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

Simulation of Tapered Pipe Turbulent Flow in Various Internal Flow Geometries

Devyashree Krishnamurt[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 13 May 2025
Accepted 11 June 2025
Available online 26 June 2025

Keywords:
Computational fluid dynamics (CFD); geometry; k-ε turbulence

ABSTRACT

This research focuses on established turbulent flow profiles in tapered pipes of different lengths but fixed minimum external diameter and constant pipe material properties, and uses computational fluid dynamics (CFD). Three geometries were analysed: Variety of three profiles of 50 cm, 75 cm and 100 cm lengths and outlet diameter. To test the suitability of the node density, a grid independence test (GIT) was performed on the 50 cm length tapered pipe geometry to obtain an accurate yet efficient model. The mentioned derived mesh settings were adopted and applied on other geometries. Computational results were verified by comparing the velocity and pressure fields with the numerically obtained fields as well as the published findings. The k-ε turbulence model, combined with the finite volume method of the ANSYS Fluent simulated the velocity and the pressure of the flow. At the same time, marginal variations at the outlet may suggest the limitations of the chosen model. Velocity distribution and flow behaviour are depicted by contour plots where colour gradients represent geometrical effects. The present work is useful in understanding the parametric variations of the CFD model necessary for internal flow applications.

1. Introduction

Analyzing different parameters of a system like fluid flow, heat transfer and computational fluid dynamics (CFD) solves a system of equations with the help of a computer system was stated by Mund *et al.,* [1]. Leschziner [2] points out that, while, for instance, the computing power and methods for numerical and visualization have advanced rapidly, the predictive aspects of statistical turbulence models are weaker and advance slowly, although there has been much intensive work in the recent past Hence, the use of Computational Fluid Dynamics (CFD) analysis is of great significance as it provides more accurate results compared to experimental analysis as stated by previous studies [3-6]. In many cases ground water modeling, application of finite-difference approach to solve ground water flow equation is popular and many of these models need a comparatively fine grid of the computer domain to simulate the selected process in limited areas of interest [7]. The turbulence

* *Corresponding author.*
E-mail address: dd210047@student.uthm.edu.my

https://doi.org/10.37934/afhme.5.1.19a

models including k-ε, k-epsilon and k-ω, omega offer a basis for the accurate estimation of energy dissipation and eddy viscosity. They also report that further refinement of the grid is important, particularly in the areas of the abrupt change in the geometry where much more detailed flow descriptions are likely to take place.

The specific goal is to analyze velocity and pressure fields of flow in the tapered pipes of various lengths. This entails investigating the impact of tapering on the flow behavior in terms of velocity enhancement and pressure decrease in relation to uncomplicated models as well as other published information. It is an important goal to perform a GIT on one geometry in order to decide the number of nodes adequate for simulations or the density of a mesh that provides the best compromise between calculation time and precision. After determination of the best number of nodes for one of the geometries, the presented mesh refinement technique was used on the other two geometries to obtain accurate and reproducible results. The use of ANSYS Fluent software can be the approach that displays the clear predictions [8].

In the present studies, ANSYS Fluent will be used to investigate turbulent flow science, ascertain the difference between the two geometries, and analyze selected flow phenomena such as flow separation, stream wise velocity, and turbulence intensity. CFD has the potential to be used as an advanced design tool, rather than just a prediction tool [6]. The first one regards the investigation of the velocity and pressure fields in converge or diverge pipes with arbitrary lengths. These include investigating the effects of tapering on the flow dynamics in terms of velocity increase and pressure drop on the models, and comparing the calculated values with theoretical and published data. An important aim is to perform a GIT on one geometry in order to find out how many nodes or mesh density is sufficient to get a good computational performance of the algorithm with the required accuracy of results. After determining the best number of nodes for achieving the best results, this mesh refinement approach was also applied to the other two geometries for improved accuracy.

The basic flow characteristics considered are pressure and velocity. According to Bernoulli's theorem, these parameters (pressure and velocity) vary with respect to one another as fluid flows through reduction section in a pipe system [9]. Tapered pipe fluid dynamics is such a problem area, and CFD is now a necessary aid in analyzing many problems that involve the fluid flow in complex forms in turbulent regime. Mixed representations are adapted using a technique called mesh adaptation, which involves the subdivision or for coarsening of groups of cells according to a given refinement criterion [10]. A criterion has to be linked to the flow problem and the turbulence model applied for the decision-making process.

The numerical simulations also reveal that for the converging tubes, multiplying the convergence angle results into a reduction of the value of the mean apparent viscosity at the tube exit, even though the mass flow rate reduces with a specific pressure drop [11]. One key aspect of this research area is the simulation of turbulent flow within tapered pipes. This technique seems to hold the promise of giving nearer estimates of turbulent flow phenomena than the Reynolds Averaged Navier-Stokes models which use statistical turbulence models to mimic the influence of turbulence [12]. For most of these applications controlling the computational cost by the grid resolution by regions is insufficient because the required grid resolution changes within the domain [13]. The present work discusses the analysis of flow behaviour in tapered pipe geometries of different lengths and external diameters. It also involves a grid independence test for the selection of an optimal mesh that would make the results independent of grid resolution.

2. Methodology
2.1 Geometry Details

Length and the diameter of the external and internal walls of three samples are presented in Table 1, which contains geometric parameters of the samples. The length of the samples varies from 50 cm through 100 cm; the external diameters of the samples have decreased from 0,9 cm to 0,5 cm. However, internal diameter of all samples of tubes has remained almost the same with a value of 1.3 cm. Table 1 portray these values give essential dimensional parameters necessary for additional analysis or computation regarded the structural or functional properties of the samples.

Table 1
The values of geometry

Sample	Length (cm)	External diameter (cm)	Internal diameter (cm)	Nodes obtained
1	50	0.9		63910
2	75	0.7	1.3	61132
3	100	0.5		61683

2.2 Boundary Conditions

The boundary conditions of the tapered pipe are as shown in the table below as assigned. Boundary Conditions are vitally important in determining the environment and the settings, allowing better comparisons of data and analysis. Table 2 portrays the boundary conditions of the tapered pipe.

Table 2
The value of boundary condition

Boundary condition	Unit
Turbulence intensity (%)	5
Inlet velocity(m/s)	0.297
Flow conditions	Water
Reynolds number	3840
Inlet diameter(cm)	1.3

2.3 Computational Grids

A grid independence test was performed to establish the optimal mesh density. The grid independence test was performed only to the first geometry with the shortest length of 50 cm pipe length and outlet diameter of 0.9 cm to find the optimal nodes to be applied for following two geometries.

3. Results
3.1 Viscosity Variation and Shear Rate Profiles

Published data by Cao *et al.,* [11] stated that Carreau-Yasuda model demonstrated shear thinning behaviour with visibly increased viscosity at increasing shear rates, especially near the outlet walls of converging tubes. These trends conform with the experimental evidence of the decrease in viscosity with the non-Newtonian fluids. The CFD results reveal the same effects of shear thinning in all cases. Contours derived from the analysis show areas of low viscosity near walls but where shear rates are expected to be high. Such findings are in concordance to the theoretical profiles illustrated in Figure 1 in Cao's work.

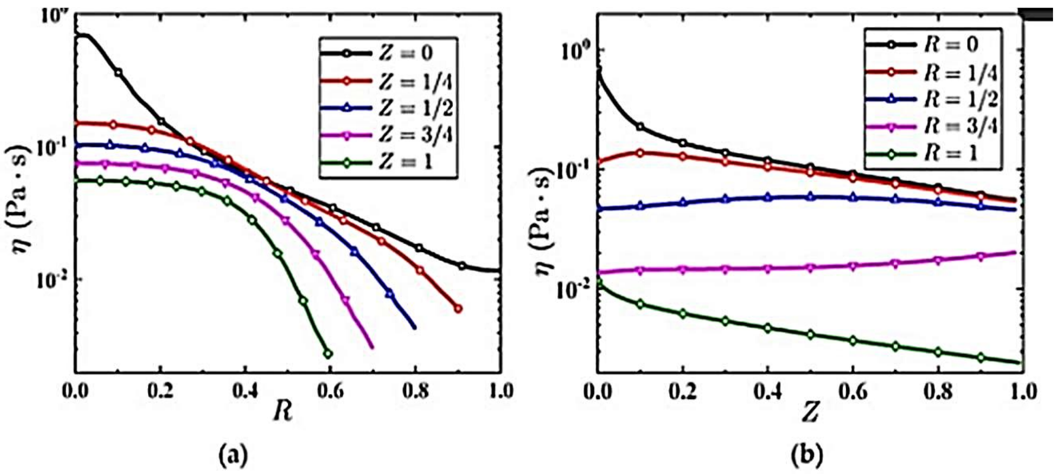

Fig. 1. Reference figure from published paper from Cao *et al.,* [11]

The velocity profiles and pressure distributions obtained in the CFD simulations were contrasted to data in literature of internal flow in comparable tapered designs. It produces similar trends to what has been observed in earlier studies including the linear velocity rise along tapered sections as caused by effects of continuity besides geometry. This acceleration behavior has been evidenced earlier by Jing *et al.,* [14] when they compared with similar kind of geometries in turbulent flows for the same range of Reynolds numbers. In this study, the standard k-ε model was adopted for the simulations and analysis of the flow field because of its applicability to internal flow problems while offering good accuracy and moderate computational cost. Launder *et al.,* [15] affirm the utility of this type of model in modeling turbulent flows in pipes. Nevertheless, slight variation in velocity close to the outlet implies that higher-order turbulence models such as recalculated k-ω SST may be useful in subsequent investigations. These flow geometries were found to have lower velocity development compared with that reported in other published studies because the peak flow regions at the bottom near the outlet section is underdeveloped. Velocity and pressure profiles for the 100 cm pipe were found to be in close agreement with theoretical and experimental data due to fully developed flow conditions of the pipes.

3.2 Grid Independence Test

The first geometry performs the grid independence test, for the next two geometries the repetition is done by changing the element size and mesh to calculate the exact value of the nodes. The Grid Independence Test confirms that the obtained data of CFD simulation (for instance, velocity, pressure distribution) does not considerably depend on the selected mesh density. By refining the grid, you verify whether refining the grid more will yield insignificant changes to the results. The grid is improved through altering the element sizes at the mesh, which, with decreasing steps, greatly increases the number of nodes. Table 3 portrays the values of nodes obtained by duplicating the setting and decreasing the element size.

Table 3
The value of geometry using 50 cm pipe length

Sample	Length (cm)	External diameter (cm)	Nodes	Element size (cm)
1-1			27936	0.2
1-2			33246	0.19
1-3	50	0.9	42752	0.16
1-4			54824	0.14
1-5			63910	0.13

It makes certain that the outcome of CFD simulation (like velocity pressure distribution) is not influenced by mesh density through GIT. With the help of refining the proposed grid, it is possible to analyze how significant increase in node count led to negligible changes can be made. are not significantly affected by the mesh resolution. By refining the grid, the analysis of increase in node count led to negligible changes in the results can be done. The given curves represent the velocity profiles versus node numbers, with velocities increasing with node numbers Since the optimal value of node numbers is unknown, one attempts to identify the interval within which the velocity profiles essentially coincide: the interval (54824, 63910). As the mesh is refined further from 54824 nodes, there is a minor variation in results and both the profiles are almost constant as the mesh size reaches 63910 nodes. Figure 2 and Figure 3 portrays the velocity and pressure chart obtained respectively by comparison due to the different element sizes.

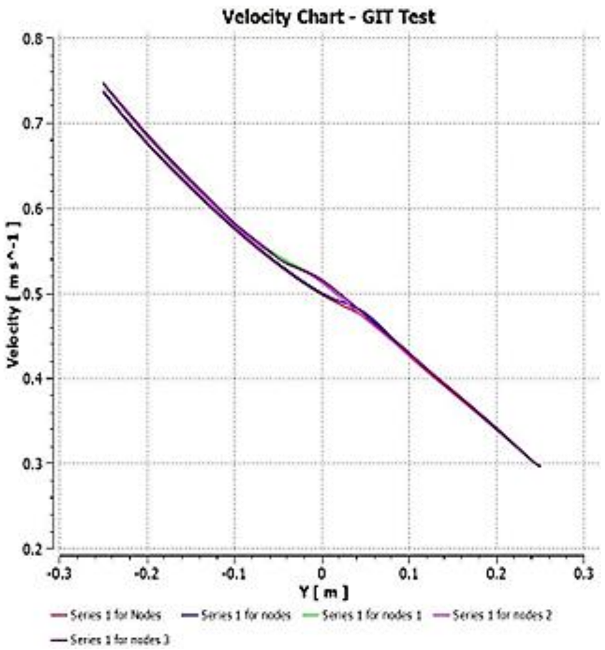

Fig. 2. Velocity chart for GIT test

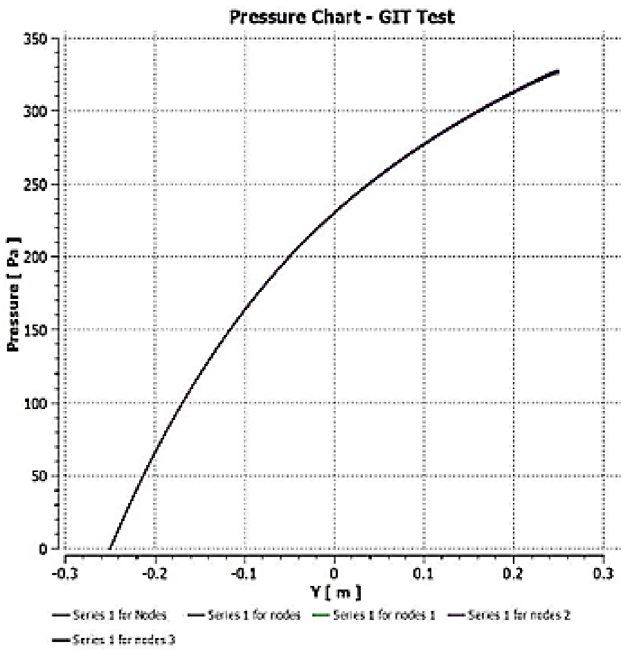

Fig. 3. Pressure chart for GIT test

This suggests that the results have become grid independent within this range, therefore the solution does not depend on the grid density. Hence, it is obtained that any additional number of nodes increasing (above 63910) of this network will make it more complex in terms of computation but adds almost negligible value to improve the accuracy. Shorter computation times with fewer nodes leads to lower, or in some cases, very inaccurate solutions. Additional nodes are exhibited to improve accuracy while at the same time profoundly escalating computational complexity and simulation time. The node ranges of 54824 to 63910 have been chosen to meet these concerns without causing too much additional computation. Starting from the depictions based on node counts, based on the velocity chart where the velocities cross each other we obtain the data in the range of 54824 and 63910 nodes. As we considered number of nodes larger than 54824, there is not much fluctuations noticed in the results and the profile is almost same as the mesh when we have considered number of nodes up to 63910. This shows that the results depend little on the grid used in the calculations within this range, a situation in which the solutions have become grid independent. Hence, a rise in node counts by another increment in variance (greater than 63910) incurs higher computational cost yielding slight improvement in accuracy.

3.3 Analysis Beyond the Physical Pipe Length

In the case of the 50 cm and 75 cm pipes the observations at 100 cm are actually outside the pipe length, and the observed data is extrapolated. This imposes artificial results since no natural flow occurs outside the pipe's envelope. It is important that the flow can fully develop for the entire length of the 100 cm pipe so that the velocity and pressure become stabilized at 100 cm. The flow in pipes that are short, say 50 cm and 75 cm, has less distance to stabilize and so at the extrapolated 100 cm, the profiles are underdeveloped. This leads to reduced velocities and pressures than those registered with by the 100 cm pipe. Velocity: Due to the nature of the pipes as being shorter, the development of the boundary layer is not fully developed as extrapolated to a 100 cm position. Thus, flow velocity is less as noted less power is transferred from the flow inlet to the output. The nature of flow causes pressure drop in the process as the flow occurs throughout the pipe. Since the pipes length are smaller and the analysis goes beyond the pipe length the pressure values obtained are lower and less precise than the ideal fully developed 100cm pipe pressure drop values. Figures 4 and 5 shows the velocity and pressure difference with the variations of geometry.

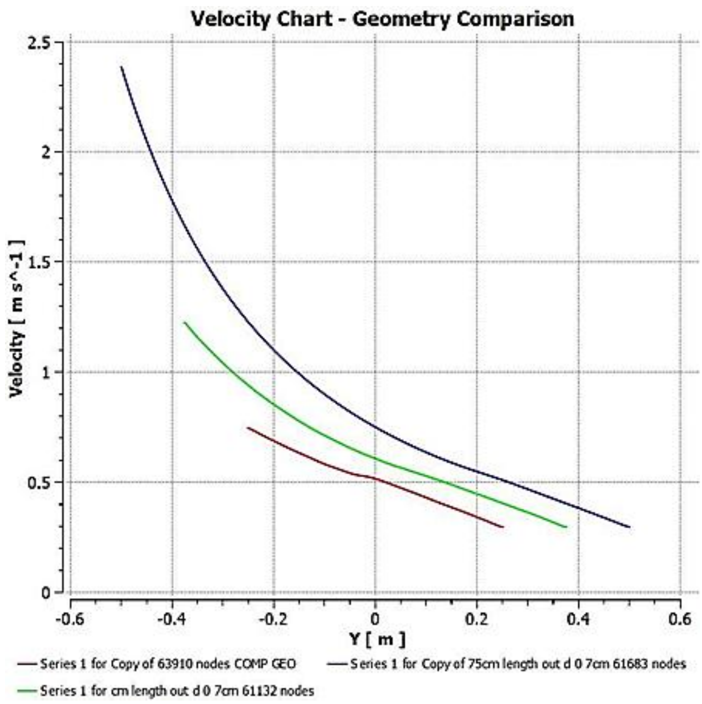

Fig. 4. Velocity chart comparing pipe length of 50 cm, 75 cm and 100 cm

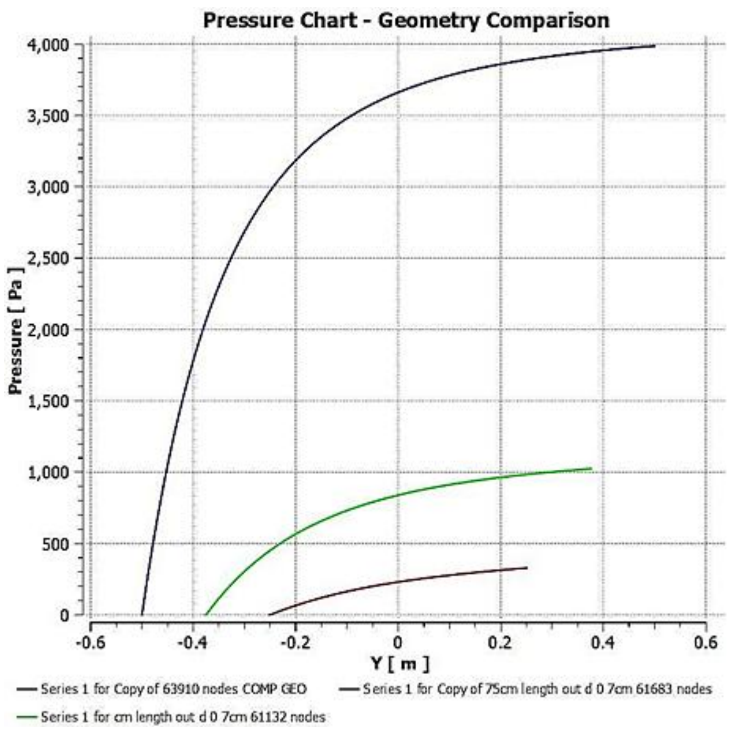

Fig. 5 Pressure chart comparing pipe length of 50 cm, 75 cm and 100 cm

3.4 Velocity Contour Comparison for Different Geometries

The analysis focuses on the velocity distribution in tapered pipes of varying lengths (50 cm, 75 cm, and 100 cm) using color gradients in contour plots, ranging from blue (low velocity) to red (high velocity). In the 50 cm pipe, the shorter length limits flow development, resulting in predominantly low velocities (blue and cyan) (Figure 6). The 75 cm pipe shows a broader range of velocities, with colors shifting towards green as the increased length allows for greater acceleration (Figure 7). The 100 cm pipe exhibits the highest velocity dispersion, transitioning smoothly from blue at the inlet to red at the outlet, as the longer pipe provides more space for acceleration (Figure 8).

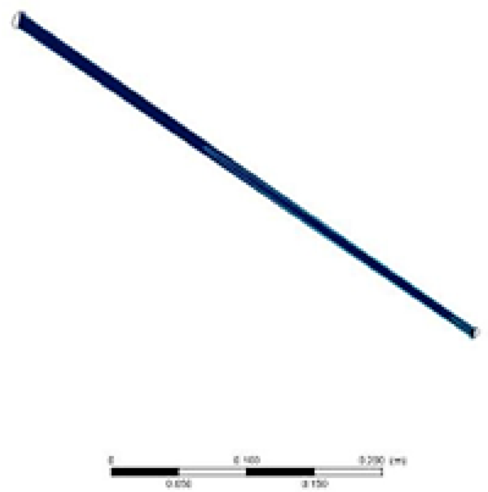

Fig. 6. Velocity contour for pipe length 50 cm

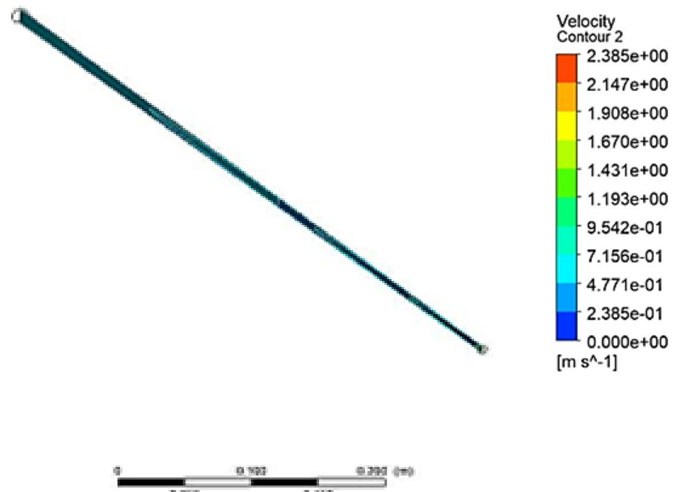

Fig. 7. Velocity contour for pipe length 75 cm

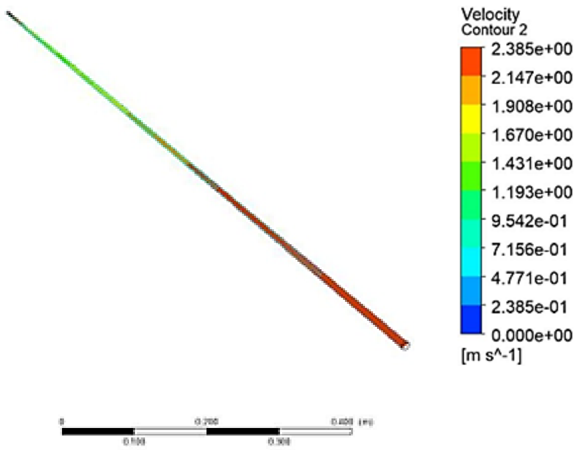

Fig. 8. Velocity contour for pipe length 100 cm

Furthermore, the reduction in cross-sectional area downstream, in line with the principle of mass conservation, leads to an increase in velocity, depicted by the progression of colors along the pipe length. Despite variations in pipe geometry, all contours consistently represent the same physical process of flow acceleration in the converging section, highlighting how pipe length influences velocity profiles. Together, the contours effectively illustrate the relationship between geometry, flow dynamics, and velocity progression.

4. Conclusions

The study successfully characterized flow patterns in tapered pipes, conducted a Grid Independence Test (GIT) for optimal meshing, and investigated the effect of element size on pressure and velocity fields. Results showed that the pipe length significantly influences flow characteristics, with longer pipes yielding extended velocity and pressure profiles. Reducing the external diameter from 0.9 cm to 0.5 cm, while maintaining a constant internal diameter of 1.3 cm, caused notable changes in flow behavior. The findings aligned with theoretical and past studies, showing non-linear pressure losses and velocity increases in tapered regions. The GIT identified an optimal node range (54,824 to 63,910) that balanced numerical precision and computational efficiency. The k-ε turbulence model effectively analyzed turbulent flow but showed deviations near the outlet, suggesting that models like k-ω SST could improve accuracy. Overall, the study highlights the impact of geometry and mesh density on flow characteristics and offers guidelines for designing tapered pipes, with potential extensions involving higher-order geometries and advanced turbulence models for industrial applications.

References

[1] Mund, Chinmaya, Sushil Kumar Rathore, and Ranjit Kumar Sahoo. "A review of solar air collectors about various modifications for performance enhancement." *Solar Energy* 228 (2021): 140-167. https://doi.org/10.1016/j.solener.2021.08.040

[2] Leschziner, M. A. "Modelling turbulent separated flow in the context of aerodynamic applications." *Fluid Dynamics Research* 38, no. 2-3 (2006): 174-210. https://doi.org/10.1016/j.fluiddyn.2004.11.004

[3] TrongBui, Bogdan-AlexandruBelega. "CFD analysis of flow in convergent-divergent nozzle." In *International Conference of Scientific Paper AFASES*. 2015.

[4] Patel, Malay S., Sulochan D. Mane, and Manikant Raman. "Concepts and CFD analysis of De-Laval nozzle." *International Journal of Mechanical Engineering and Technology* 7, no. 5 (2016): 221-240.

[5] Sudhakar, B. V. V. N., B. Purna Chandra Sekhar, P. Narendra Mohan, and Md Touseef Ahmad. "Modeling and simulation of convergent-divergent nozzle using computational fluid dynamics." *International Research Journal of Engineering and Technology* 3, no. 08 (2016): 346-350.

[6] Tominaga, Yoshihide. "CFD simulations of turbulent flow and dispersion in built environment: A perspective review." *Journal of Wind Engineering and Industrial Aerodynamics* 249 (2024): 105741. https://doi.org/10.1016/j.jweia.2024.105741

[7] Mehl, Steffen, Mary C. Hill, and Stanley A. Leake. "Comparison of local grid refinement methods for MODFLOW." *Groundwater* 44, no. 6 (2006): 792-79. https://doi.org/10.1111/j.1745-6584.2006.00192.x

[8] Debtera, Baru, Venkatesa Prabhu Sundramurthy, and Ibsa Neme. "Computational fluid dynamics simulation and analysis of fluid flow in pipe: Effect of fluid viscosity." *Journal of Computational and Theoretical Nanoscience* 18, no. 3 (2021): 805-810. https://doi.org/10.2139/ssrn.4201717

[9] Dang Le, Quang, Riccardo Mereu, Giorgio Besagni, Vincenzo Dossena, and Fabio Inzoli. "Computational fluid dynamics modeling of flashing flow in convergent-divergent nozzle." *Journal of Fluids Engineering* 140, no. 10 (2018): 101102. https://doi.org/10.1115/1.4039908

[10] Antepara, Oscar, O. Lehmkuhl, R. Borrell, J. Chiva, and A. Oliva. "Parallel adaptive mesh refinement for large-eddy simulations of turbulent flows." *Computers & Fluids* 110 (2015): 48-61. https://doi.org/10.1016/j.compfluid.2014.09.050

[11] Cao, Qin-Liu, Mehrdad Massoudi, Wen-He Liao, Feng Feng, and Wei-Tao Wu. "Flow characteristics of water-HPC gel in converging tubes and tapered injectors." *Energies* 12, no. 9 (2019): 1643. https://doi.org/10.3390/en12091643

[12] Tafti, Danesh K., Long He, and K. Nagendra. "Large eddy simulation for predicting turbulent heat transfer in gas turbines." *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences* 372, no. 2022 (2014): 20130322. https://doi.org/10.1098/rsta.2013.0322

[13] Gutiérrez Suárez, Jairo Andrés, Carlos Humberto Galeano Urueña, and Alexánder Gómez Mejía. "Adaptive mesh refinement strategies for cost-effective eddy-resolving transient simulations of spray dryers." *ChemEngineering* 7, no. 5 (2023): 100. https://doi.org/10.3390/chemengineering7050100

[14] Jing, Hongmiao, Jitao Zhang, Mengjiao Han, Weikang Li, Wanru Zhao, and Qingkuan Liu. "Numerical study on the aerodynamic characteristics and flow field of two tandem tapered square cylinders with different spacing ratios." *Ocean Engineering* 295 (2024): 116875. https://doi.org/10.1016/j.oceaneng.2024.116875

[15] Launder, Brian Edward, and Dudley Brian Spalding. "The numerical computation of turbulent flows." In *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, p. 96-116. Pergamon, 1983. https://doi.org/10.1016/0045-7825(74)90029-2