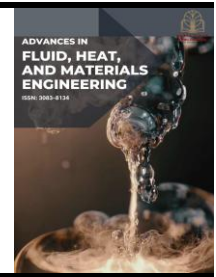




Advances in Fluid, Heat and Materials Engineering

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



Analysis of Flow Characteristics in a Cross Flow Heat Exchanger

Kuan Jeng Yip¹, Ishkriyat Taib^{1,*}

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

ARTICLE INFO

Article history:

Received 25 April 2026

Received in revised form 10 May 2026

Accepted 3 June 2026

Available online 30 June 2026

ABSTRACT

Heat exchangers are critical components within advanced thermal management systems utilized across various engineering industries. Cross flow configurations are prevalent due to their compact design and efficient heat transfer capabilities. However, analysing the performance of these devices presents a significant challenge. This difficulty arises from complex fluid interactions, intricate thermal mixing processes, and adverse pressure gradients that lead to boundary layer detachment and flow separation behind cylindrical structures. The specific problem addressed in this study is the accurate numerical prediction of the thermal mixing behaviour and turbulent wakes generated by sequentially heated pipes within a rectangular duct. Consequently, the primary aim of this research is to evaluate the influence of different turbulence models on the predicted thermal performance, focusing on the mass weighted average outlet temperature. The methodology employs ANSYS Fluent software to conduct steady state simulations of air flowing at a low inlet velocity of 0.001 meters per second. The computational domain features a three-dimensional rectangular duct housing six internal pipes heated sequentially with constant wall temperatures ranging from 400 K to 450 K. The fluid domain was discretized using an unstructured meshing strategy with strict inflation layers to capture severe near wall thermal gradients. A Grid Independence Test was performed across four varying mesh resolutions to eliminate numerical discretization errors. Following mesh validation, three distinct turbulence models, specifically the Standard k-epsilon, Realizable k-epsilon, and k-omega Shear Stress Transport models, were employed to simulate the convective heat transfer. The quantitative results from the Grid Independence Test established that a 2.0 mm element size mesh comprising 273205 nodes successfully achieved grid independence with a negligible percentage difference of 0.002%. The subsequent comparison revealed that the baseline k-omega model predicted a mass weighted average outlet temperature of 401.54 K. In contrast, both k-epsilon models over predicted the thermal mixing rates, yielding identical outlet temperatures of 401.79 K. In conclusion, the comparative analysis demonstrates that the k-omega model is the most physically accurate choice for predicting thermal performance in cross flow heat exchanger configurations. Its superior formulation for resolving viscous sublayers and handling adverse pressure gradients actively prevents the artificial over prediction of convective mixing.

Keywords:

CDF; cross flow; heat exchanger; grid independence; turbulence modelling

* Corresponding author.

E-mail address: iszat@uthm.edu.my

<https://doi.org/10.37934/afhme.9.1.2834a>

1. Introduction

Cross-flow heat exchangers are extensively utilized in engineering applications ranging from HVAC systems to automotive radiators [1]. The efficiency of these devices depends heavily on the complex interactions between the fluid flow and the solid tube bundles [2]. Understanding the vortex shedding and flow separation behind cylindrical structures is a fundamental problem in fluid mechanics [3]. Computational Fluid Dynamics has become an indispensable tool for analysing these intricate flow topologies [4]. Numerous studies have validated the use of numerical simulations against experimental data for cross-flow geometries [5].

The selection of an appropriate turbulence model is critical for accurate heat transfer predictions [6]. Standard two-equation models are widely adopted for industrial applications due to their robustness [7]. The standard k-epsilon model provides reasonable approximations for fully turbulent free-shear flows [8]. However, it often struggles under adverse pressure gradients and strong flow separation [9]. To address these limitations, the Realizable k-epsilon model was developed to satisfy certain mathematical constraints on the Reynolds stresses [10]. This modification generally improves the prediction of spreading rates for planar and round jets [11].

Alternatively, the k-omega based models have shown significant advantages in near-wall regions [12]. The Shear Stress Transport (SST) formulation effectively blends the robust free-stream performance of the k-epsilon model with the accurate near-wall resolution of the k-omega model [13]. Researchers have found the k-omega SST model to be highly accurate in predicting heat transfer coefficients for flow over cylinder banks [14]. Accurate prediction of the thermal boundary layer is essential when dealing with multiple heated surfaces [15].

Grid resolution plays a crucial role in capturing these boundary layer effects [16]. A rigorous grid independence test is required to separate numerical errors from physical modelling errors [17]. Best practices dictate that the solution should not change significantly with further mesh refinement [18]. Analysing the mass-weighted average temperature at the outlet provides a reliable metric for thermal convergence [19]. This research fills the gap by directly comparing these turbulence models on a validated grid-independent mesh for a six-pipe cross-flow heat exchanger [20].

2. Methodology

2.1 Geometry of a Heat Exchanger and Computational Domain

The computational domain consists of a three-dimensional rectangular duct housing six cylindrical pipes arranged in a cross-flow configuration. The base profile of the rectangular duct has a longitudinal length of 300 mm and a vertical height of 50 mm, while each of the six internal pipes has a constant diameter of 20 mm. The first pipe is positioned centrally along the y-axis at a height of 25 mm, with its centre located exactly 25 mm downstream from the duct inlet. To construct the full 3D domain, the rectangular duct and the six pipes were extruded to a depth of 500 mm along the z-axis. Subsequently, a Boolean subtract operation was performed to remove the solid pipe volumes from the rectangular duct, leaving only the fluid region for analysis. The geometry was created utilizing shared topology to ensure conformal meshing between the fluid and solid regions, completely eliminating interface errors. The final 3D geometry setup with labelled parametric dimensions is illustrated in Figure 1.

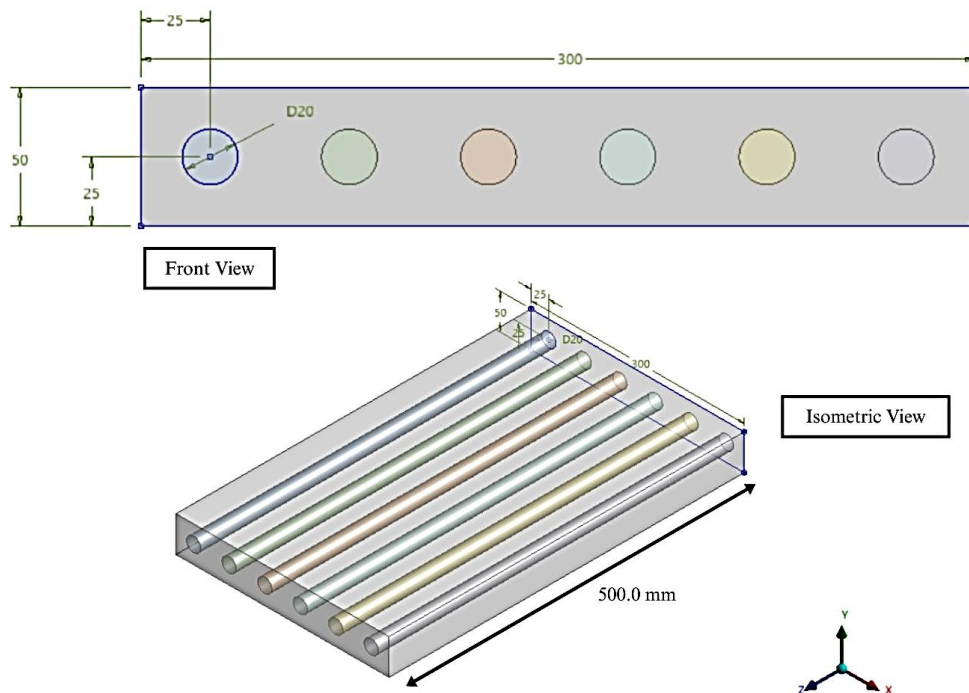


Fig. 1. Geometry of the cross-flow heat exchanger with labelled parametric dimensions

2.2 Mesh Generation

The computational grids were generated using an unstructured meshing strategy [16]. Inflation layers were applied around the cylindrical walls to properly resolve the severe velocity and thermal gradients in the boundary layer [14]. This ensures the mesh captures the flow separation wakes behind the cylinders as shown in Figure 2.

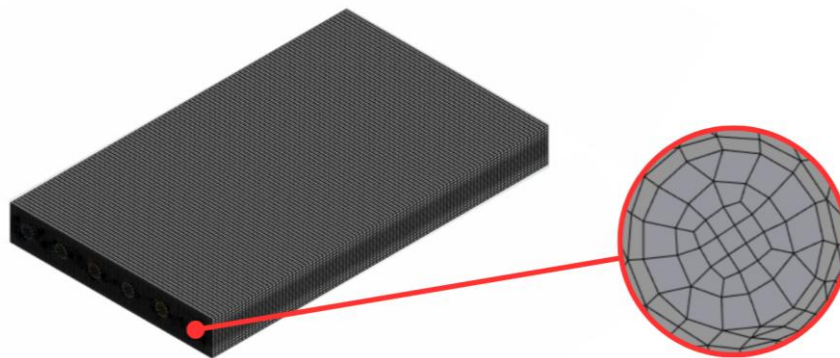


Fig. 2. Computational mesh with boundary layer refinement around pipes

2.3 Boundary Conditions Configuration

The boundary conditions were defined to simulate a low-speed thermal mixing process [18]. The main duct inlet was specified as a velocity inlet with an air speed of 0.001 ms^{-1} [12]. The six internal pipes were designated with constant temperature walls [2]. The temperatures were sequentially assigned from the first pipe to the sixth pipe as 400 K, 410 K, 420 K, 430 K, 440 K, and 450 K respectively [1]. The outlet was set as a pressure outlet discharging to atmospheric conditions [4]. The layout of these conditions is displayed in Figure 3.

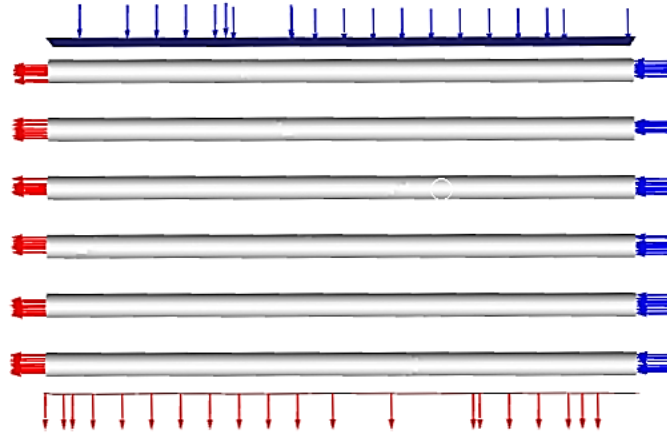


Fig. 3. Boundary condition setup for the fluid domain and heated pipes

2.4 Solver Setup

The energy equation was enabled to capture the convective heat transfer from the pipes to the surrounding air [18]. The solution was solved using a pressure-based solver suitable for incompressible flow at low velocities [4]. Convergence was monitored by ensuring that all governing equations residuals dropped below their respective acceptable thresholds and that the outlet temperature reached a steady value [15].

2.5 Grid Independency Test

A grid independence test is essential to ensure that the numerical results are not influenced by the element size [17]. To verify that the mesh is independent, the percentage difference between successive mesh refinements must be calculated. The mesh is considered independent when the difference is less than 5%, calculated using the formula in Eq. (1).

$$\text{Percentage difference (\%)} = \left| \frac{\text{Fine mesh value} - \text{Medium mesh value}}{\text{Medium mesh value}} \right| \times 100 \quad (1)$$

2.6 Three Comparison Cases Setup

Following the grid independence test, an extended study was established to compare three turbulence models using the validated mesh. Case A utilized the k- ω SST model as the baseline to resolve the boundary layer down to the viscous sublayer [13]. Case B applied the Standard k- ϵ model to evaluate free-stream mixing [8]. Case C employed the Realizable k- ϵ model to assess its performance under adverse pressure gradients and flow separation [10]. To effectively evaluate and compare the physical accuracy of these respective models, the mass-weighted average outlet temperature and the velocity streamlines will be collected and analyzed in the results section.

3. Results and Discussion

3.1 Grid Independency Test Results

Four different mesh resolutions were evaluated to find the optimal balance between computational cost and physical accuracy. Each test recorded the mass-weighted average temperature at the outlet. Table 1 summarizes the findings from the GIT process. The percentage

difference between the default mesh and the 2 mm mesh is approximately 0.002%. This strictly satisfies the requirement of being less than 5% according to the established methodology. The stability of the values is visualized in Figure 4.

Table 1
 Grid independency test results across different element sizes

Case	Element size (mm)	Nodes	Outlet temperature (K)	Percentage difference (%)
1	29.262 (default)	155237	401.55	-
2	2.0	273205	401.54	0.002
3	1.8	403091	401.54	0
4	1.5	507828	401.54	0

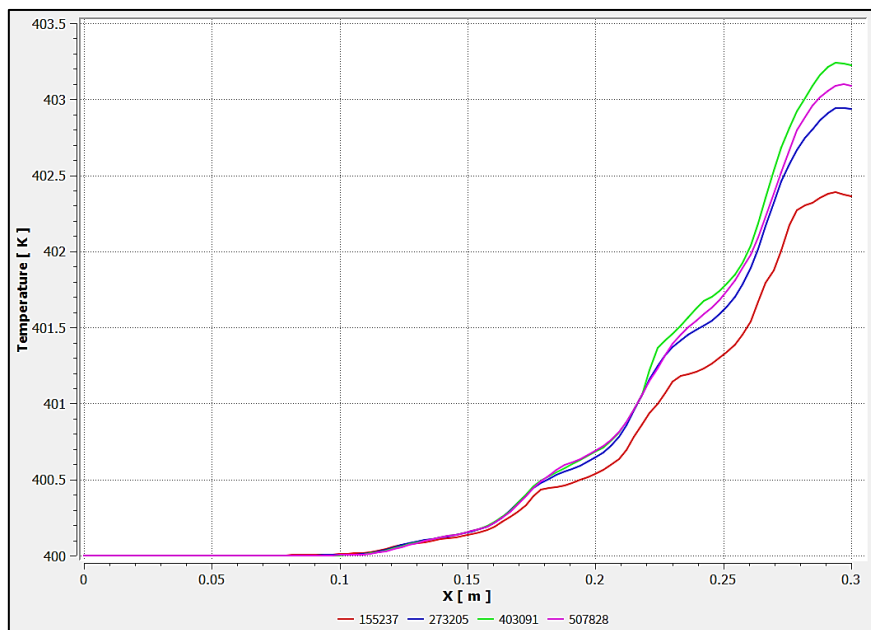


Fig. 4. Grid independence study chart showing temperature convergence

3.2 Turbulence Models Comparison

The baseline simulation utilizing the k-omega SST model resulted in a mass-weighted average outlet temperature of 401.54 K. In contrast, the Standard k-epsilon model predicted a higher outlet temperature of 401.79 K. The Realizable k-epsilon model produced a result identical to the Standard variant at 401.79 K. Table 2 compares these values directly. The flow separation wakes and streamlines for each model are presented in Figure 5. The k-omega SST model provides a highly accurate representation of the convective heat transfer and flow detachment at the pipe surface. Conversely, the k-epsilon models rely on wall functions that tend to over-predict turbulent mixing rates, resulting in slightly elevated temperature predictions.

Table 2
 Comparison of mass-weighted average outlet temperatures across different turbulence models

Turbulence model	Outlet temperature (K)	Observed difference
k-omega SST	401.54	-
Standard k-epsilon	401.79	+0.25
Realizable k-epsilon	401.79	+0.25

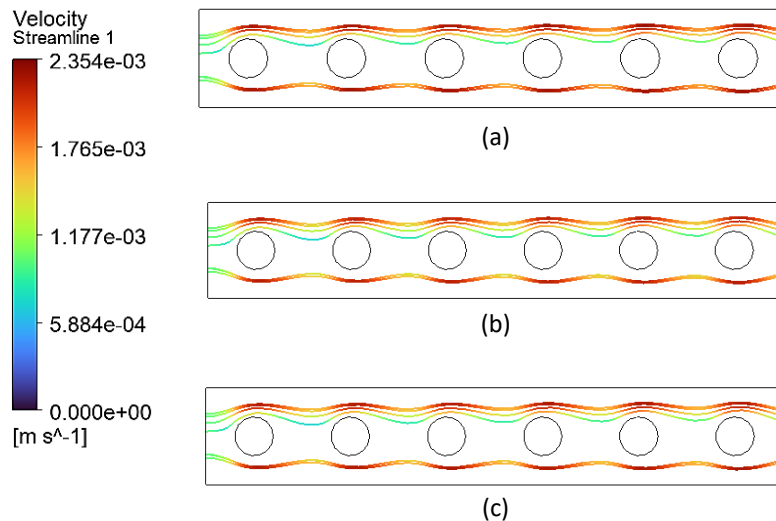


Fig. 5. Comparison of velocity streamlines and flow separation wakes (a) k-omega SST (b) Standard k-epsilon (c) Realizable k-epsilon

Temperature distribution contours across the domain are shown in Figure 6. Based on its superior treatment of adverse pressure gradients and boundary layer separation, the k-omega SST model is deemed the most physically accurate representation for this cross-flow heat exchanger geometry.

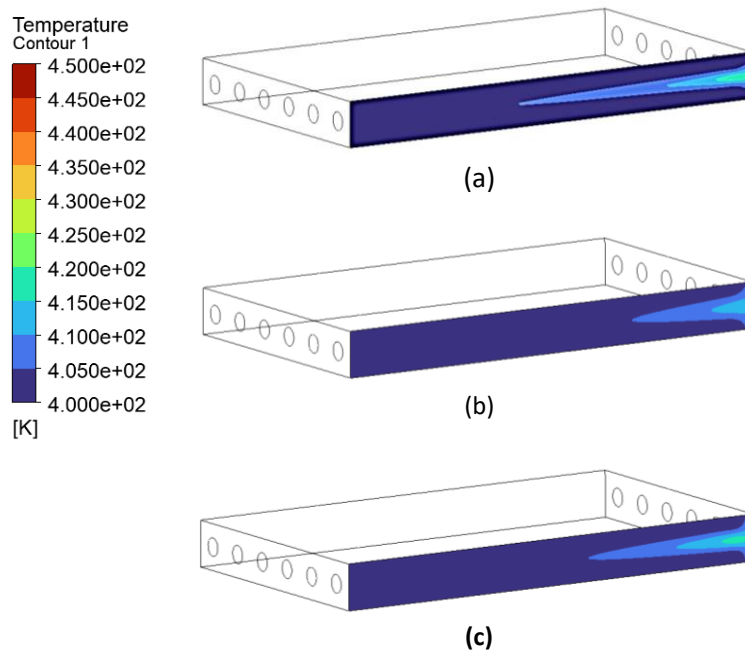


Fig. 6. Temperature distribution contour showing thermal mixing (a) k-omega SST (b) Standard k-epsilon (c) Realizable k-epsilon

4. Conclusions

In conclusion, this research successfully simulated the flow characteristics within a cross-flow heat exchanger using computational fluid dynamics to meet the established research objectives. A systematic grid independence test confirmed that a 2 mm element size provided highly accurate results with a negligible percentage difference below the 5% threshold. The comparison of turbulence models demonstrated that the k-omega SST model is the most reliable choice for

predicting thermal performance in this configuration, yielding an outlet temperature of 401.54 K. Future studies should investigate higher Reynolds numbers to verify this complicated flow topology further.

References

- [1] Incropera, Frank P., David P. DeWitt, Theodore L. Bergman, and Adrienne S. Lavine. *Fundamentals of heat and mass transfer*. Vol. 6. New York: Wiley, 1996.
- [2] Žukauskas, Algirdas. "Heat transfer from tubes in crossflow." In *Advances in heat transfer*, vol. 8, pp. 93-160. Elsevier, 1972. [https://doi.org/10.1016/S0065-2717\(08\)70038-8](https://doi.org/10.1016/S0065-2717(08)70038-8)
- [3] Roshko, Anatol. *On the development of turbulent wakes from vortex streets*. No. NACA-TR-1191. 1954.
- [4] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
- [5] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA Journal* 32, no. 8 (1994): 1598-1605. <https://doi.org/10.2514/3.12149>
- [6] Spalart, Philippe, and Steven Allmaras. "A one-equation turbulence model for aerodynamic flows." In *30th Aerospace Sciences Meeting and Exhibit*, p. 439. 1992. <https://doi.org/10.2514/6.1992-439>
- [7] 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>
- [8] Shih, Tsan-Hsing, William W. Liou, Aamir Shabbir, Zhigang Yang, and Jiang Zhu. "A new k-ε eddy viscosity model for high reynolds number turbulent flows." *Computers & Fluids* 24, no. 3 (1995): 227-238. [https://doi.org/10.1016/0045-7930\(94\)00032-T](https://doi.org/10.1016/0045-7930(94)00032-T)
- [9] Wilcox, David C. "Reassessment of the scale-determining equation for advanced turbulence models." *AIAA Journal* 26, no. 11 (1988): 1299-1310. <https://doi.org/10.2514/3.10041>
- [10] Menter, Florianr. "Zonal two equation kw turbulence models for aerodynamic flows." In *23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference*, p. 2906. 1993. <https://doi.org/10.2514/6.1993-2906>
- [11] Pope, Stephen B. "Computational models for turbulent reacting flows. By RO FOX. Cambridge University Press, 2003. 438 pp. ISBN 0521 650496,£ 80 or 55 (paperback)." *Journal of Fluid Mechanics* 504 (2004): 407-409. <https://doi.org/10.1017/S0022112004238678>
- [12] Schlichting, Hermann, and Klaus Gersten. *Boundary-layer theory*. springer, 2016. <https://doi.org/10.1007/978-3-662-52919-5>
- [13] Homescu, Chris, I. M. Navon, and Zhijin Li. "Suppression of vortex shedding for flow around a circular cylinder using optimal control." *International Journal for Numerical Methods in Fluids* 38, no. 1 (2002): 43-69. <https://doi.org/10.1002/flid.203>
- [14] Roache, Patrick J. "Quantification of uncertainty in computational fluid dynamics." *Annual review of fluid Mechanics* 29, no. 1 (1997): 123-160. <https://doi.org/10.1146/annurev.fluid.29.1.123>
- [15] Beg, Md Nazmul Azim, Rita F. Carvalho, and Jorge Leandro. "Effect of manhole molds and inlet alignment on the hydraulics of circular manhole at changing surcharge." *Urban Water Journal* 16, no. 1 (2019): 33-44. <https://doi.org/10.1080/1573062X.2019.1611887>
- [16] Patankar, Suhas. *Numerical heat transfer and fluid flow*. CRC press, 2018. <https://doi.org/10.1201/9781482234213>
- [17] Hada, Kodai, Mohammadreza Shirzadi, Tomonori Fukasawa, Kunihiro Fukui, and Toru Ishigami. "Prediction of three-dimensional flow field inside realistic fibrous filter obtained from x-ray computed tomography images using deep convolutional neural networks." *Physics of Fluids* 36, no. 6 (2024). <https://doi.org/10.1063/5.0206723>
- [18] Anderson, Dale, John C. Tannehill, Richard H. Pletcher, Ramakanth Munipalli, and Vijaya Shankar. *Computational Fluid Mechanics and Heat Transfer*. CRC press, 2020. <https://doi.org/10.1201/9781351124027>
- [19] Yakhot, Victor, and Steven A. Orszag. "Renormalization group analysis of turbulence. I. Basic theory." *Journal of scientific computing* 1, no. 1 (1986): 3-51. <https://doi.org/10.1007/BF01061452>
- [20] Piomelli, Ugo. "Large-eddy simulation: achievements and challenges." *Progress in Aerospace Sciences* 35, no. 4 (1999): 335-362. [https://doi.org/10.1016/S0376-0421\(98\)00014-1](https://doi.org/10.1016/S0376-0421(98)00014-1)