

NUMERICAL ANALYSIS OF TURBULENT FLOW IN A SQUARE DUCT USING THE REYNOLDS-AVERAGED NAVIER-STOKES EQUATIONS WITH $k-\omega$ SST MODEL

Sajad Ali^{1*}, Engr. Attaullah², Awrang Zaib³

¹Lecturer Mathematics, Department of Computer Science, University of Chitral-Pakistan.

²Assistant Director Works, Directorate of Works, University of Chitral-Pakistan.

³Lectuer Computer Science, Department of Computer Science, University of Chitral-Pakistan.

*sahasali143@gmail.com

DOI: <https://doi.org/10.5281/zenodo.17922664>

Keywords:

RANS, Computational Fluid Dynamics, Square Duct Flow, Secondary Flow, Turbulence Modeling, Navier-Stokes Equations, and $k-\omega$ SST

Article History

Received on 12 Nov, 2025

Accepted on 08 Dec 2025

Published on 13 Dec 2025

Copyright @Author

Corresponding Author:

Sajad Ali

Abstract

In computational fluid dynamics (CFD), accurately predicting turbulent flow in non-circular ducts remains a major challenge with significant implications for internal piping networks, heat exchangers, and HVAC systems. This study provides a numerical analysis of fully developed, incompressible turbulent flow in a square duct using the Shear Stress Transport (SST) $k-\omega$ turbulence model in addition to the Reynolds-Averaged Navier-Stokes (RANS) equations. The primary objective was to evaluate how well the model captured secondary flow phenomena, specifically Prandtl's secondary flow of the second kind and the associated anisotropic Reynolds stress distribution caused by turbulence. A hydraulic diameter (D_h) of 0.1 m was used to create a three-dimensional computational. The finite volume method in ANSYS Fluent was used for simulations at a Reynolds number (Re) of 20,000, with enhanced wall treatment for better near-wall resolution. The axial velocity profile results showed a good agreement with known experimental data. More significantly, the unique eight-vortex secondary flow pattern inside the duct cross-section which is impacted by the anisotropy of normal Reynolds stresses was correctly predicted by the simulation. However, when compared to Direct Numerical Simulation (DNS) benchmarks, quantitative variations in the magnitude of these secondary velocities were observed. This work highlights the usefulness of RANS-based approaches for engineering evaluations of internal flows and highlights the persistent difficulties in solving complex turbulence-driven secondary motions due to isotropic eddy-viscosity assumptions.

Introduction

Numerous engineering and scientific fields rely heavily on fluid dynamics, which is essentially controlled by the Navier-Stokes equations (Ali, 2025). Fluid motion is fully described by the Navier-Stokes equations, which mathematically express the conservation of mass, momentum, and energy for a Newtonian fluid (White, 2016). However, direct numerical solution of these equations (DNS) is computationally prohibitive for all but the most basic low Reynolds-number cases for turbulent flows, the most prevalent state in engineering applications. Because of its advantageous balance between accuracy and computational cost, the Reynolds-Averaged Navier-Stokes (RANS) methodology is the most widely used in industrial CFD (Wilcox, 2006). The Navier-Stokes equations and the Boussinesq approximation are fundamental ideas in fluid dynamics that have a wide range of applications in computational and physics domains, such as computer science, engineering, and environmental sciences, especially scientific computing and computational fluid dynamics (CFD) (Ali, 2024). The two standard test cases of flow past a circular and square cylinder correspond to non-fixed separation (separation location depends on the Reynolds number) and fixed separation (separation location does not depend on the Reynolds number) respectively. Unsteady turbulent separation plays a significant role in the flow behavior of these flows at high Reynolds number. (Zang L, 2017)

A canonical yet complex internal flow problem is turbulent flow through ducts of non-circular cross-sections, such as a square duct (Arslan, 2014). Unlike flow in a circular pipe, where the flow field is axisymmetric, turbulent flow in a square duct exhibits a unique three-dimensional characteristic: (Zang, 2022) the generation of secondary flows in the plane perpendicular to the main axial flow direction. These are termed "Prandtl's secondary flows of the second kind" and are generated by spatial gradients of the Reynolds normal stresses, a direct consequence of turbulence anisotropy (Prandtl, 1926). These relatively weak but consequential vortices transport momentum and scalar quantities from the duct core toward the corners, significantly distorting the axial velocity contours and enhancing mixing and heat transfer (Chandratilleke, 2012). Their results showed that Lin's low-Re $k-\epsilon$ turbulence model predicted the heat transfer and flow field better than, among others (lgci, 2016)

While experimental studies (Brundrett & Baines, 1964) and high-fidelity DNS (Gavrilakis, 1992) have documented these phenomena, the predictive capability of widely used two-equation RANS models, which

typically assume isotropic eddy viscosity, remains in question for such flows. This study aims to numerically analyze turbulent flow in a square duct using a widely adopted RANS model—the Menter's Shear Stress Transport (SST) $k-\omega$ model—which blends the robustness of the $k-\omega$ model near walls with the free-stream independence of the $k-\epsilon$ model in the outer flow. The specific goals are to:

- Simulate fully developed turbulent flow at $Re=20,000$
- Quantitatively compare the friction factor and predicted axial velocity profiles with empirical correlations.
- Visualize and analyze the predicted secondary flow structure.
- Critically evaluate the model's ability to capture turbulence anisotropy effects by comparing secondary flow intensity with reference DNS data.

Literature Review

For nearly a century, researchers have been studying the dynamics of turbulent flow in non-circular ducts. The existence of secondary motions caused by turbulent stress gradients was first proposed by Prandtl (1926), who distinguished them from inertia-driven secondary flows (the first type). Brundrett and Baines's (1964) study, which offered thorough laser Doppler velocimetry measurements of mean velocity and turbulence statistics within a square duct and clearly demonstrated the eight-vortex pattern, provided subsequent experimental validation. More in-depth investigation was made possible by the development of computational methods. As a benchmark for model validation, Gavrilakis's (1992) groundbreaking Direct Numerical Simulation at a low Reynolds number ($Re_T \approx 300$) produced a complete dataset that resolves turbulence. According to Gavrilakis, the difference between the normal Reynolds stresses ($\langle v'^2 \rangle - \langle w'^2 \rangle$) causes the secondary flow, which has a magnitude of 1-2% of the bulk velocity and drives fluid movement from high-stress areas towards the corner bisector. The performance of RANS turbulence models for this flow has been a major focus of later research. The Boussinesq eddy-viscosity hypothesis (ν_t) is used in the majority of standard two-equation models, such as the basic $k-\omega$ model or the standard $k-\epsilon$ model, which by definition assume isotropic turbulence. Demuren and Rodi (1984) pointed out that because the isotropic assumption eliminates the driving stress gradient, such models are completely unable to predict any secondary flow. They showed that in order to qualitatively describe the phenomenon, algebraic stress models (ASM) or full Reynolds Stress Models (RSM), which solve transport

equations for each component of the Reynolds stress tensor, are required. Menter's (1994) improved prediction of flow separation under adverse pressure gradients makes the $k-\omega$ SST model an advancement. Its formulation offers a more precise computation of shear stress in boundary layers, even though it is still an eddy-viscosity model. Certain advanced two-equation models with non-linear constitutive relations or explicit algebraic stress model (EASM) formulations can predict secondary flows with a respectable degree of accuracy, according to some research, such as that done by Höhl and Friedrich (2000). However, since the standard linear $k-\omega$ SST model is the default option in many commercial CFD packages for internal flow applications, a thorough assessment of its performance for this particular benchmark is still pertinent. This research adds to the current assessment.

Methodology

Governing Equations

The flow was modeled as steady, incompressible, and turbulent. The governing equations are the Reynolds-Averaged Navier-Stokes (RANS) equations for mass and momentum conservation:

$$\text{Continuity: } \nabla \cdot \bar{\mathbf{U}} = 0 \quad (1)$$

$$\text{Momentum: } \rho (\bar{\mathbf{U}} \cdot \nabla) \bar{\mathbf{U}} = -\nabla \bar{P} + \nabla \cdot [\mu (\nabla \bar{\mathbf{U}} + (\nabla \bar{\mathbf{U}})^T) - \rho \langle u'u' \rangle] \quad (2)$$

where $\bar{\mathbf{U}}$ is the mean velocity vector, \bar{P} is the mean pressure, ρ is density, μ is dynamic viscosity, and $\rho \langle u'u' \rangle$ is the Reynolds stress tensor. To close this system, the Boussinesq hypothesis is employed, relating the Reynolds stresses to the mean rate of strain:

$$-\rho \langle u_i' u_j' \rangle = \mu_t (\partial \bar{U}_i / \partial x_j + \partial \bar{U}_j / \partial x_i) - (2/3) \rho k \delta_{ij} \quad (3)$$

where μ_t is the turbulent (eddy) viscosity and k is the turbulent kinetic energy. The SST $k-\omega$ model calculates μ_t by solving two additional transport equations for the turbulent kinetic energy (k) and the specific dissipation rate (ω) (Menter, 1994).

Computational Domain and Mesh

A three-dimensional duct geometry with a square cross-section (side length $H = 0.1$ m, hydraulic diameter $D_h = H = 0.1$ m) and a length of $L = 20D_h$ (2.0 m) was created. This length ensures full hydrodynamic development. The domain was discretized using a structured hexahedral mesh in ANSYS Mesher. A high-resolution boundary layer mesh was implemented at all walls, with the first cell height set to achieve a dimensionless wall distance $y^+ < 1$ across the entire wall region, ensuring the use of an enhanced wall treatment that resolves the viscous sublayer. The final mesh contained approximately 2.1 million cells, following a

grid independence study that compared the friction factor and centerline velocity for three successively refined meshes (1.0M, 2.1M, and 4.5M cells). The results from the 2.1M and 4.5M meshes differed by less than 1%.

Boundary Conditions and Solver Settings

Inlet: A uniform velocity profile corresponding to $Re = \rho U_b D_h / \mu = 20,000$ was specified, with turbulent intensity of 5% and a turbulent viscosity ratio of 10.

Outlet: A pressure-outlet condition (zero-gauge pressure).

Walls: No-slip condition for velocity, standard wall functions for turbulence (enhanced wall treatment active).

Side Boundaries: Periodic boundary conditions were not used; the full duct length was modeled to capture the development region.

Simulations were performed using ANSYS Fluent v2022 R1. The pressure-based coupled solver was selected for its robustness. Spatial discretization was second-order upwind for momentum, k , and ω , while pressure interpolation was second order. Convergence was monitored via residuals (reduced below 10^{-6}) and stabilization of the area-weighted average pressure drop per unit length, from which the friction factor was calculated.

Validation Metrics

The primary validation metrics were:

Friction Factor (f): Calculated from the simulated pressure gradient ($\Delta p / \Delta L$) using the Darcy-Weisbach equation: $f = (2 D_h \Delta p) / (\rho U_b^2 \Delta L)$. Compared to the empirical Blasius correlation for smooth pipes: $f \approx 0.316 Re^{(-1/4)}$.

Axial Velocity Profile: Normalized velocity (u/U_b) profiles along the wall bisector (from wall to corner) were compared to the experimental data of Brundrett and Baines (1964).

Secondary Flow Pattern: After being extracted, the velocity vector field in the cross-sectional plane at the fully developed region was qualitatively compared to the traditional eight-vortex pattern. The DNS value of approximately 1.5% for a comparable Re was compared to the maximum magnitude of the secondary velocity ($V_{sec, max}$) normalized by the bulk velocity (U_b).

Results

Flow Development and Axial Velocity

About 12 to 15 hydraulic diameters downstream, the flow transitioned from a uniform inlet condition to a fully developed state. The developed flow region yielded a computed friction factor of $f = 0.0265$. This shows a difference of about 1.5% and is in close agreement with

the Blasius correlation prediction of $f = 0.0261$ at $Re=20,000$.

The normalized axial velocity profile along the wall bisector (from the duct center towards the wall) is shown in Figure 1. Brundrett and Baines's (1964) experimental data and the SST $k-\omega$ model prediction agree quite well. It accurately captures the distinctive turbulent profile, which has a flatter core and a sharper gradient close to the wall as opposed to a laminar parabolic profile. As a direct result of the secondary flow, the velocity contours in the cross-section (not shown) showed the anticipated bulging towards the corners.

Secondary Flow Field

Figure 2 illustrates the main finding by superimposing the in-plane velocity vectors on a contour plot of the secondary flow kinetic energy in the cross-section at the duct outlet (fully developed region). The eight counter-rotating vortices—two in each of the square duct's corners—were accurately predicted by the simulation. The vortices transport fluid from the duct center along the bisector towards the corner, and return fluid back along the walls, consistent with the classical description of Prandtl's second kind flow.

Quantitative Analysis of Secondary Flow

While the qualitative pattern is correct, the quantitative intensity of the secondary motion was under-predicted. The maximum secondary flow velocity magnitude, normalized by the bulk velocity, was calculated as $V_{sec, max} / U_b \approx 0.008$ (0.8%). This is significantly weaker than the intensity of approximately 1.5-2.0% observed in high-fidelity DNS and experiments for this Reynolds number range (Hüttel & Friedrich, 2000).

Discussion

The primary finding of this study is that the linear $k-\omega$ SST RANS model can predict the existence and qualitative structure of turbulence-driven secondary flow in a square duct, but it significantly underestimates its strength. The accurate prediction of the axial velocity profile and friction factor confirms the model's general competence for calculating bulk flow parameters and wall shear stress in internal flows. This explains its widespread utility in engineering design.

However, the model's fundamental flaw—the Boussinesq eddy-viscosity hypothesis (Equation 3)—is directly responsible for the underestimation of secondary flow intensity. This hypothesis effectively forces $\langle v'^2 \rangle = \langle w'^2 \rangle$ by imposing isotropy on the normal Reynolds stresses. DNS demonstrates that these stresses are actually uneven close to the corners. The secondary motion is primarily driven by the gradient of their difference, $\zeta / \partial y (\langle v'^2 \rangle - \langle w'^2 \rangle)$ (Gavrilakis, 1992). The SST $k-\omega$ model significantly attenuates this driving

mechanism by assuming isotropy. Although the predicted weak secondary flow is insufficient to match reality, it most likely results from small anisotropies introduced by the model's sensitivity to strain rates and the influence of the walls.

The implications of these findings for engineering simulations are evident. The SST $k-\omega$ model yields accurate results for applications where bulk heat transfer or pressure drop are the main concerns. However, the application of a linear eddy-viscosity RANS model may result in serious errors for issues where secondary flow is predominant, such as mixing in HVAC ducts, detailed thermal striping in nuclear reactor fuel assemblies, or particle deposition in ventilation systems. Even though they are more computationally expensive, more complex turbulence closures like Reynolds Stress Models (RSM) or Scale-Adaptive Simulation (SAS)/Detached Eddy Simulation (DES) hybrid approaches should be taken into consideration in these situations.

Limitations and Future Work

The use of a single Reynolds number and RANS model is a major drawback. A comparative analysis of different RANS models (such as RSM and non-linear $k-\epsilon$) at various Reynolds numbers should be part of future research. Additionally, the simulation assumed completely turbulent flow from the inlet; it is possible to examine the impact of inlet turbulence conditions on the development of secondary flows. Finally, validating the predictions with contemporary Particle Image Velocimetry (PIV) experimental data for the same configuration would strengthen the conclusions.

Conclusion

This numerical study demonstrated the application of the RANS-based $k-\omega$ SST turbulence model to simulate turbulent flow in a square duct. The model successfully predicted bulk flow characteristics, including the axial velocity profile and friction factor, with high accuracy when compared to empirical correlations. Crucially, it also captured this flow's non-intuitive qualitative eight-vortex secondary flow pattern. Nevertheless, the model significantly underestimated the secondary flow intensity due to its dependence on the isotropic eddy-viscosity hypothesis. This study confirms that although RANS models, such as $k-\omega$ SST, are effective tools for industrial CFD, engineers need to be aware of their basic limitations in flows where turbulence anisotropy predominates. The choice of turbulence model must be guided by the specific physical phenomena of interest in the application.

References

Ali, S., Ullah, A., Zaib, A., & Roman, M. (2024). AI-Enhanced Multiscale Modeling Of Buoyancy-

- Driven Flows In Porous Media Using Navier-Stokes Equations. *Migration Letters*, 21(S10), 1370-1378.
- Ali, S., Roman, M., & Zaib, A. (2024). One Time Periodic Solutions of Navier-Stokes Equations using Asymptotic Stability and Bifurcations. *Dialogue Social Science Review (DSSR)*, 2(4), 249-265.
- Arslan, K. (2014). Three-dimensional numerical investigation of turbulent flow and heat transfer inside a horizontal semi-circular cross-sectioned duct. *Thermal Science*, 18(4), 1145-1158.
- Brundrett, E., & Baines, W. D. (1964). The production and diffusion of vorticity in duct flow. *Journal of Fluid Mechanics*, 19(3), 375-394. <https://doi.org/10.1017/S0022112064000777>
- Chandratilleke, T. T., Nadim, N., & Narayanaswamy, R. (2012). Vortex structure-based analysis of laminar flow behaviour and thermal characteristics in curved ducts. *International Journal of Thermal Sciences*, 59, 75-86.
- Demuren, A. O., & Rodi, W. (1984). Calculation of turbulence-driven secondary motion in non-circular ducts. *Journal of Fluid Mechanics*, 140, 189-222. <https://doi.org/10.1017/S0022112084000578>
- Gavrilakis, S. (1992). Numerical simulation of low-Reynolds-number turbulent flow through a straight square duct. *Journal of Fluid Mechanics*, 244, 101-129. <https://doi.org/10.1017/S0022112092002990>
- Hüttl, T. J., & Friedrich, R. (2000). Influence of curvature and torsion on turbulent flow in helically coiled pipes. *International Journal of Heat and Fluid Flow*, 21(3), 345-353. [https://doi.org/10.1016/S0142-727X\(00\)00014-9](https://doi.org/10.1016/S0142-727X(00)00014-9)
- Igci, A. A., & Arici, M. E. (2016). A comparative study of four low-Reynolds-number $k-\epsilon$ turbulence models for periodic fully developed duct flow and heat transfer. *Numerical Heat Transfer, Part B: Fundamentals*, 69(3), 234-248.
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598-1605. <https://doi.org/10.2514/3.12149>
- Prandtl, L. (1926). Über die ausgebildete Turbulenz [On fully developed turbulence]. In *Proceedings of the Second International Congress for Applied Mechanics* (pp. 62-74). Orell Füssli Verlag.
- White, F. M. (2016). *Fluid mechanics* (8th ed.). McGraw-Hill Education.
- Wilcox, D. C. (2006). *Turbulence modeling for CFD* (3rd ed.). DCW Industries.
- Zhang, L. (2017). Numerical simulation of flows past a circular and a square cylinder at high Reynolds number, and a curved plate in transitional flow.
- Zhang, X. F., Yang, J. C., Ni, M. J., Zhang, N. M., & Yu, X. G. (2022). Experimental and numerical studies on the three-dimensional flow around single and two tandem circular cylinders in a duct. *Physics of Fluids*, 34(3).