The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2024

Volume 32, Pages 611-618

IConTES 2024: International Conference on Technology, Engineering and Science

Numerical Computation of Turbulent Flow in a Square-Sectioned 90° Bend Using Open Foam

Bouabdellah Abed
University of Sciences and Technology M.B

Mohammed Hamel
University of Sciences and Technology M.B

Abiddine DebaB
University of Sciences and Technology M.B

Abstract: This study presents the results of Computational Fluid Dynamics (CFD) simulations using the open-source software OpenFOAM to evaluate the Reynolds-averaged Navier-Stokes steady (RANS) by reproducing the fluid flow in a 90° square-section curved pipe. The simulations were carried out using three turbulence models: the standard k-ε model, the RNG k-ε model and SST k-ω model. The results were compared to previously published experimental results. The objective was to investigate the performance of RANS models in turbulent flow in a curved pipe with a 90° square section at a Reynolds number of 40000. The velocity distribution and pressure drop across the entire computational domain were analyzed, with particular attention to regions within and after the bend where secondary vortex structures were illustrated. Flows at the elbow are characterized by the presence of counter-rotating cells responsible for disrupting the flow. Analysis of the numerical results obtained from all turbulence models found good agreement with the experimental data, but there are differences between the models.

Keywords: CFD analysis, OpenFOAM, Turbulent flow, 90° Square-section bend.

Introduction

Over the years, the turbulent flow mechanism in such curved conduits has been the subject of extensive research, both numerical and experimental, due to its academic interest and industrial importance. The three-dimensional characteristics of flow occurring in square-curved pipes pose challenges for computational fluid dynamics (CFD) simulations and serve as an effective benchmark for evaluating turbulence models. Square curved pipes are frequently used in the installation of energy and electrical systems, including fuel lines, exhaust systems, and various other applications. In the case of flow in straight and rectangular pipes, the pressure distributions over different sections tend to be relatively uniform. Conversely, when dealing with flow in installed pipes and curved conduits, the velocity and pressure fields become considerably complex. Inertial forces, particularly at the elbows, generate significant pressure gradients from the inner wall to the outer wall. The vortical zones that develop immediately before and after the bend contribute to the phenomenon of secondary flows within the curvature zone, leading to additional pressure losses. Various experimental and numerical studies have been carried out to analyze and characterize these complex flow behaviors.

Lyne (1971) is one of the first to address the phenomenon of flow induced by pulsation within a curved pipe. Researchers such as Humphrey et al. (1986), Enayet et al. (1982), Azzola et al. 1(986), Cheah et al. (1996). Employed Doppler Anemometry laser techniques to assess the velocity field and visualize flow within a 90° elbow of square cross-section. Iacovides et al. (1996) utilized hot-wire anemometry to conduct measurements in

- Selection and peer-review under responsibility of the Organizing Committee of the Conference

curved conduits. Additionally, Sudo et al. (2001) performed experimental investigations on turbulent flow in a 90° section of a curved square duct at a Reynolds number of 40000 using a hot-wire anemometer. Munch and Métais (2004) investigated the impact of the section's aspect ratio on compressible and three-dimensional turbulent flows through Large Scale Simulation, specifically in curved conduits with a rectangular cross-section. Sugiyama and Mukai (2008) employed the Reynolds algebraic constraint model (ARSM) to examine three-dimensional turbulent flow within a 180° bend. Additionally, Yang et al. (2005), Chu et al. (2010), Ono et al. (2011) conducted both experimental and numerical studies on the flow characteristics in curved rectangular micro-channels. The exploration of turbulent flows in curved conduits continues to be a significant challenge, presenting critical technological implications and remaining a compelling area of research within fluid mechanics.

The present study aims to evaluate the performance of different turbulence closure models on fluid flow through a square-section curved pipe with a 90° elbow. The numerical simulations were conducted using OpenFOAM v2212, an open-source computational fluid dynamics (CFD) software. The time-averaged velocity components and pressure fields flow through the square-sectioned pipe with a bend were calculated numerically. The results obtained underwent rigorous validation by comparing them with the relevant experimental data provided by (Sudo et al., 2001).

Method

Description of the Study Case

The problem considered in this study is the turbulent flow of air through a square-sectioned 90° bend pipe. To carry out the numerical computations, the modeled geometry shown in Fig. 1 is built. The computational domain is composed of a square-sectioned duct a × a = 80 × 80 (mm) with an installed 90-degree bend with a mean radius of curvature Rm = 160 (mm). The computational domain section length of the upstream and downstream pipes of the bend is taken as 50 Dh (Dh = 80 mm).

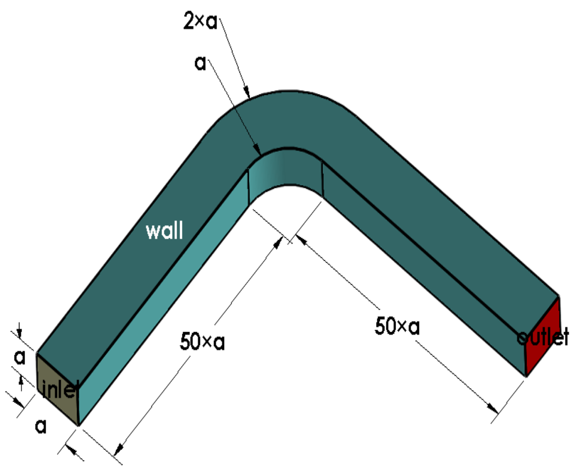

Figure 1. Geometry dimensions

Table 1. Dimensions

Dimensions	Length (mm)
Hydraulic diameter, Dh, a	80
Before elbow, L inlet	50×a
Bend curvature Radius, Rm	2×a
after the elbow, L outlet	50×a

The turbulent flow through the curved pipe is considered an incompressible isothermal flow. The numerical simulations are performed using OpenFOAM v2012 code. The CFD code uses the finite-volume method to solve governing equations for the incompressible turbulent. In this study, the numerical model solves the 3D steady incompressible Navier-Stokes and continuity equations in conjunction with two transport equation turbulence models: the k-epsilon model, the RNG model, and the k-omega model SST.

Mesh Generation

Mesh generation is an essential aspect of CFD analysis and has a significant influence on computation performance and accuracy. The computational mesh utilized in this work is generated using the blockMesh utility, a meshing tool in the open-source software OpenFOAM. A 3D multi-block structure of hexahedral meshes with inflation near the walls is used for the numerical domain considered, as shown in Fig. 2. Several meshes (Table 1) are evaluated to ensure that the solution is not mesh-dependent. As a result, a grid of one million cells will be used to illustrate the numerical results.

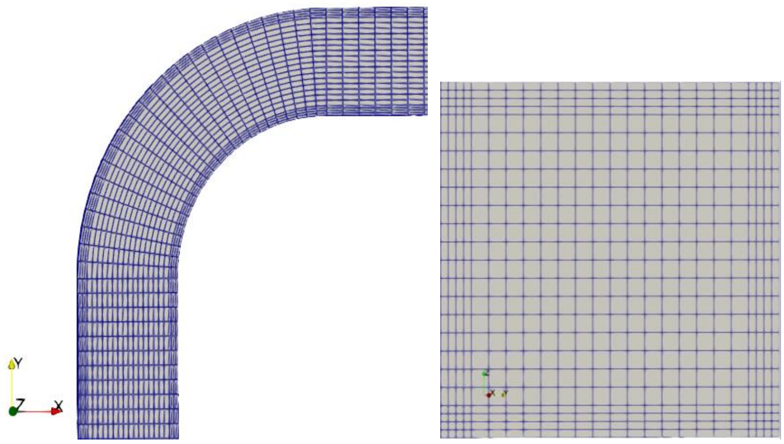

Figure 2. Computational grid (Mesh 1)

Table 2. Mesh Information

Mesh	Cells
Mesh 1	301824
Mesh 2	694656
Mesh3	1155336

Boundary Conditions

The resolution of the governing equations is achieved by utilizing the defined boundary conditions as follows: At the inlet boundary, the flow rate is determined by the Reynolds number (40000), and the flow direction is normal to the inlet surface. The velocities are zero in the remaining two directions. The pressure at this boundary is defined as a zero normal gradient. At the outlet, the velocity is established with a zero normal gradient, while the pressure is specified to be zero. For pipe walls, no-slip wall conditions are applied

(u = v = w = 0), and the wall surfaces are treated as smooth. At the pipe walls, the pressure is set with a zero normal gradient. The values of the turbulent kinetic energy and its specific dissipation rate are calculated from the inlet turbulence intensity and pipe diameter (Dutta et al., 2016).

Numerical Solution Procedure

The runs are performed using the linear upwind gradient (U) scheme as demonstrated in Beam and Warming (1976). To discretize the convective term in the momentum equation and the linear interpolation scheme for diffusion terms, while the upwind scheme as shown by Launder et al. (1983) is chosen for the divergence related to turbulence. For the gradient and divergence operators, the Gauss linear scheme is applied. The Gauss linear scheme is used for the Laplacian operator. Additionally, the interpolation scheme is set to linear. The pressure-velocity coupling is resolved using the SIMPLEC algorithm. The linear solvers used for each variable are defined as follows: The GAMG solver (Generalized Geometric-Algebraic Multigrid) (Wesseling & Oosterlee, 2001). For the pressure equation with a Gauss-Seidel type smoother and PbiCGStab (stabilized Preconditioned bi-conjugate gradient (Van der Vorst, 1992) solver for the velocity and equations with DILU preconditioner, which is a simplified diagonal-based incomplete LU preconditioner for asymmetric matrices. Tolerance values of all solvers are set to 10^{-6}.

Dependence of the Flow Solution on Grid Resolution

The dependence of the flow solution on grid resolution is investigated using three meshes. The number of grid points is multiplied by 2 in the xyz directions upstream, inside, and downstream of the elbow. The differences between results computed using different meshes are shown in Fig. 3. At U/Vc, longitudinal velocity components averaged in cross section profiles at locations y/Dh=-1 and x/Dh=1 computed using three mesh resolutions are depicted along with experimental data (Sudo et al., 2001). It seems that the differences between results calculated using different grids are very small, and henceforth, the grid with 694656 cells is used to analyze the numerical results.

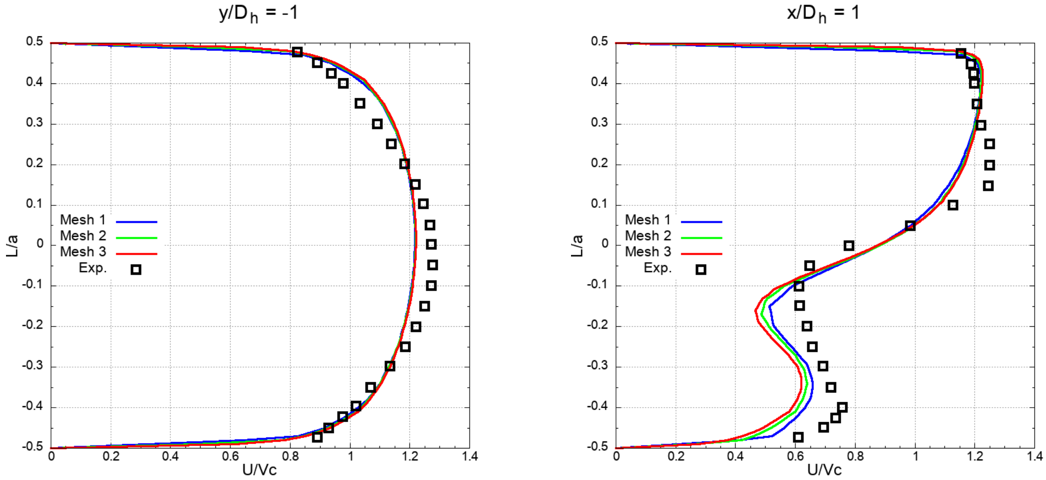

Figure 3. Comparison of the velocity profiles computed using different grids at y/Dh = -1.0 and x/Dh = 1.0 with k-ε turbulence model.

Results and Discussion

This study involves conducting a three-dimensional numerical simulation using the open-source computational fluid dynamics software, OpenFOAM version 2212, to investigate turbulent flow through a square-section curved pipe with a 90° elbow. The Reynolds number for the flow was Re = 40000, and the corresponding value of the Dean number was 20000 (Sudo et al., 2001). The Dean number is a dimensionless quantity that is used to analyze flows through curved pipes and represents the ratio of the inertial and centripetal forces to the viscous forces. The simulations were carried out using three turbulence models: the standard k-ε model, the RNG k-ε model, and the k-ω SST model. Figures 1 through 6 present a comparison between the calculated values and the measurements obtained from five stations along the streamwise direction at y/Dh = -1, φ = 30°, 60°, x/Dh = 1 and x/Dh = 10. Firstly, and in order to verify the numerical methods applied, a comparison between the numerical method utilized and the experimental results from Sudo et al. (2001) has been performed. This section presents the results of the velocity component profiles normalized with inlet velocity along the different positions in the central symmetry plane. The results indicate a significant quantitative consistency with the experimental results.

The data illustrated in Figures 4-a through 4-e indicates that there is no discernible discrepancy between this calculation and (Sudo et al., 2001). At the stations φ = 30 ° and 60 ° in the elbow, the agreement between code calculations and experimental results is very good. But at the outlet of the elbow at x/Dh = 1, computational results show no good agreement with the experiment. This difference can be produced by a deficiency of a turbulence model to predict well the flow fluctuation induced by the movement separation region and the acceleration of flow velocity.

According to Fig. 4-a, at y/Dh=-1, the flow is not yet influenced by the bend and is slightly accelerated near the inner wall and decelerated near the outer wall. At φ = 30° and φ = 60°, the flow is accelerated near the inner wall and decelerated near the outer wall. The maximum velocity occurs both near the inner wall of the bend and caused by the unfavorable pressure gradient. At the bends exit, at x/Dh = 1, the velocity profile value becomes more complex, and the fluid is further accelerated and moved near the outer wall due to the secondary flow formed in the presence of the elbow. Further downstream, z/Dh = 10, the flow has leaning to exhibit a velocity profile of a turbulent, developed flow.

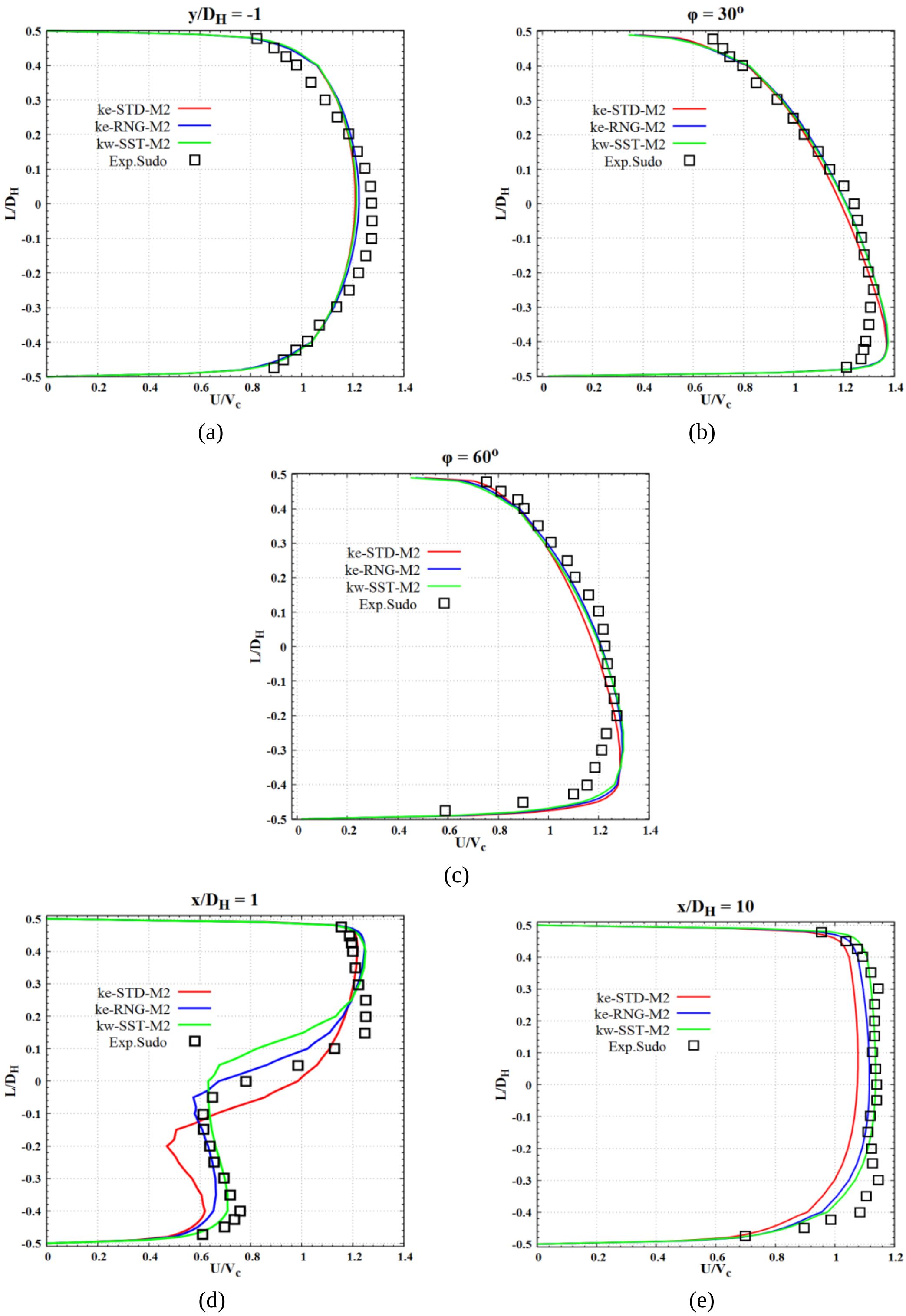

Figure 4. Normalized velocity profiles at different positions obtained with Standard $k - \varepsilon$, $k - \varepsilon$ RNG and $k - \omega$ SST models and compared with experimental results of Sudo et al. (2001)

Visualizing streamlines contours provides key information on the flow field characteristics. Figure 5 illustrates the simulation results through the surface streamlines collected at various streamwise stations over curved pipe and also at 1 Dh upstream of the bend enter and at 10 Dh downstream of the bend exit. At 30°, 45°, and 60° sections in the bend, the secondary flow peculiar to the curved duct flow appears in the cross-section, and it forms two counter-rotating vortices that circulate outwards in the central part of the duct and inwards near the upper and lower walls (the formation of the counter-rotating vortices of Dean). The fluid crossing the elbow is decelerated near the outer wall and accelerated near the inner wall caused by the unfavorable pressure gradient. After the bend exit at x/Dh=1, owing to the centrifugal force, the velocity contours are more distorted. The fluid moves further toward the outer wall due to the strong secondary flow and produces distorted velocity contours, which concave outwards in the central region near the inner wall.

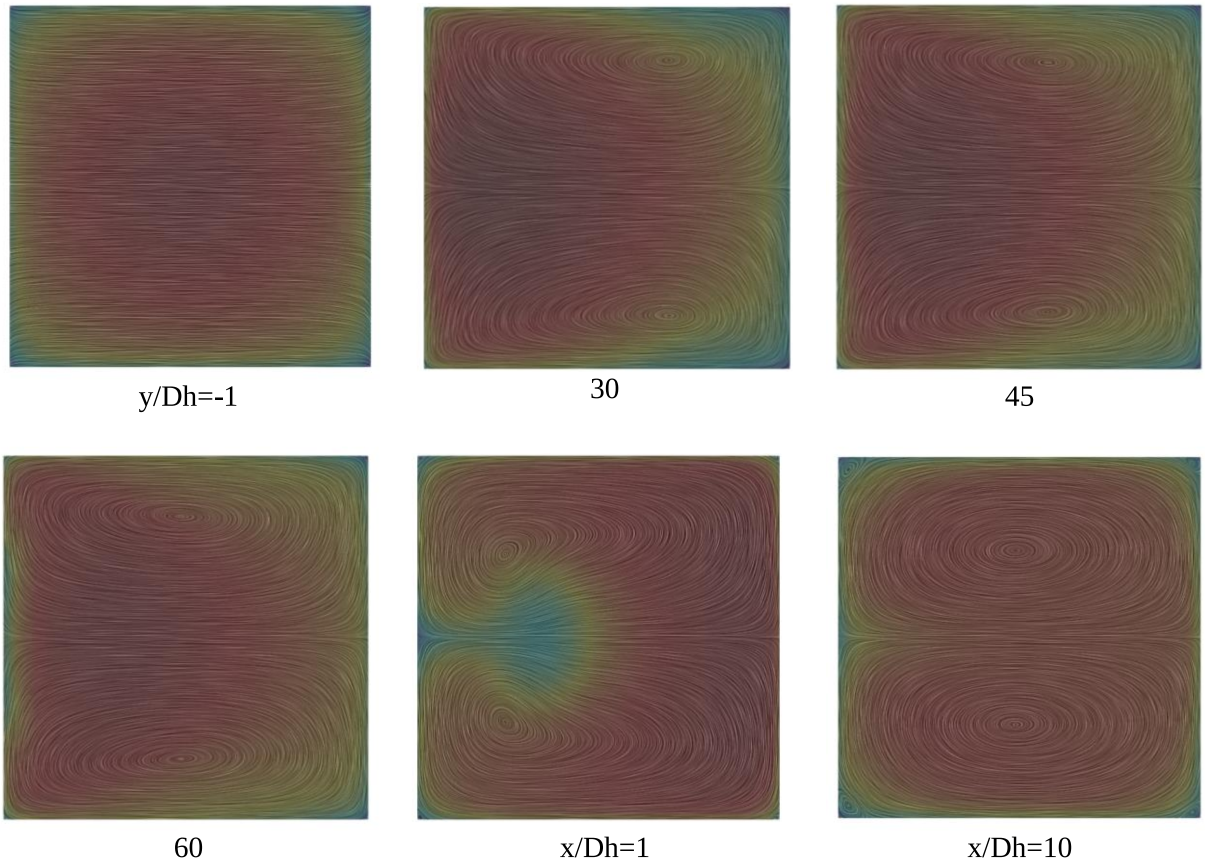

Figure 5. Surface streamlines in 90° curved pipe (k-ε turbulence model).

Figure 6 illustrates the variations in the pressure coefficient Cp along the outer and inner walls of the pipe with normalized bend cross-sectional length and for different angles α. The curves present results of numerical calculations by using different closure models. The reference pressure p_{ref} is the pressure located at the outer bend at the cross-section *y/D=-17.6* from the inlet of the bend. The pressure coefficient is defined as $Cp = \left(p - p_{ref}\right) / \left(\rho V_C^2 / 2\right)$, where V_C represents the mean velocity (m/s). From Fig. 6, it can be seen that the maximum pressure coefficient is observed on the outer wall of the bend (at the angle of 60°) and the minimum near the inner wall of the bend (at the angle of 30°), as expected due to the presence of centrifugal force.

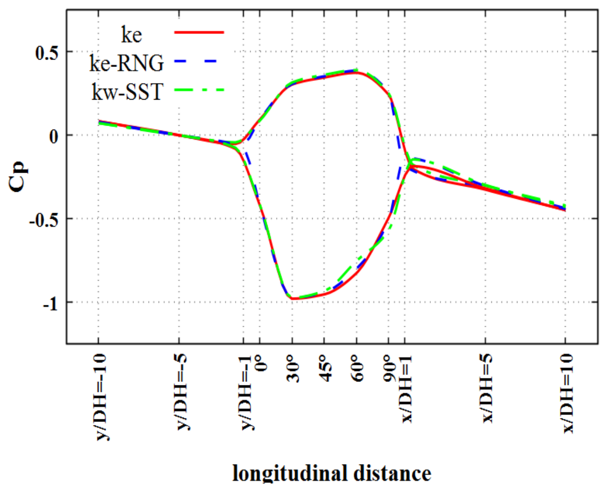

Figure 6. Pressure coefficient along the inner and outer walls in mid-plane.

Furthermore, it can be clearly observed that the Cp increases in the direction of the flow at the inner wall of the bend, by which the adverse pressure gradient occurs. Additionally, all turbulence models (the standard k-ε model, the RNG k-ε model, and the SST k-ω model) produce very similar Cp values along the inner and outer walls.

Conclusion

The current study, CFD modeling of a turbulent air flow through a square pipe with a 90 ° elbow at a Reynolds number of 40000, was performed using the open-source CFD code OpenFOAM. The study uses RANS-based CFD models to capture the main features of the turbulent 90° bent pipe with three turbulence models embedded in the code, namely, the standard k-ε model, the RNG k-ε model, and the SST k-ω model. The problem geometry and the mesh were generated by the blockMesh Dictionary tool of OpenFOAM v2212. The SIMPLEC algorithm was used for the treatment of the pressure-velocity coupling, and the PDEs conservation equations were solved using the finite volume method with a convergence criterion set to 10−6 for all variables.

Results curves of the numerical model showed a good agreement with the experimental data given by (Sudo et al., 2001). The results presented by streamlines at different stations show that the geometry of the bend induces a pair of counter-rotating vortices in the streamwise direction of the flow. The driving force of the secondary flows is a result of the centrifugal force-radial pressure gradient imbalance, which acts on the moving fluid along the side walls of the bend and downstream. As one of the main conclusions of this work, the open-source OpenFOAM CFD code and the numerical model adopted sufficiently captured the flow field and can be used to study fluid fluids over square curved pipes.

Scientific Ethics Declaration

The authors declare that the scientific ethical and legal responsibility of this article published in EPSTEM Journal belongs to the authors.

Acknowledgements or Notes

* This article was presented as an oral presentation at the International Conference on Technology, Engineering and Science (www.icontes.net) held in Antalya/Turkey on November 14-17, 2024.

* The authors are very thankful to the Laboratory of Applied Mechanics; Department of Mechanical Engineering; University of Sciences and Technology “Mohamed Boudiaf” Oran, Algeria.

References

Azzola, J., Humphrey, J. A. C., Iacovides, H., & Launder, B. E. (1986). Developing turbulent flow in a U-bend of circular cross-section: Measurement and computation. *Journals of Fluids Engineering, 108*(2), 214-221.

Beam, R. M., & Warming, R. F. (1976). An implicit finite-difference algorithm for hyperbolic systems in conservation-law form. *Journal of Computational Physics*, *22*(1), 87-110.

Cheah, S. C., Iacovides, H., Jackson, D. C., Ji, H., & Launder, B. E. (1996). LDA investigation of the flow development through rotating U-ducts. *Journal of Turbo Machinery, 118*(3), 590-596.

Chu, J. C., Teng, J. T., & Greif, R. (2010). Experimental and numerical study on the flow characteristics in curved rectangular microchannels. *Applied Thermal Engineering*, *30*(13), 1558-1566.

Dutta, P., Saha, S. K., Nandi, N., & Pal, N. (2016). Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling. *Engineering Science and Technology, an International Journal*, *19*(2), 904-910.

Enayet, M. M., Gibson, M. M., Taylor, A. M. K. P., & Yianneskis, M. (1982). Laser-Doppler measurements of laminar and turbulent flow in a pipe bend. *International Journal of Heat and Fluid Flow, 3*(4), 213-219.

Humphrey, J. A. C., Whitelaw, J. H., & Yee, G. (1981). Turbulent flow in a square duct with strong curvature. *Journal of Fluid Mechanics, 103*, 443-463.

Lyne, W. H. (1971). Unsteady viscous flow in a curved pipe. *Journal of Fluid Mechanics*, *45*(1), 13-31.

Lacovides, H., Launder, B. E., Loizou, P. A., & Zhao, H. H. (1990). Turbulent boundary-layer development around a square-sectioned U-bend: Measurements and computation. *Journal of Fluids Engineering, 112*(4), 409-415.

Launder, B. E., & Sharma, B. I. (1974). Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Letters in Heat and Mass Transfer*, *1*(2), 131-137.

Launder, B. E., & Spalding, D. B. (1983). The numerical computation of turbulent flows. In *Numerical prediction of flow, heat transfer, turbulence and combustion* (pp.96-116). Pergamon.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, *32*(8), 1598-1605.

Münch, C., Hébrard, J., & Métais, O. (2004). Large Eddy Simulations of turbulent flow in curved and s-shape ducts. *Direct and Large-Eddy Simulation V: Proceedings of the Fifth International ERCOFTAC Workshop* (pp.527-536). Netherlands: Springer.

Ono, A., Kimura, N., Kamide, H., & Tobita, A. (2011). Influence of elbow curvature on flow structure at elbow outlet under high Reynolds number condition. *Nuclear Engineering and Design*, *241*(11), 4409-4419.

Sudo, K., Sumida, M., & Hibara, H. (2001). Experimental investigation on turbulent flow in a square-sectioned 90-degree bend. *Experiments in Fluids*, *30*(3), 246-252.

Sugiyama, H., Tanaka, T., & Mukai, H. (2008). Numerical analysis of turbulent flow separation in a rectangular duct with a sharp 180-degree turn by algebraic Reynolds stress model. *International Journal for Numerical Methods in Fluids*, *56*(12), 2207-2228.

Van der Vorst, H. A. (1992). Bi-CGSTAB: A fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems. SIAM *Journal on scientific and Statistical Computing*, *13*(2), 631-644.

Wesseling, P., & Oosterlee, C. W. (2001). Geometric multigrid with applications to computational fluid dynamics. *Journal of Computational and Applied Mathematics*, *128*(1-2), 311-334.

Yakhot, V. S. A. S. T. B. C. G., Orszag, S. A., Thangam, S., Gatski, T. B., & Speziale, C. (1992). Development of turbulence models for shear flows by a double expansion technique. *Physics of Fluids A: Fluid Dynamics*, *4*(7), 1510-1520.

Yang, W. H., Zhang, J. Z., & Cheng, H. E. (2005). The study of flow characteristics of curved microchannel. *Applied Thermal Engineering*, *25*(13), 1894-1907.

Zhang, J., Wang, D., Wang, W., & Zhu, Z. (2022). Numerical investigation and optimization of the flow characteristics of bend pipe with different bending angles. *Processes*, *10*(8), 1510.

Author Information

Bouabdellah Abed
University of Sciences and Technology M.B
Oran, Algeria
Contact e-mail : bouabdellah.abed@univ-usto.dz

Mohamed Hamel
University of Sciences and Technology M.B
Oran, Algeria

Abiddine Debab
University of Sciences and Technology M.B
Oran, Algeria

To cite this article:

Abed, B., Hamel, M., & Debab, A. (2024). Numerical computation of turbulent flow in a square-sectioned 90° bend using OpenFOAM. *The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 32,* 611-618.