KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134

A Comparative Study of Internal Flow Dynamic Using CFD: Simulation of Turbulent Flow in Diffuser Pipes

Annur Adilia Maisarah Mat[1,*]

[1] Department of Mechanical Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 5 May 2025
Received in revised form 18 May 2025
Accepted 11 June 2025
Available online 26 June 2025

Keywords:
Annular diffuser pipe; turbulent flow; ANSYS fluent; internal flow; grid independence test

ABSTRACT

Diffusers play an important role in fluid flow systems by reducing fluid velocity and converting the kinetic energy into pressure energy, which improves system effectiveness and reduces pressure losses. However, optimizing diffuser geometry is challenging due to the complex relationship between velocity, pressure distribution, and turbulence, which traditional analytical methods struggle to capture accurately. This research addresses this problem by using computational fluid dynamics (CFD) with ANSYS Fluent to simulate fluid flow through a diffuser pipe, aiming to improve pressure recovery and minimize energy losses. The simulation focuses on the impact of diffuser geometries on key flow characteristics such as velocity reduction, pressure distribution, and turbulence effects. By modelling steady-state, incompressible water flow at room temperature and comparing results with theoretical predictions, the study provides insights into optimizing diffuser design. The findings demonstrate the advantages of CFD in accurately predicting complex flow behaviours, offering valuable guidance for enhancing diffuser efficiency in various engineering applications.

1. Introduction

Diffusers play a vital role in fluid flow systems, commonly utilized across engineering fields to control and improve fluid movement through ducts, compressors, and HVAC systems. Their primary function is to slow down fluid flow, converting kinetic energy into pressure energy, thus increasing static pressure within a system [1]. This process is crucial for reducing pressure losses and improving energy efficiency, particularly in industries like aerospace and automotive engineering. The flow through axisymmetric expansions, as seen in wide-angle diffuser geometries, can lead to turbulent and separated flow fields in practical situations [2]. Additionally, diffuser pipes help adjust high Mach number slanted flows by increasing pressure ratios in centrifugal compressors [3]. Using ANSYS Fluent in computational fluid dynamics (CFD), this study simulates fluid flow through a diffuser pipe, focusing on the effects of geometry on velocity reduction, pressure recovery, and turbulence.

* *Corresponding author.*
E-mail address: gd240024@student.uthm.edu.my

https://doi.org/10.37934/afhme.5.1.1018a

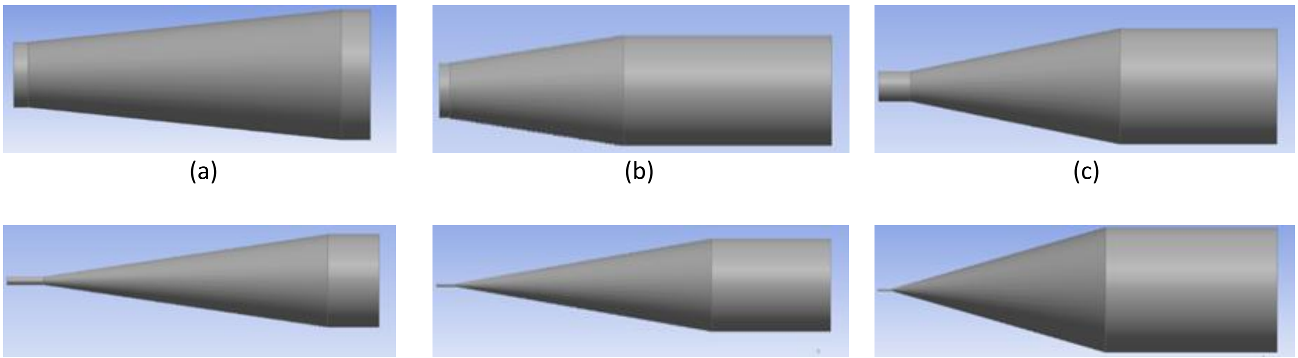
The objectives include analysing flow separation, turbulence effects, and pressure distribution to optimize the diffuser design for efficient fluid management. Water at room temperature is used as the working fluid, with the assumption of steady-state, incompressible, single-phase flow, neglecting surface roughness and heat transfer. By simulating these conditions, the study aims to predict key performance metrics, such as pressure recovery and areas prone to flow separation, ultimately contributing to better diffuser designs and reduced energy losses [4].

2. Methodology
2.1 Geometry

The dimensions of the diffuser pipes are varied. This can be explained through the simulation where the diameter of the inlet, outlet, angle and the length of a diffuser is varied based on the Table 1. The geometries of the diffuser pipe are varied depending on the parameters chosen. It is visualized based on these figure as shown in Figure 1(a) to Figure 1(f).

Table 1
Parameter of diffuse pipes

Geometry	D_{inlet}(m)	D_{outlet}(m)	$\Theta_{diffuser}$	L_{pipe}(m)
G1	0.200	0.40	6.50˚	1.0
G2	0.100	0.20	4.30˚	1.5
G3	0.026	0.10	8.00˚	0.5
G4	0.013	0.15	5.14˚	1.0
G5	0.013	0.40	8.50˚	2.0
G6	0.013	0.90	15.40˚	3.0



(a) (b) (c)

(d) (e) (f)

Fig. 1. Geometries of diffuser pipes in different angle of inlet (a) G1 (b) G2 (c) G3 (d) G4 (e) G5 (f) G6

2.2 Boundary Condition Parameter

A constant inlet velocity of 0.297 m/s is applied to the diffuser pipe based on the Reynolds number, ensuring a stable flow for pressure recovery analysis. The outlet is set to 0 Pa, representing atmospheric pressure, allowing for pressure adjustments as the fluid decelerates. The pipe wall is modelled with a no-slip condition and zero roughness, essential for capturing viscous effects and boundary layer development. The k-ε turbulence model is used to handle boundary layers and flow separation [5]. The diffuser geometry is discretized with a tetrahedral mesh, chosen for its adaptability to complex shapes and ability to capture irregular flow patterns. Inflation layers near the wall ensure proper resolution of the turbulent boundary layer. Mesh quality was maintained with skewness below 0.25, and a final mesh size of 272,128 elements was chosen after performing a grid independence test (GIT) to balance accuracy and computational cost.

2.3 Discretization of Governing Equation

To summarize the governing equations of diffuser pipe flow, Navier-Stokes equations. Eq. (1) is used for incompressible flow [6].

$$(u \cdot \nabla)u = -\frac{1}{\rho}\nabla p + \upsilon\nabla^2 + f \tag{1}$$

where ρ is density of fluid, p is static pressure, f is any external body force, $(u \cdot \nabla)u$ is convective term and $\frac{\partial u}{\partial t}$ is velocity vector. For incompressible and steady state, the equation is simplified into Eq. (2).

$$\nabla \cdot u = 0 \tag{2}$$

The diffuser angle using fluid dynamics principles can be determined based on Eq. (3):

$$tan\,\Theta = \frac{D_{outlet}}{D_{inlet}} \tag{3}$$

where $\mathrm{D_{outlet}}$ is the diameter of the diffuser outlet, $\mathrm{D_{inlet}}$ is the diameter of the diffuser inlet. The turbulence intensity at the inlet needs to be setting based on the Eq. (4).

$$\text{Turbulence Intensity} = \frac{0.16}{(Re_D)^{\frac{1}{8}}} \tag{4}$$

2.4 Grid Independence Test (GIT)

The grid independence test (GIT) is crucial in diffuser pipe simulations to ensure the accuracy and reliability of the results. In CFD simulations, the mesh discretizes the governing equations, and its quality effects solution accuracy. The GIT refines the mesh in successive steps by comparing the results such as pressure recovery, velocity distribution and flow separation. convergence indicates that the solution is independent of the grid resolution, ensuring the mesh captures critical flow features such as boundary layer development and flow separation without excessive computational cost. For internal diffuser flow, where complex phenomena occur, the GIT ensures accurate, mesh independent results for pressure recovery and flow efficiency predictions.

3. Results

3.1 Velocity Distribution

Based on Figure 2(a) to Figure 2(f), it is evident that the velocity has gradually decreases along the length of the pipe as the flow expands. The colour map indicates that the area of high velocity near the inlet transitioning to lower velocity near the outlet, as expected from a diffuser where kinetic energy is converted into pressure. Such in Figure 2(a) and Figure 2(b), these geometries show a relatively smooth deceleration of flow with minimal separation, indicating a stable flow and efficient pressure recovery. However, geometry in Figure 2(c) has revealed some flow instability, with a noticeable drop in velocity occurring midway, suggesting minor flow separation. Velocity distribution in Figure 2(d) has shown more stable than Figure 2(c) but shows a moderate reduction in velocity which making it somewhat effective in pressure recovery. Moreover, such in geometries in Figure

2(e) and Figure 2(f) exhibit significant flow separation where the larger angle causes substantial recirculation and velocity loss near the walls of the pipe [7]. This would lead to inefficiency in pressure recovery. The most efficient design in term of pressure recovery and minimizing flow separation appears to be in Figure 2(a) as it balances a sufficient opening angle to allow expansion while maintaining stable flow conditions and minimizing losses. Geometry on Figure 2(b) performs well, yet the smaller opening might slightly limit the rate of pressure recovery compared to geometry in Figure 2(a).

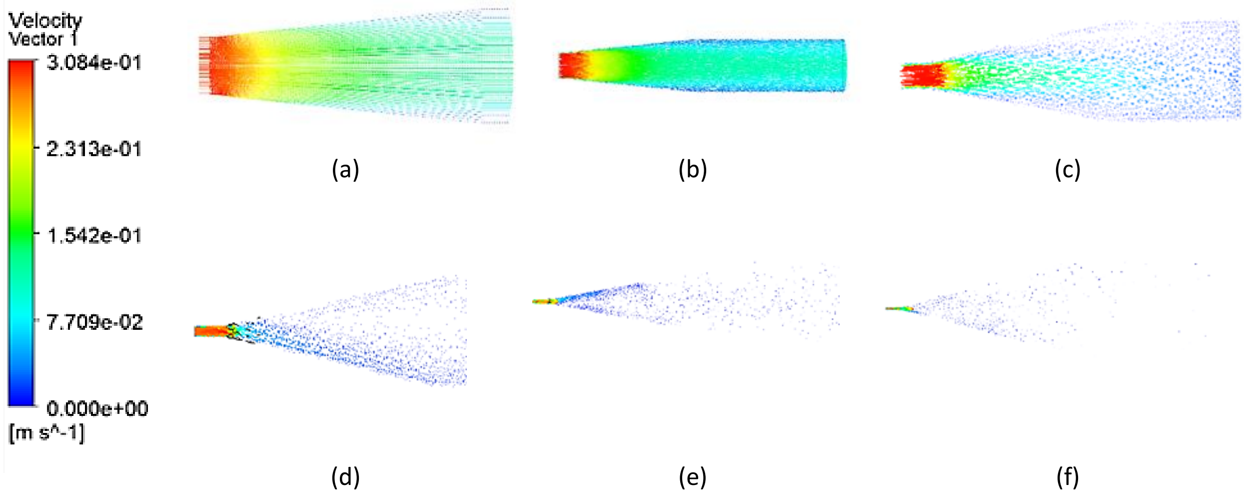


Fig. 2. Velocity distribution of diffuser pipes in different angle of inlet (a) G1 (b) G2 (c) G3 (d) G4 (e) G5 (f) G6

3.2 Pressure Distribution

In Figure 3, the pressure distribution across G1 is more stable and favourable compared to the larger-angle diffusers. The small diffuser angle provides a gradual expansion of the floe, which minimizes the adverse pressure gradient and helps to maintain the attached flow along the walls of the diffuser. This results in less flow separation and lower recirculation zones, ensuring the majority of flow follows the diffuser walls smoothly [8]. As the flow decelerates, there is an effective conversion of kinetic energy into pressure. The pressure gradually increases along the diffuser length, leading to efficient pressure recovery. The controlled expansion of flow results in a uniform pressure distribution while reducing energy losses due to turbulence or separation. This geometry is more reliable as in maintaining a high-efficiency pressure recovery, making it the most suitable design for diffuser applications where smooth flow and energy efficiency are crucial.

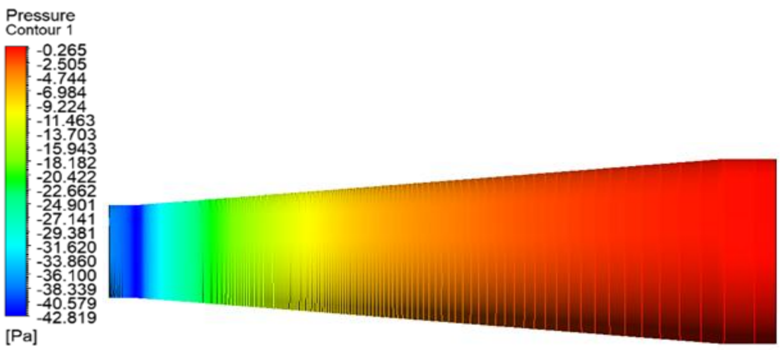
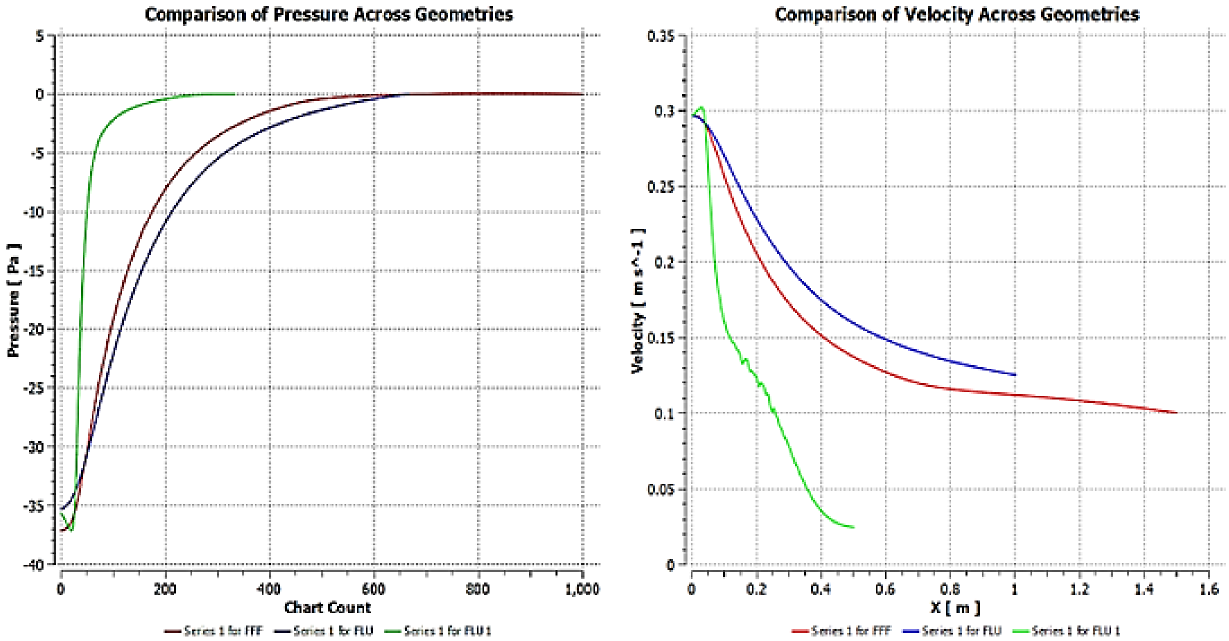


Fig. 3. Pressure distribution across G1

3.3 Comparison of Pressure and Velocity Across Geometries

Since the data from Figure 4 points for shorter pipes extended beyond the pipe length, these pipes clearly exhibit velocity and pressure profile distinct of 150 cm pipe. Thus, it does not intersect at any flow development stage at 100 cm analysis point. This happened due to velocity and pressure data were analyzed at the distance 150 cm which is beyond 100 cm and 50 cm respectively. This might lead to extrapolated values which deviate quite significantly from the fully developed flow profile on 150 cm of diffuser pipe [9,10].



(a) (b)

Fig. 4. Comparison of (a) Pressure (b) Velocity across geometries

3.4 Angle of Expansion, ϴ˚ and Length of Pipe

The diffuser angle of G1 (6.5˚) promotes smoother flow with minimal separation, leading to higher pressure recovery efficiency as the flow remains attached to the diffuser wall. A smaller angle between the expanding walls enhances performance and reduces turbulence. Equally, in G6 (15.4˚) has seen that the larger angle creates higher turbulence and pressure drops due to flow separation [11]. The results align with theory, where larger angles typically lead to greater recirculation and efficiency [12]. While increasing the outlet diameter and length of the diffuser helps reduce velocity and improve pressure recovery, the optimal angle remains undetermined, varying with pipe dimensions. This simulation confirms that a lower diffuser angle enhances efficiency. Meanwhile, a longer diffuser pipe also allows for more gradual flow expansion which leads to smoother velocity and potentially less flow separation. It can be explained through the geometry of G1 has 1 m length and lower diffuser angle which makes it more suitable design for a diffuser. Plus, a longer diffuser pipe will provide more length for pressure recovery [13]. That means no rapid expansion which can cause turbulence and increase in pressure drop [14]. This characteristic can be seen in G1.

3.5 Grid Independence Test (GIT)

GIT confirms that the obtained data of CFD simulation does not considerably depend on the selected mesh density. GIT across geometry G1 is tabulate in Table 2. The aim of the GIT is to ensure the results of the pressure drop and velocity are independent of the mesh size. As the element size decreases, the number of elements is increased. The pressure drops across the diffuser pipe is calculated at each mesh refinement level [15]. Pressure drop starts emerge with the finer mesh which indicating the result are more accurate. The maximum velocity achieved in the flow domain where the velocity slightly increases from 0.304096 m/s (coarse) to 0.308360 m/s (fine). The percentage difference in velocity between the mesh level is calculated as below:

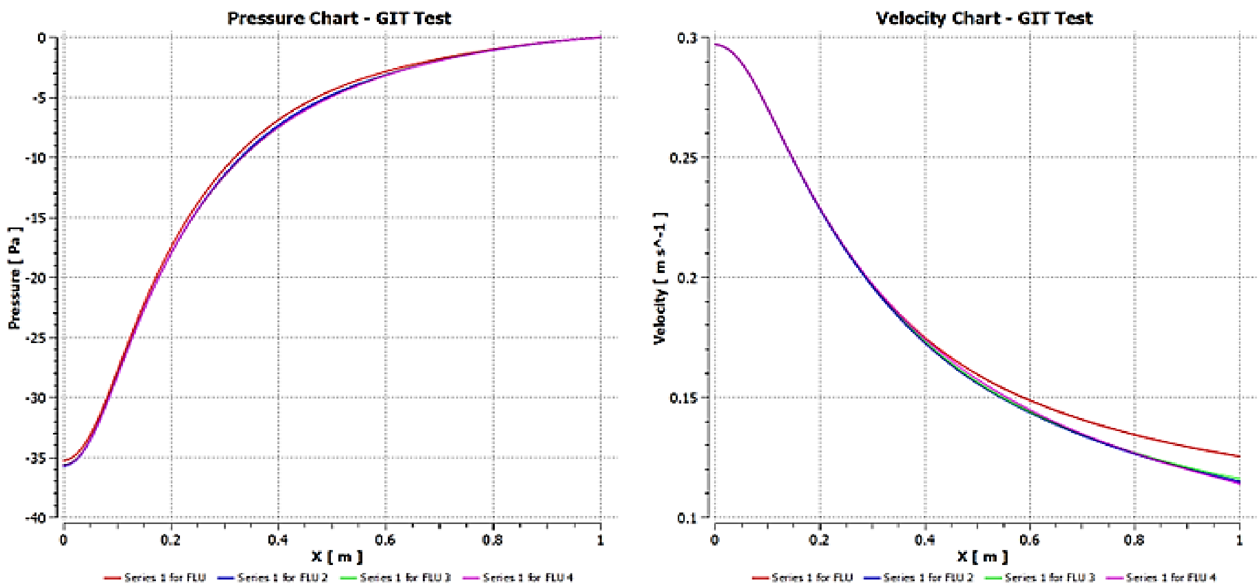
$$\% \text{ difference } = \frac{|V_{\text{fine}} - V_{\text{medium}}|}{V_{\text{medium}}} \times 100 \tag{5}$$

$$\% \text{ difference} = \frac{|0.308360 - 0.304792|}{|0.304792|} \times 100 = 1.17\% \tag{6}$$

Table 2
Grid independence test (GIT) for 1 m pipe, with angle of 6.5˚

Mesh level	Element size (mm)	No. of elements	Pressure drop, P_{drop}(Pa)	Maximum velocity, V_{max}
Coarse	8.0	301,473	0	0.304096
Medium	6.0	617695	0	0.304792
Fine	5.8	695,975	0.00112356	0.308360

The percentage indicates that there is only about 1.17% difference between the result from the medium and fine mesh which showing the percentage is small. This suggests that the solution is converging and the mesh is sufficient. Hence, the fine mesh can be considered as sufficient for accurate result. Moreover, during the fine mesh, longer time is needed. This is might be because of high number of elements. the GIT is achieved. The given curves in Figure 5 represent the velocity and pressure distribution profiles against node numbers, which we can conclude that the velocities increase with node numbers.



(a) (b)

Fig. 5. Chart for grid independence test (a) Pressure chart (b) Velocity chart

As the mesh is refined further, there is a minor variation in results, and both profiles are almost constant as the mesh size reaches 695,975 elements. This suggests the results have become grid-independent within the range [16]. In conclusion, the solution does not depend on the grid density. Hence, it is obtained that if any additional number of elements increases above 695,975 of this software will make it more complex in terms of computations but adds almost negligible value to improve the accuracy of the result.

3.6 Evidence of Convergence and Accuracy

Based on the Figure 6 below, the solution is converged before iteration of 400 which means that the solution has stabilized and no longer changes significantly with additional iterations. This can be concluded that the solution is both accurate and stable where no decrease significantly or oscillate [17].

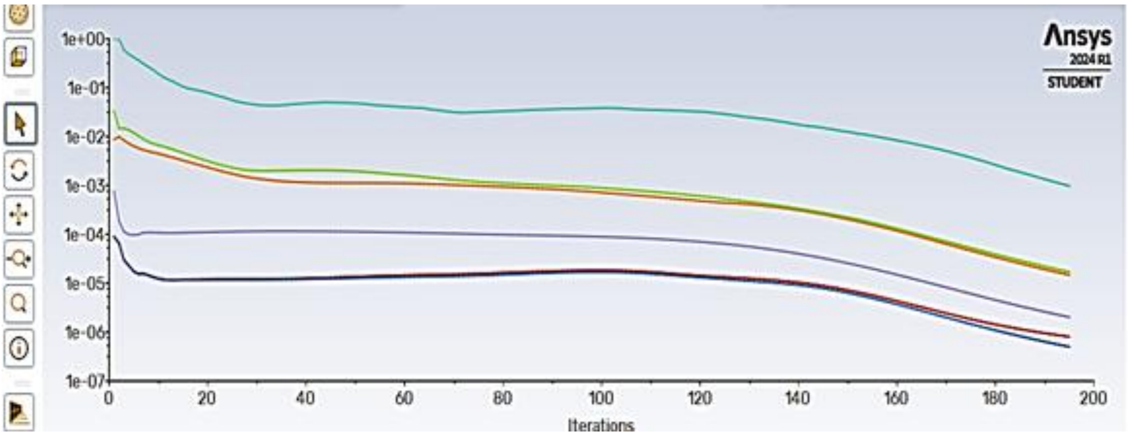


Fig. 6. Convergence of solution

3.7 Comparison with Published Values

All the data from the simulation on different parameters which are includes the diffuser angle, diameter of the outlet and length of the diffuser pipe are tabulate in the Table 3 shown below. Based on the Table 3, it comprises that the diffuser pipe of G1 and G2 has the better design as no flow separation or circulation occurs. That justify the theory of lower angle of wall expansion, bigger diameter of outlet, longer pipe of diffuser can reduce the flow separation and reduce the turbulence happened in the pipe [18,19]. In contrast, higher angle expansion such in G6 would introduce to flow separation and turbulence flow. The inlet diameter of 0.013 m needs to be changed as it is too small for a diffuser pipe and simulation cannot detect a good reading [20]. However, in the previous study reviewed, no published values for the specific angles used in this study were found. This is primarily because the exact geometries used in those studies differ from the one considered here. The geometries in the earlier works were not identical to the current model. Therefore, the specific angles examined in this research do not have direct comparisons with previously published data.

Table 3
Comparison of different parameters of diffuser pipe

Geometry No.	Diffuser Angle (ϴ˚)	Diameter of outlet (m)	Length of pipe, L (m)	Velocity inlet (m/s)	Reynolds number, Re	Pressure drop	Maximum velocity	Flow separation	Turbulent intensity (%)
G1	6.50˚	0.4	1	0.297	66541	0.000	0.304	No	4%
G2	4.30˚	0.2	1.5	0.297	33270	0.079	0.300	No	4%
G3	8.00˚	0.1	0.5	0.297	8650	0.015	0.015	No	5%
G4	5.14˚	0.15	1	0.297	4325	0.961	0.960	Yes	6%
G5	8.50˚	0.4	2	0.297	4325	0.005	0.005	Yes	6%
G6	15.40˚	0.9	3	0.297	4325	0.060	0.060	Yes	6%

4. Conclusions

In conclusion, the simulation study on the turbulent flow in a diffuser pipe demonstrated that mesh refinement had a significant impact on the accuracy of the result, with the fine mesh providing the most reliable results based on GIT. The geometry of a diffuser, especially on the angle, diameter of inlet, and length of the pipe play a crucial role in influencing flow characteristics such as velocity distributions, pressure recovery, and flow separation. Larger angle of expansion resulted in greater flow separation and pressure drops while smaller angle of expansion led to uniform flow. The simulation results aligned well with the theoretical prediction which showing a balance between diffuser efficiency and turbulence. Nevertheless, limitation such as assumption steady-state flow, boundary conditions, limitation of elements due to student version and the use of standard turbulence model should be considered as real-world conditions might involve more complexity. Recommendation on future study could explore more on transient's simulations and the effect of other variables such as surface roughness for diffuser design optimization.

References

[1] Han, Ge, Chengwu Yang, Shixun Wu, Shengfeng Zhao, and Xingen Lu. "The investigation of mechanisms on pipe diffuser leading edge vortex generation and development in centrifugal compressor." *Applied Thermal Engineering* 219 (2023): 119606. https://doi.org/10.1016/j.applthermaleng.2022.119606

[2] Liu, Yicai, Kai Chen, Tianlong Xin, Lihong Cao, Siming Chen, Lixin Chen, and Weiwu Ma. "Experimental study on household refrigerator with diffuser pipe." *Applied Thermal Engineering* 31, no. 8-9 (2011): 1468-1473. https://doi.org/10.1016/j.applthermaleng.2011.01.022

[3] Han, Ge, Chengwu Yang, Shixun Wu, Shengfeng Zhao, and Xingen Lu. "The investigation of mechanisms on pipe diffuser leading edge vortex generation and development in centrifugal compressor." *Applied Thermal Engineering* 219 (2023): 119606. http://doi.org/10.1016/j.applthermaleng.2022.119606

[4] Singh, Hardial, and B. B. Arora. "Effect of swirl flow on characteristics of the annular diffuser." In *IOP Conference Series: Materials Science and Engineering*, vol. 1168, no. 1, p. 012029. IOP Publishing, 2021. https://doi.org/10.1088/1757-899X/1168/1/012029

[5] Firat, Ilker, Sendogan Karagoz, Orhan Yildirim, and Fatin Sonmez. "Experimental investigation of the thermal performance effects of turbulators with different fin angles in a circular pipe." *International Journal of Thermal Sciences* 184 (2023): 107969. https://doi.org/10.1016/j.ijthermalsci.2022.107969

[6] Shojaee, Seyedamir. *A data-driven neural network model to correct derived features in a rans-based simulation of the flow around a sharp-edge bluff body*. Master Diss., University of Alberta, 2023. https://doi.org/10.7939/r3-7743-mq83

[7] Singh, Hardial, and Bharat Bhushan Arora. "Performance characteristics of flow in annular diffuser using CFD." *International Journal of Turbo & Jet-Engines* 40, no. 3 (2023): 281-291. https://doi.org/10.1515/tjj-2021-0003

[8] Masoodi, Faiz Azhar, and Rahul Goyal. "Aspects of vortex breakdown phenomenon in hydraulic turbines." *Experimental Thermal and Fluid Science* 150 (2024): 111051. https://doi.org/10.1016/j.expthermflusci.2023.111051

[9] Lopez-Santana, Gabriela, Andrew Kennaugh, and Amir Keshmiri. "Experimental techniques against RANS method in a fully developed turbulent pipe flow: evolution of experimental and computational methods for the study of turbulence." *Fluids* 7, no. 2 (2022): 78. https://doi.org/10.3390/fluids7020078

[10] Lim, Desmond C., Hussain H. Al-Kayiem, and Jundika C. Kurnia. "Comparison of different turbulence models in pipe flow of various Reynolds numbers." In *AIP Conference Proceedings*, vol. 2035, no. 1. AIP Publishing, 2018. https://doi.org/10.1063/1.5075553

[11] Chassaing, Patrick. *Fundamentals of Fluid Mechanics*. Springer: Berlin/Heidelberg, Germany, 2023. https://doi.org/10.1007/978-3-031-10086-4

[12] Chashechkin, Yuli D. "Foundations of engineering mathematics applied for fluid flows." *Axioms* 10, no. 4 (2021): 286. https://doi.org/10.3390/axioms10040286

[13] Ilić, Dejan, Jelena Svorcan, Đorđe Čantrak, and Novica Janković. "Experimental and numerical research on swirl flow in straight conical diffuser." *Processes* 13, no. 1 (2025): 182. https://doi.org/10.3390/pr13010182

[14] Mfona, Samuel A., Sunday B. Alabia, Etim S. Udoetokb, Uchechukwu H. Offora, Emmanuel U. Nseka, Zdenek Tomasc, and Tomas Miklík. "A semi-empirical model for estimation of pressure drop coefficient of a conical diffuser." *Chemical Engineering* 74 (2019).https://doi.org/10.3303/CET1974168

[15] Agarwal, A., and L. Mthembu. "CFD analysis of conical diffuser under swirl flow inlet conditions using turbulence models." *Materials Today: Proceedings* 27 (2020): 1350-1355. https://doi.org/10.1016/j.matpr.2020.02.621

[16] Pinna, Francesco, Battista Grosso, Alessio Lai, Ouiza Bouarour, Cristiano Armas, Maurizio Serci, and Valentina Dentoni. "Design, validation and CFD modeling of an environmental wind tunnel." *Atmosphere* 15, no. 1 (2024): 77. https://doi.org/10.3390/atmos15010077

[17] Talo, Ozer, and Yurdal Sever. "Γ-convergence of double sequences of functions, and minimizers." *Facta Universitatis, Series: Mathematics and Informatics* (2023): 771-791. https://doi.org/10.22190/FUMI230521050T

[18] Huang, Lim Gim, Normayati Nordin, Lim Chia Chun, Nur Shafiqah Abdul Rahim, Shamsuri Mohamed Rasidi, and Muhammad Zahid Firdaus Shariff. "Effect of turbulence intensity on turning diffuser performance at various angle of turns." *CFD Letters* 12, no. 1 (2020): 48-61.

[19] Wang, Jun, Heping Xie, and Cunbao Li. "Anisotropic failure behaviour and breakdown pressure interpretation of hydraulic fracturing experiments on shale." *International Journal of Rock Mechanics and Mining Sciences* 142 (2021): 104748. https://doi.org/10.1016/j.ijrmms.2021.104748

[20] Shan, XiangLin, YiLang Liu, WenBo Cao, XuXiang Sun, and WeiWei Zhang. "Turbulence modeling via data assimilation and machine learning for separated flows over airfoils." *AIAA Journal* 61, no. 9 (2023): 3883-3899. https://doi.org/10.2514/1.J062711