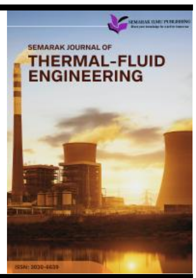
Semarak Journal of Thermal-Fluid Engineering

Journal homepage: https://semarakilmu.my/index.php/sjotfe/index
ISSN: 3030-6639



Simulation of Turbulent Flow in Helical Coil Pipe

Muhammad Danish Farhan Rosli@Rosdi[1*]

[1] Department of Mechanical and Manufacturing Engineering, Fakulti Kejuruteraan Mekanikal, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja. Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 2 June 2025
Received in revised form 9 July 2025
Accepted 8 August 2025
Available online 29 September 2025

Keywords:
CFD simulation; helical coil pipe; turbulent kinetic energy; pressure drop; velocity profile; turbulence intensity; flow behavior; centrifugal effects; dean vortices; curved pipe flow

ABSTRACT

Many systems involving heat or mass transfer use helical coil pipes due to their small size and strong transfer properties. Because their shape is curved, the flow inside the pipe changes, so analysing accurately is crucial for achieving good performance. However, it is still difficult to understand how turbulent flow relates to the diameter of the pipes under different conditions. The goal of this study is to investigate the relationship between the diameter of a coiled pipe and how turbulent flow develops in terms of pressure, how velocity changes and how turbulent conditions occur. In SolidWorks, 2 helical coils were created to have the same coil pitch and number of revolutions (15 mm and 10 mm). Standard k–ε turbulence model in steady-state, incompressible flow was used in ANSYS Fluent to conduct CFD simulations. A grid independence analysis was conducted to check if the mesh was stable. The process was modelled using input speeds of 0.297 m/s, 0.397 m/s and 0.497 m/s for the inlet. It was found that the pipe with a smaller diameter led to an increase in pressure drops due to the added forces from both the walls and the centrifugal force. Tests showed that the flow's velocity is high in the middle and less in the wall regions. Turbulence increased in areas where the flow was close to bends, though Dean vortices did not develop at the specified conditions. Ultimately, the investigation revealed that the size of the pipes has a big impact on the turbulence found inside the coils. Thanks to these findings, fluid system engineers have better control over their coils in terms of pressure and equal flow distribution.

1. Introduction

Curved geometries such as helical coil pipes are an interest in engineering and scientific application, especially in internal flow dynamics and such, internal flow dynamics are widely applicable including in chemical processing, power generation and heat exchange systems. Sigalotti *et al.,* [1] said that helically coiled pipes, due to their lightweight and certain compact design and enhanced heat transfer properties are suitable for industrial applications. Secondary flow is attributed to the most notable feature of flow helical pipe where centrifugal force effect velocity profiles and heat transfer characteristics [1,2]. Research by Das *et al.*, [3] found that helical coil flow

* *Corresponding author.*
E-mail address: Muhammad.danish.farhan@gmail.com

https://doi.org/10.37934/sjotfe.6.1.1527a

is different from plain pipe flow as it has a secondary flow pattern. Some researchers have focused on predicting velocity and temperature fields, friction factors, and heat transfer coefficients in both laminar and turbulent flow conditions [4].

Helical coils undergo an additional complexity to the flow field with the introduction of centrifugal forces, which ultimately create secondary flows or Dean vortices. This has been reviewed by several authors such as Datta *et al.,* [5] and Tang *et al.,* [6]. The secondary flows cause noticeable changes of velocity and pressure distributions, as well as altering heat and mass transfer rates and pressure losses. Understanding how changes in coil pipe sizes influence the turbulence in these coils is still something that experts find hard to clarify. It is difficult to accurately calculate the effects of pressure loss, separated flow and Dean vortices, as these issues are caused by both the shape of the body and turbulence, especially when using straightened turbulence models. This study therefore aims to understand how pressure and turbulence in helical coil pipes are affected by pipe diameter by running CFD simulations.

Because of helical coil configuration, the curvature of the pipe causes an imbalance between centrifugal and inertial forces, and the secondary circulations are superimposed on primary axial flow. The strength of these secondary flows also depends very strongly on geometric factors such as the coil diameter, the pipe diameter, the coil pitch and the flow parameters such as Reynolds and Dean numbers, these also have been reviewed by authors Cheng *et al.,* [7] and Faraj *et al.,* [8]. Based on Tang *et al*., [6] as the coil radius decreases, as it tends to tighten its coils, the centrifugal effects become stronger, increasing Dean vortex strength and pressure drops, and as the diameters become larger, the centrifugal effects are weaker.

Research by Kováts *et al*., [9] said that typically, in the turbulent flow regimes, where the Reynolds numbers exceed 4000, the secondary flows add to the complicating of the velocity profile. It has been shown that a pair of counter-rotating Dean vortices form a stable pair in the range of transitional Reynolds numbers (~140–220) obtained after approximately half a turn of the coil. The vortices accelerate the maximum velocity towards the outer wall of the coil, which dramatically modifies the axial velocity profile when a coil is used as opposed to straight pipe flow based on Springer *et al*., [10]. Datta *et al.,* [5] said that secondary flow structures continue to be influential but, as the flow further becomes fully turbulent, the momentum transfer is dominated by turbulent eddy and classical vortex patterns may be obscured by momentum transfer. It was said that the transition to turbulence in a straight pipe downstream of a helical coil, is Reynolds number dependent and the transition point moves upstream as Reynolds number increases by Hon *et al.,* [11].

Helical coil pressure loss characteristics have been widely studied both by CFD simulations and experiments. Cheng *et al.,* [7] have observed that, for the coil Darcy friction factor, the radius ratio of the coil is directly related and inversely correlated to Reynolds number. An increase in the coil diameter decreases the pressure loss per unit length because there is a weakening of secondary flow intensity [6]. Also, Faraj *et al.,* [8] showed that bifurcation of flow behavior is influenced by coil pitch, while it is more affected by curvature (Dean number).

CFD techniques have been essential tools used for simulating such complex flow fields. In helical coils, numerous studies have been carried out using standard k-ε and SST k-ω turbulence models to describe the turbulent structures among them is by Nashine and Thokchom [12]. Hasabnis and Vivek [13] however do research on helical coils combined with pipe pinching have shown potential for increasing heat transfer, although the reduced helical coils outperform normal helical coils and provide up 10-20% greater performance in turbulent flow. For accurate modeling, constant temperature or heat flux boundary conditions may produce wrong results and conjugate heat transfer and temperature dependent fluid properties are of high importance [14].

An important consideration as well is flowing stability. Even though the secondary vortices are relatively stable in transitional regimes, flow instability leads to unsteady and complex behaviours in fully developed turbulence, studied by Tang *et al.,* [15]. Based on Ciofalo *et al.,* [16] the secondary flow is strengthened by increasing curvature, it increases friction coefficient and Nusselt number, and it alters the turbulence characteristics.

Grid independence test (GIT) must be done for reliable numerical prediction in order that mesh size does not affect on the results. Computational fluid dynamics (CFD) research on grid independence tests has indicated the value of having an optimal grid for good results. In their work, Lee *et al.,* [17] proposed an improved method that depends on grid resolution and characteristic length to ascertain optimal grid conditions. In microgrid networks, reliability assessment of distribution network is inherent to many aspects; therefore, this requires new reliability assessment techniques and real test systems, as stated by Lopez-Prado *et al.,* [18]. Berrett and Richard [19] introduced the U statistic permutation (USP) test, which has better control of test size and power than such standard methods as Pearson's chi squared and G test for statistical independence testing. Bjerager *et al.,* [20] carry out a meta-analysis of the Amsler grid test in medical diagnostics of neovascular age-related macular degeneration suggesting moderate sensitivity (67-71%), variable specificity (63-99%) based on the control group limiting its use in patient monitoring.

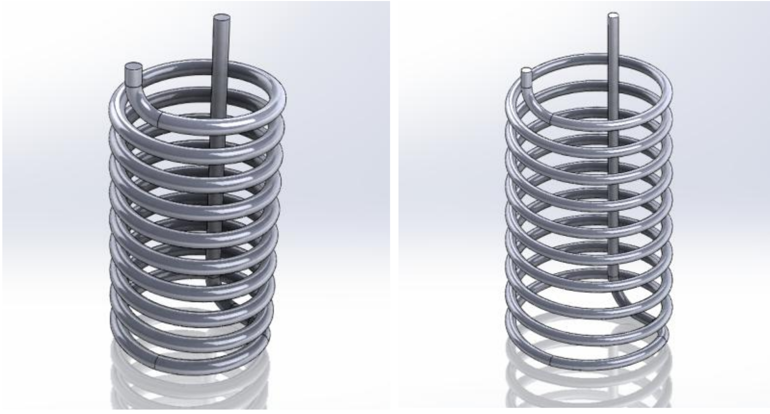
2. Methodology

2.1 Software

SolidWorks from Dassault Systèmes used as 3D software to model the geometry for this analysis. Many mechanical engineers use SolidWorks because it is easy to use and allows powerful parametric modeling. In this work, SolidWorks was used to design 3D models of helical coil pipes by choosing values for their diameter, coil pitch and the number of turns. Their use was important due to the need to consistently and correctly display the object in the simulations. All completed models were converted to STEP (.step) and IGES (.iges) files to make sure they can be used in the simulation program. ANSYS Fluent, a primary product developed by ANSYS Inc., a company that Dr. Swanson formed in 1970, was used to perform the Computational Fluid Dynamics (CFD) analysis. Many people consider ANSYS Fluent to be effective due to its advanced solver and precise results in simulations of complex fluids, turbulence and heating processes. Investigating the turbulent flow behavior in helical coils was made reliable and simple by using SolidWorks for modeling and ANSYS Fluent for analysis.

2.2 Geometry Modeling

Two helical coil pipe geometries shown in Figure 1 were designed using CAD software with pipe diameters chosen to examine how pipe diameters affect the turbulent flow behavior. The structures of the two models were identical, with the same coil pitch of 30 mm and 9 complete revolutions from 0° angular orientation. This produced a total coil height of 270 mm for each model. The two designs differed in the pipe diameter, the first model had an internal diameter of 15 mm which will be called geometry A, while the second was reduced to 10 mm also called as geometry B. To keep the curvature and spacing constant, the coils were built with a constant pitch on a circular helical path. Fully developed flow condition and entrance effects were eliminated through addition of straight inlet and outlet sections to both geometries. They were then imported into ANSYS Fluent for importing and meshing for computational analysis.



(a) (b)

Fig. 1. Helical coil pipe geometry for A and B

2.3 Meshing

The helical pipe geometries were input into the meshing tool in ANSYS Workbench 2023 R1 and the fluid domain extracted from those was discretized. Being the helically shaped coil, the mesh was also unstructured tetrahedral, owing to its complex curved geometry, as mesh will be flexible in capturing the flow within helical turns. Figure 2 shows unstructured tetrahedral meshing on the geometry.

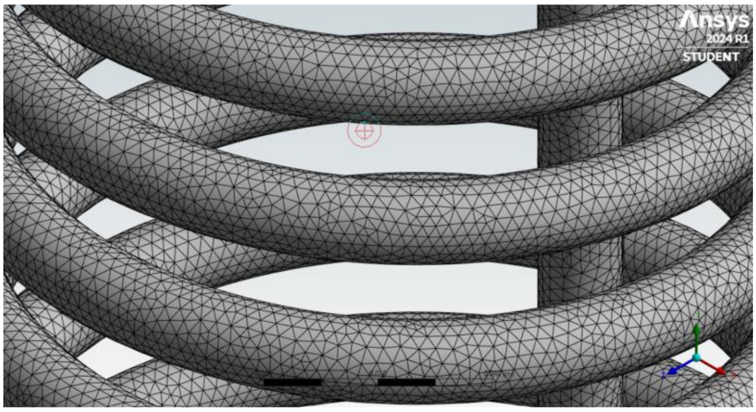


Fig. 2. Unstructured tetrahedral meshing on the geometry

2.3.1 Grid independence test

A grid independence test is carried out in this CFD project to guarantee that simulation accuracy is not influenced by the mesh size. A range of meshes are formed and simulations are conducted with comparable boundaries and parameters for each one. Measuring velocity, pressure and flow rate at set places is done using key output parameters. The results are checked to locate the mesh size after which improvements become minor which means the simulation is grid independent. The best choice of mesh gives you accurate results without taking too much computing time.

2.4 Governing Equations

Fluid mechanics and thermodynamics are the ones governing the fluid flow and heat transfer in a helical coil pipe. Momentum, mass and energy conservation are represented by the Navier-Stokes equations. The fluid assumed in this study is under steady state turbulent flow, incompressible, and Newtonian. Computational Fluid Dynamics (CFD) is used to solve the governing equations using ANSYS Fluent. The equations are based on research by Launder and Spalding [21]. The continuity equation (mass conservation) is given by Eq. (1). While, the Navier-Stokes equations (momentum conservation) are expressed as in Eq. (2):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0 \quad (1)$$

$$\rho \left(\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right) = -\nabla p + \mu \nabla^2 \vec{v} + \vec{F} \quad (2)$$

where ρ is fluid density, $\vec{v}$ is the velocity vector, p is pressure, μ is dynamic viscosity, $\vec{F}$ represent body forces (if any).

The flow in the helical coil is operated at high Reynolds number so it is assumed to be fully turbulent. This study uses the standard k–ε model since it is proven accurate and computationally efficient for internal flow applications. In addition, this model solves two further transport equations (for turbulent kinetic energy, k and its dissipation rate, ε) for turbulence's impact on the flow. In most engineering applications, the k–ε model does well in predicting such secondary flows and enhanced mixing induced by helical geometries. The model is implemented in the form of appropriate near wall treatment in ANSYS Fluent to yield accurate representation of the wall bounded turbulence.

2.5 Boundary Condition and Parameter Selection
2.5.1 Inlet velocity

Three different inlet velocities were applied to both geometries to observe the effects on variations in flow characteristics for this study. For this, the selected velocities were 0.297 m/s, 0.397 m/s and 0.497 m/s. These values were chosen such that the largest helical coil pipe (1.50 cm in diameter) would operate with a Reynolds number of about 4338 at the maximum velocity. Uniform velocity was used at the pipe entrance, this being a velocity inlet boundary condition. The flow settings permitted direct comparison between the helical coil and the straight pipe (10 cm diameter) based on equal flow conditions.

2.5.2 Outlet pressure

At the exit of both geometries, a pressure outlet boundary condition was applied. The flow was free exit hence the outlet pressure was set to 0 Pa gauge pressure. This is an ideal setting that makes the simulation behave like a fully developed flow condition at the outlet and stabilizes solution numerically.

2.5.3 Wall conditions

By treating all the pipe walls as no-slip boundary and stationary wall, the fluid velocity at the wall surface is assumed to be zero. This is a realistic behavior of internal flow in enclosed pipes.

2.5.4 Working fluid

For both cases the working fluid was water at room temperature. The fluid was assumed to be incompressible and Newtonian. The simulations used the following physical properties.

i. Density (ρ) = 997 kg/m³
ii. Dynamic viscosity (μ) = 0.001003 Pa·s

These constant properties had been applied to all velocity cases and geometries.

2.5.5 Operating conditions

The flow was assumed to be incompressible, steady state and turbulent. The standard k–ε model was used to model the turbulence, and its use in simulating internal pipe flow is effective. It results in a reasonable tradeoff between computational cost and accuracy and is applicable to capture generalities of turbulence without complicated modeling.

2.6 Analysis
2.6.1 Flow separation

Fluid separation refers to the case where fluid on the pipe wall is detached (detaches) from the wall due to the existence of an adverse pressure gradient. This phenomenon is more likely to happen in helical coil pipes at bends where the flow makes sharp turns. However, recirculation zones follow from the resulting detached boundary layer and may incur dead spots and low efficiency of the system. It is important to be able to identify these flow separation zones as they represent places where the velocity of the fluid dramatically decreases. As these regions are measured by pressure contours, it is possible to see the adverse pressure gradients producing the flow separation. The degree of separation varies depending upon the bend radius, pipe diameter and flow rate in helical pipes.

2.6.2 Secondary flow structures (dean vortices)

Centrifugal forces acting on the fluid in curved pipes like helical coils result in formation of secondary flow structures, especially Dean vortices. A counter-rotating flow pattern which emerges in the bend of the pipe, is called a vortex, redistributing the fluid flow, and these vortices are used in any bend in the pipe. Uneven flow patterns associated with Dean vortices are expected to be present, depending on the degree of their presence, in the velocity distribution across the pipe's cross section. In some cases, the vortices can promote mixing in the pipe and in some cases contribute to flow instability. The vortices can then be identified using the use of vorticity plots and streamline diagrams to visualize the counter rotating flow patterns and how they influence the bulk fluid dynamics inside of the coil.

2.6.3 Pressure distribution

It is essential to understand the fluid's behavior in the pipe to understand the pressure distribution along the helical coil. Factors like friction, curvature, as well influx the pressure drop that is caused in the coil. The sharp bends in the coil can cause a significant loss in pressure, thus affecting the flow efficiency as well as the overall system performance. Stabilization may also be referred to as pressure recovery, where the pressure generally increases after a bend. Pressure contours over length of the coil can be analyzed around the bends to see where there is high loss of pressure. If these are high loss regions, there might be a design modification needed to them that may help include pipe increase diameter or change the bend angles to help reduce the pressure drop.

2.6.4 Velocity profiles

The velocity profiles give information about how the fluid moves through the pipe, especially near unto bends where the centrifugal forces involve the fluid into behaving otherwise in the straight sections. In helical pipe, the velocity is usually faster at the center of the pipe and slower towards the walls because of curvature. This variation in velocity in the bends is more pronounced and it can result in non-uniform flow patterns. Die velocity profiles in different points in the coil allow the distribution of the flow to be analyzed, especially in regions where the velocity distribution can be affected by secondary flows (Dean vortices). The velocity at different radial locations along the pipe axis allows to gain valuable information on flow behavior and efficiency.

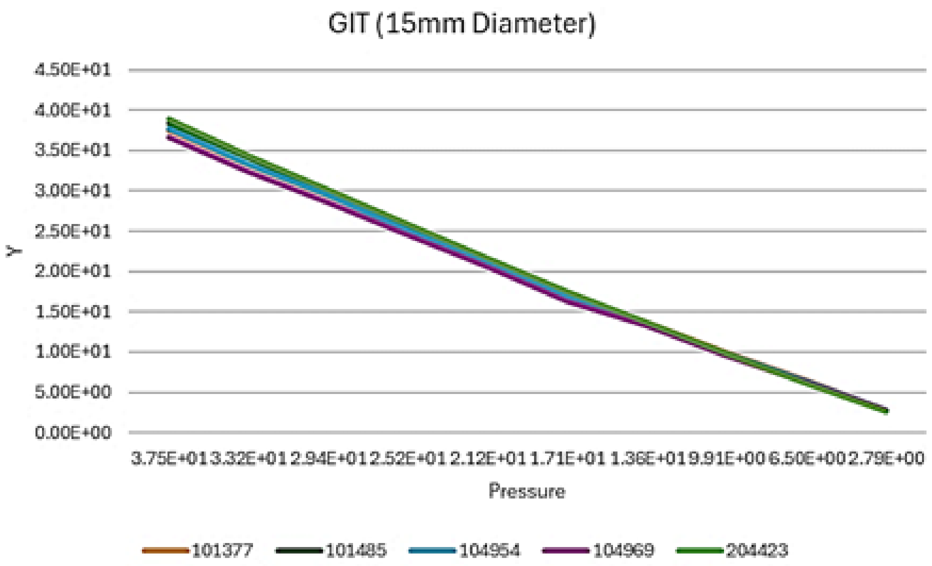
2.6.5 Turbulence intensity

Fluctuation in the velocity field is turbulence intensity, and it is a very important parameter to determine flow stability and mixing efficiency. Bends in the helical coil pipes tend to cause flow to turbulent in the bends due to the directional changes in the fluid. Where the flow is subject to friction and boundary layers, the turbulence intensity is higher further from the pipe walls. Such regions of high turbulence can improve mixing, which might be an asset for, for instance, heat exchange and chemical reactions. This, however, leads to higher friction losses and increase on the pipe wall wear. Contour plots of turbulence intensity and velocity fluctuations help determine the influence of turbulence on the whole flow behavior as well as the recommended design optimization.

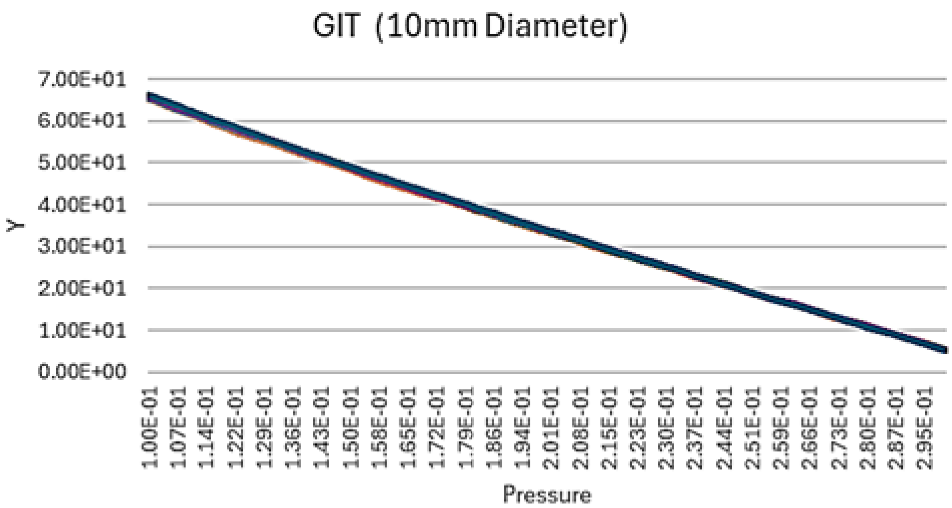
3. Results

3.1 Grid Independence Test

Cell size was varied in a grid independence test. The initial mesh was 20 mm cell, and the mesh was further refined to 3.0 mm and 1.5 mm in critical regions. Pressure at the same flow conditions were used to evaluate the meshing level. The mesh in the final form, with the mesh cell size of 3.0 mm and having a node number of 104969 was determined for stable pressure drop results with less than 5% variation in pressure drop when compared to the smaller mesh cell. The refinement to 1.5 mm was furthered to 204,403 nodes, however this did not significantly decrease pressure, but did increase computation time significantly. For that reason, all simulations were performed using a 3.0 mm mesh with 101315 nodes in both geometries. Figure 3 shows the graph grid independence test for the geometry A and B.



(a)

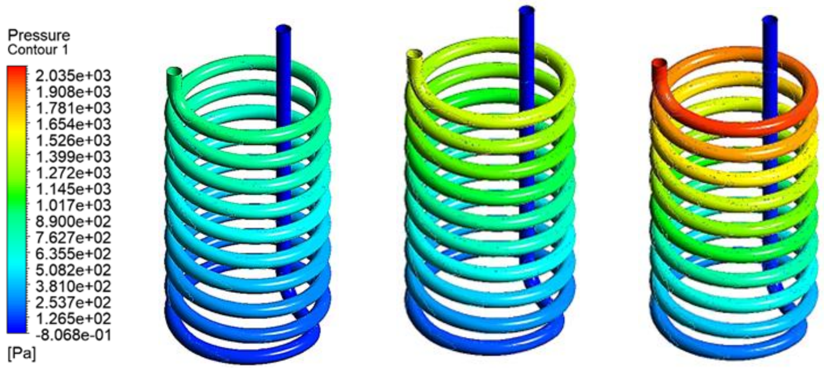


(b)

Fig. 3. GIT (a) Geometry A (b) Geometry B

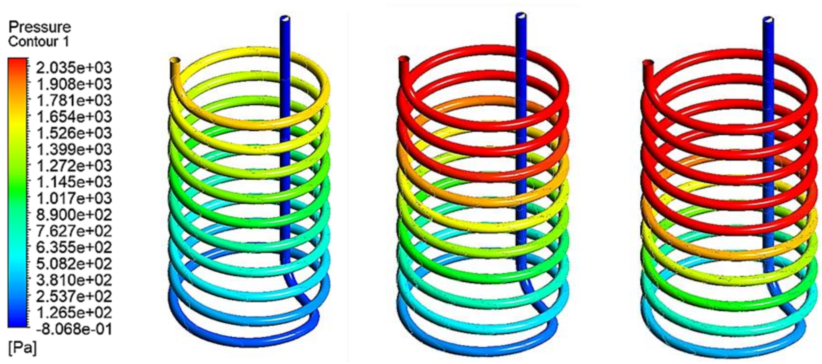
3.2 Pressure Distribution

Pressure distribution along the helical coil pipe was also studied under stationary and no slip wall conditions, and the effect of friction and curvature was seen to drop consistently from inlet to outlet. This pressure loss was also due to wall shear created by the no-slip boundary. The Figures 4 and 5 finds that the pressure is the highest at the inlet and then decreases, suggesting that energy is lost as the flow moves down the pipe. Pressure reduction in the system depends mainly on wall friction, how curved the pipe is and the flow resistance inside the helical coil. Moreover, Geometry A has a better pressure distribution than Geometry B because being larger slows the flow and reduces the friction, leading to less pressure at the tips.



(a) (b) (c)

Fig. 4. Pressure distribution of geometry A for velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

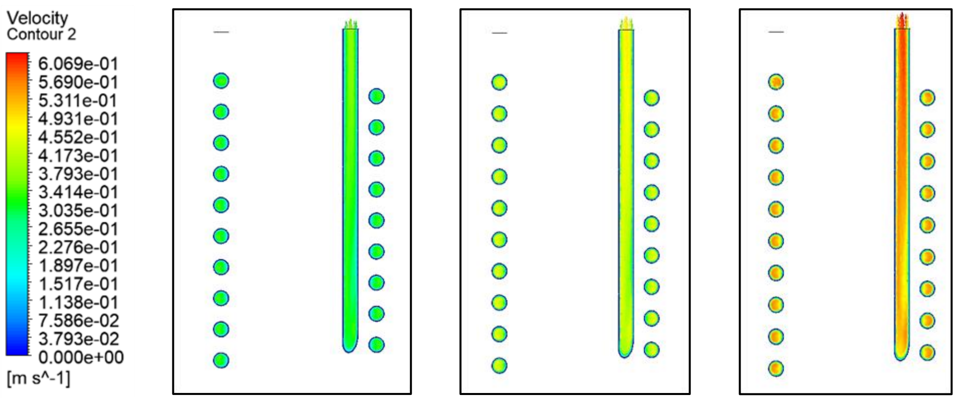


(a) (b) (c)

Fig. 5. Pressure distribution of geometry B for velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

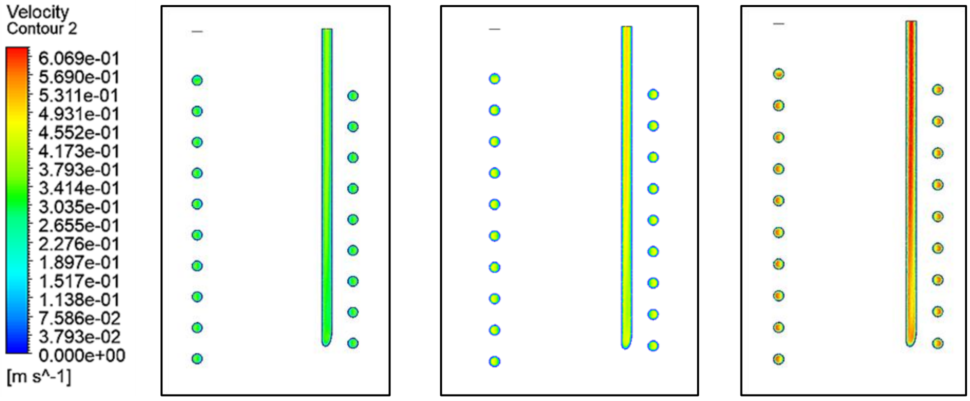
3.3 Velocity Profiles

A peculiarly helical coil pipe was connected to this machine, and the velocity profile in the helical coil pipe showed in Figures 6 and 7. A typical pattern that was influenced by a no-slip wall condition and monotonously decreasing with decreasing velocity from the centre of the pipe towards the wall; Viscous effects that result in this gradient give the fluid at the wall staying stationary to leave a region of shear near the contact.



(a) (b) (c)

Fig. 6. Velocity profile of geometry A for velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

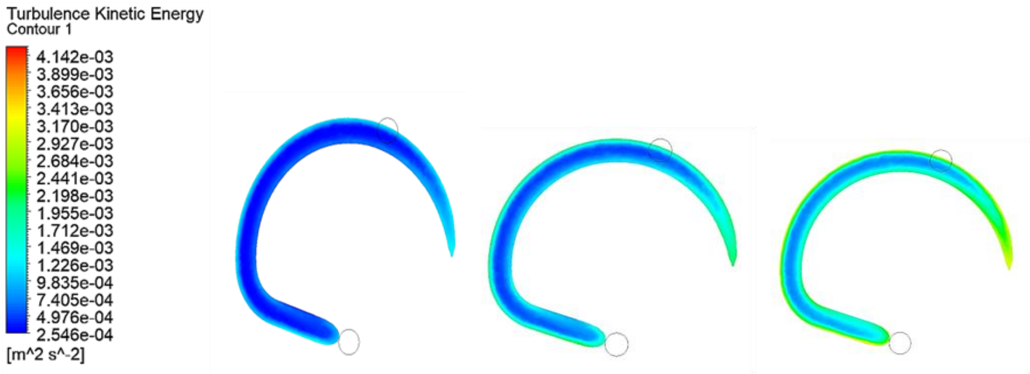


(a) (b) (c)

Fig. 7. Velocity profile of geometry B for velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

3.4 Turbulence Kinetic Energy

At the start, turbulence intensity was examined as a parameter for inlet study, but since limited data was available, turbulence kinetic energy (TKE) was used as the measure for turbulence as shows at Figures 8 and 9. Near the base view in the simulation, TKE is found to be greater at the centre than on the sides, indicating that the most turbulence happens there. Unlike the flux near the centreline, the TKE near the pipe wall is low, showing that turbulence near the wall decreases due to the boundary layer and opposition to flow. This agrees with expected problems, in which the main part of the flow is turbulent, while turbulence slows near the walls, thanks to wall friction. As a result, it appears that the model captures ordinary mixing, but the complex turbulent processes or the formation of dividing layers are not well represented within the presented simulation boundaries.



(a) (b) (c)

Fig. 8. Turbulence kinetic energy of geometry A for velocity (a) 0.297 m/s (b) 0.397 m/s (c) 0.497 m/s

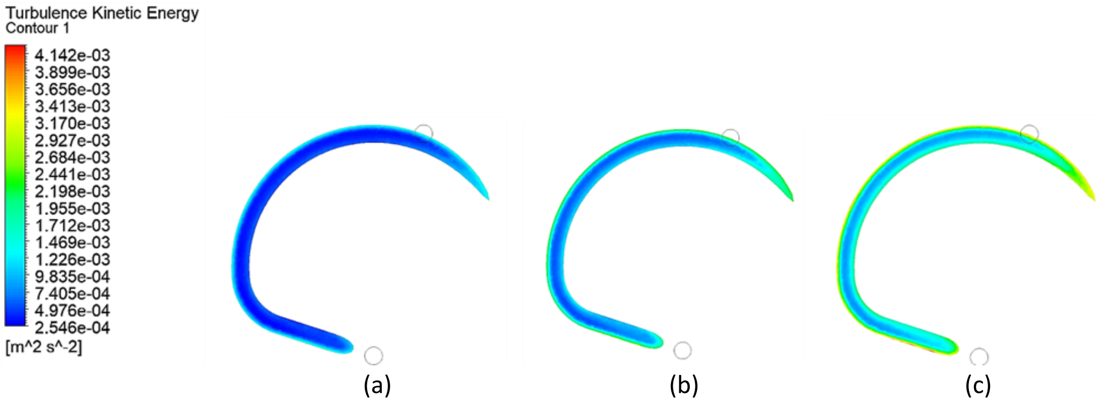


Fig. 9. Turbulence kinetic energy of geometry B for velocity (a) 0.297 m/s, (b) 0.397 m/s (c) 0.497 m/s

4. Conclusions

The influences of curvature on the flow behaviour in helical coil pipe have been effectively simulated to the pressure and velocity distribution within the helical coil pipe. It was found that pressure drop has increased notably in the axial direction of the flow at the bends. Such a behavior leads us to the centrifugal effects due to the curvature that induce an extra resistance to the flow and add to pressure loss in the system. The higher pressure drops across the bends is understood because off the change of direction of flow and of the existence of secondary flows that are common to curved geometries.

In the velocity profile, the fact that higher velocity is present in the centre of the pipe and lower velocity near the pipe walls agrees with general behavior of curved flow domains. The result of this profile suggests the flow is being pushed towards the walls of the coil by centrifugal forces and the centreline flow remains the highest velocity. Such a result is typical for flows in curved pipes, where inertial forces predominate and lead to the radial flow.

Although the simulation did not show the appearance of the usual secondary flows (such as Dean vortices), they are observed in highly curved or turbulent pipe flows. Under the current conditions, however, these phenomena are not seen, typically due to higher curvature or higher Reynolds numbers. Absence of these secondary flows may be indicative of the fact that the level of turbulence and hence Reynolds number in this simulation was not high enough to allow these flow behaviors to develop. However, despite these small deviations from the velocity profile at the bends still indicate the presence of minor curvature effects that slightly alter the flow alignment and direction at these locations.

Overall, the results show some important aspects of the fluid system, whereby low-pressure loss and uniform flow are necessary, the helical coil geometry is significant. The current simulation conditions lead to a relatively stable and laminar flow, but the helical coil provides advantages in providing smooth flow at controlled pressure losses. Although secondary flows and vortices are not induced by any secondary order curvature effects, the sensitivity to curvature effects contributes to its role in flow behavior and is necessary for optimum design for both efficiency and effective flow management.

References

[1] Sigalotti, Leonardo Di G., Carlos E. Alvarado-Rodríguez, and Otto Rendón. "Fluid flow in helically coiled pipes." *Fluids* 8, no. 12 (2023): 308. https://doi.org/10.3390/fluids8120308

[2] Patankar, Suhas V., V. S. Pratap, and D. B. Spalding. "Prediction of laminar flow and heat transfer in helically coiled pipes." In *Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion*, pp. 117-129. Pergamon, 1983. https://doi.org/10.1016/B978-0-08-030937-8.50017-9

[3] Das, Sudip Kumar. "Water flow through helical coils in turbulent condition." In *Advances in engineering Fluid Mechanics: Multiphase Reactor and Polymerization System Hydrodynamics*, pp. 379-403. Gulf Professional Publishing, 1996. https://doi.org/10.1016/B978-088415497-6/50015-2

[4] Manlapaz, Romeo L., and Stuart W. Churchill. "Fully developed laminar flow in a helically coiled tube of finite pitch." *Chemical Engineering Communications* 7, no. 1-3 (1980): 57-78.https://doi.org/10.1080/00986448008912549

[5] Datta, Anup Kumer, Yasutaka Hayamizu, Toshinori Kouchi, Yasunori Nagata, Kyoji Yamamoto, and Shinichiro Yanase. "Numerical study of turbulent helical pipe flow with comparison to the experimental results." *Journal of Fluids Engineering* 139, no. 9 (2017): 091204. https://doi.org/10.1115/1.4036477

[6] Tang, Lingdi, Shouqi Yuan, Mike Malin, and Siva Parameswaran. "Secondary vortex-based analysis of flow characteristics and pressure drop in helically coiled pipe." *Advances in Mechanical Engineering* 9, no. 4 (2017): 1687814017700059. https://doi.org/10.1177/1687814017700059

[7] Cheng, Pengxin, Nan Gui, Xingtuan Yang, Jiyuan Tu, Shengyao Jiang, and Haijun Jia. "A numerical investigation on single-phase flow characteristics and frictional pressure drop in helical pipes." *Fluid Dynamics Research* 52, no. 4 (2020): 045505. https://doi.org/10.1088/1873-7005/ab993f

[8] Faraj, Anwer F., Itimad DJ Azzawi, and Samir G. Yahya. "Pitch variations study on helically coiled pipe in turbulent flow region using CFD." *International Journal of Heat & Technology* 38, no. 4 (2020). https://doi.org/10.18280/ijht.380402

[9] Kováts, Péter, Fabio JWA Martins, Michael Mansour, Dominique Thévenin, and Katharina Zähringer. "Tomographic PIV measurements and RANS simulations of secondary flows inside a horizontally positioned helically coiled tube." *Experiments in Fluids* 61 (2020): 1-15. https://doi.org/10.1007/s00348-020-02950-6

[10] Springer, Fanny, Emilie Carretier, Damien Veyret, and Philippe Moulin. "Developing lengths in woven and helical tubes with dean vortices flows." *Engineering Applications of Computational Fluid Mechanics* 3, no. 1 (2009): 123-134. https://doi.org/10.1080/19942060.2009.11015259

[11] Hon, R., J. A. C. Humphrey, and F. Champagne. "Transition to turbulence of the flow in a straight pipe downstream of a helical coil." *Physics of Fluids* 11, no. 10 (1999): 2993-3002. https://doi.org/10.1063/1.870158

[12] Nashine, Prerana, and Thokchom Subhaschandra Singh. "Effect of dean number on the heat transfer characteristics of a helical coil tube with variable velocity & pressure inlet." *Journal of Thermal Engineering* 6, no. 2 (2020): 128-139. https://doi.org/10.18186/thermal.729149

[13] Hasabnis, Nilesh S., and Vivek V. Ranade. "Flow and heat transfer in a pinched helical coil." *Indian Chemical Engineer* 57, no. 1 (2015): 1-14.https://doi.org/10.1080/00194506.2014.910708

[14] Sunheriya, Neeraj, Jayant Giri, T. Sathish, Rajkumar Chadge, Chetan Mahatme, A. Parthiban, and Praveen Barmavatu. "Evaluation of a helical coil heat exchanger in a forced convection environment." *Interactions* 245, no. 1 (2024): 195. https://doi.org/10.1007/s10751-024-02043-4

[15] Tang, Lingdi, Yue Tang, and Siva Parameswaran. "A numerical study of flow characteristics in a helical pipe." *Advances in Mechanical Engineering* 8, no. 7 (2016): 1687814016660242. https://doi.org/10.1177/1687814016660242

[16] Ciofalo, M., M. Di Liberto, and Gaspare Marotta. "On the influence of curvature and torsion on turbulence in helically coiled pipes." In *Journal of Physics: Conference Series*, vol. 501, no. 1, p. 012025. IOP Publishing, 2014. https://doi.org/10.1088/1742-6596/501/1/012025

[17] Lee, Minhyung, Gwanyong Park, Changyoung Park, and Changmin Kim. "Improvement of grid independence test for computational fluid dynamics model of building based on grid resolution." *Advances in Civil Engineering* 2020, no. 1 (2020): 8827936. https://doi.org/10.1155/2020/8827936

[18] López-Prado, Jose L., Jorge I. Vélez, and Guisselle A. Garcia-Llinás. "Reliability evaluation in distribution networks with microgrids: Review and classification of the literature." *Energies* 13, no. 23 (2020): 6189. https://doi.org/10.1098/rspa.2021.0549

[19] Berrett, Thomas B., and Richard J. Samworth. "USP: an independence test that improves on Pearson's chi-squared and the G-test." *Proceedings of the Royal Society A* 477, no. 2256 (2021): 20210549. https://doi.org/10.1098/rspa.2021.0549

[20] Bjerager, Jakob, Miklos Schneider, Ivan Potapenko, Elon HC van Dijk, Carsten Faber, Jakob Grauslund, Kristina Pfau, Josef Huemer, Danson V. Muttuvelu, Marie L. R. Rasmussen, M. Cem Sabaner, and Yousif Subhi, MD." Diagnostic accuracy of the Amsler grid test for detecting neovascular age-related macular degeneration: a systematic review and meta-analysis." *JAMA ophthalmology* 141, no. 4 (2023): 315-323. https://doi.org/10.1001/jamaophthalmol.2022.6396

[21] Launder, Brian Edward, and Dudley Brian Spalding. "The numerical computation of turbulent flows." In *Numerical prediction of flow, heat transfer, turbulence and combustion*, pp. 96-116. Pergamon, 1983. https://doi.org/10.1016/B978-0-08-030937-8.50016-7