation of flow around a circular cylinder having porous surface. *Physics of Fluids*. 2012 Nov 1;24(11). <https://doi.org/10.1063/1.4767534>
- [7] Mallick M, Kumar A, Murmu A. Flow Modeling in various cylindrical surfaces. *Aquatic Procedia*. 2015 Jan 1;4:834-40. DOI: [10.1016/j.aqpro.2015.02.104](https://doi.org/10.1016/j.aqpro.2015.02.104)
- [8] Olsson EE, Ahrne LM, Trägårdh AC. Heat transfer from a slot air jet impinging on a circular cylinder. *Journal of Food Engineering*. 2004 Aug 1;63(4):393-401. DOI: [10.1016/j.jfoodeng.2003.08.009](https://doi.org/10.1016/j.jfoodeng.2003.08.009)
- [9] Xu F, Chen WL, Duan ZD, Ou JP. Large eddy simulation of passive jet flow control on the wake of flow around a circular cylinder. *Computers & Fluids*. 2020 Jan 15;196:104342. <https://doi.org/10.1016/j.compfluid.2019.104342>
- [10] Olsson EE, Janestad H, Ahrné LM, Trägårdh AC, Singh RP. Determination of local heat-transfer coefficients around a circular cylinder under an impinging air jet. *International Journal of Food Properties*. 2008 Jul 18;11(3):600-12. <https://doi.org/10.1080/10942910701578759>
- [11] Khademinezhad T, Talebizadeh P, Rahimzadeh H. Numerical study of unsteady flow around a square cylinder in compare with circular cylinder. In *Conference paper* 2015.
- [12] Olsson EE, Ahrne LM, Trägårdh AC. Flow and heat transfer from multiple slot air jets impinging on circular cylinders. *Journal of food engineering*. 2005 Apr 1;67(3):273-80. DOI: [10.1016/j.jfoodeng.2003.08.009](https://doi.org/10.1016/j.jfoodeng.2003.08.009)

- [13] Lei K, Tang Z, Sun Z. Harvesting airflow energy from circular cylinder wake via a thin polyvinylidene fluoride film. *Journal of Wind Engineering and Industrial Aerodynamics*. 2022 Dec 1;231:105235. DOI: [10.1016/j.jweia.2022.105235](https://doi.org/10.1016/j.jweia.2022.105235)
- [14] Isaev S, Baranov P, Popov I, Sudakov A, Usachov A, Guvernuyuk S, Sinyavin A, Chulyunin A, Mazo A, Demidov D, Dekterev A. Numerical simulation and experiments on turbulent air flow around the semi-circular profile at zero angle of attack and moderate Reynolds number. *Computers & Fluids*. 2019 Jun 30;188:1-7. DOI: [10.1016/j.compfluid.2019.03.013](https://doi.org/10.1016/j.compfluid.2019.03.013)
- [15] Firat E, Oruç V, You R, Biçer U, Devocioğlu AG. Reduction of fluid forces on a circular cylinder in the laminar flow regime using a pair of airfoils. *Ocean Engineering*. 2025 Nov 15;339:122074. DOI: [10.1016/j.oceaneng.2025.122074](https://doi.org/10.1016/j.oceaneng.2025.122074)
- [16] Xu Z, Redo MA, Watanabe M. Convective heat transfer with condensation in crossflow of a circular cylinder: Experiments and simulations. *International Journal of Heat and Mass Transfer*. 2024 Jun 1;224:125371. DOI: [10.1016/j.ijheatmasstransfer.2024.125371](https://doi.org/10.1016/j.ijheatmasstransfer.2024.125371)
- [17] Kondjoyan A, Boisson HC. Comparison of calculated and experimental heat transfer coefficients at the surface of circular cylinders placed in a turbulent cross-flow of air. *Journal of Food Engineering*. 1997 Nov 1;34(2):123-43. DOI: [10.1016/S0260-8774\(97\)00103-9](https://doi.org/10.1016/S0260-8774(97)00103-9)
- [18] Bari S, Saad I. CFD modelling of the effect of guide vane swirl and tumble device to generate better in-cylinder air flow in a CI engine fuelled by biodiesel. *Computers & fluids*. 2013 Sep 15;84:262-9. DOI: [10.1016/j.compfluid.2013.06.011](https://doi.org/10.1016/j.compfluid.2013.06.011)
- [19] Gao DL, Chen WL, Li H, Hu H. Flow around a slotted circular cylinder at various angles of attack. *Experiments in Fluids*. 2017 Oct;58(10):132. <https://doi.org/10.1007/s00348-017-2417-8>
- [20] Csernyei C, Straatman AG. Forced convective heat transfer on a horizontal circular cylinder due to multiple impinging circular jets. *Applied Thermal Engineering*. 2016 Jul 25;105:290-303. DOI: [10.1016/j.applthermaleng.2016.05.166](https://doi.org/10.1016/j.applthermaleng.2016.05.166)
- [21] Sokolov P, Virk M. Study of airflow behavior for duplex circular cylinders. *The International Journal of Multiphysics*. 2023 Sept 16;17(3).