RESEARCH ARTICLE | JANUARY 25 2019

Verification of heat transfer enhancement in tube with spiral corrugation

Nurul Izzwa Nadila; Tholudin Mat Lazim; Shabudin Mat

AIP Conf. Proc. 2062, 020032 (2019)
https://doi.org/10.1063/1.5086579

Verification of Heat Transfer Enhancement in Tube with Spiral Corrugation

Nurul Izzwa Nadila[1, a)] Tholudin Mat Lazim[2, b),] Shabudin Mat[3, c)]

[1, 2, 3] *School of Mechanical Engineering, Faculty of Engineering, Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia.*

[a)] Corresponding author: izzwanadila@gmail.com
[b)] tholudin@gmail.com
[c)] shabudin@utm.my

Abstract. Demand of high performance heat exchanger in industrial application is increasing since the depletion of energy resources such as in food processing plant, air-conditioning system and power plant. One of the ways of saving energy is by enhancing heat transfer performance, which in return will give a high performance heat exchanger. Existing enhancing techniques can be classified into three different categories which are active method, passive method and compound method. Spirally corrugated tube is one of the passive heat transfer enhancement method which involve surface extensions. The type of surface extension used in heat transfer performance will contribute to heat transfer coefficient and pressure drop, and directly affected the heat transfer performance. Traditionally, experimental studies were carried out to get the desired results but with the help of technology, numerical simulation is one of the promising alternatives in predicting reliable results for real application. For reliable numerical results, experimental studies are still necessary for the validation process and the verification process which comes before it is also mandatory. Thus, this paper focus on verification the numerical simulation of flow in a double-pipe heat exchanger with spirally corrugated internal tube. It was done for laminar flow with Reynolds number of 1000. Numerical simulation model using commercial CFD software were run with different mesh sizes to determine the grid independent solution and to choose which mesh is suitable to use for the whole simulation process. Minimizing and choosing the right mesh were done through Grid Convergence Index (GCI). Solutions of three different grids with their residuals convergence are also presented to fulfil the verification steps. The level of grid independence is evaluated using a form of Richardson extrapolation and the study shows that the finest grid solution has a GCI of less than 5%.

INTRODUCTION

Heat exchangers perform heat transfer by transferring energy from one system to another as a result of temperature difference [1] obeying the second law of thermodynamics in all chemical reactions and unit operations. The transfer of heat is always from high temperature to a lower temperature region before it stops when the two medium involved in a system has reach the same temperature. It was divided into three modes which are the conduction, convection and radiation. All these three plays different roles from one another as conduction involves the interactions between the more energetic particles to the adjacent less energetic one which can take place in either solids, liquids or gasses; convection is the transfer of heat which involves the combined effects of conduction and fluid motion at the same time as define by Greitzer *et al.* [2] in 2004, it happen between a solid surface and adjacent liquids or gasses in motion; on the other hand, radiation is the one which neither involves the interaction between solid to liquid or gas but it happen when matter in the form of magnetic waves emit energy before changing it into electronic configuration of atoms or molecules and it is the fastest among the three modes which is at the speed of light.

The 10th International Meeting of Advances in Thermofluids (IMAT 2018)
AIP Conf. Proc. 2062, 020032-1–020032-8; https://doi.org/10.1063/1.5086579
Published by AIP Publishing. 978-0-7354-1790-8/$30.00

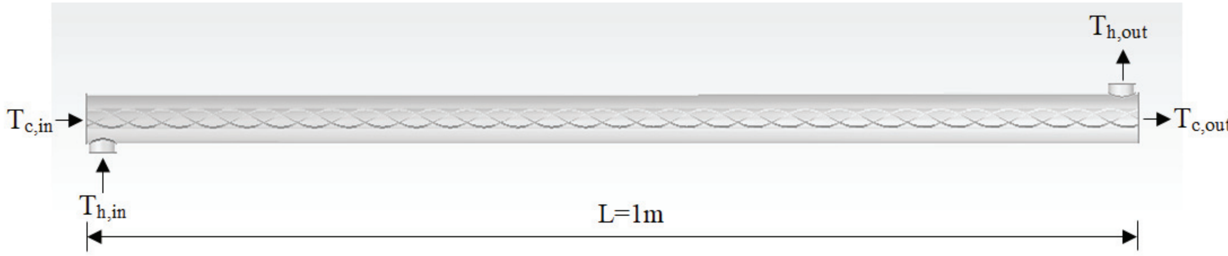


FIGURE 1. Schematic of a double pipe parallel flow heat exchanger

Heat exchanger is a device or a medium of transferring heat without mixing of the two fluids at different temperature. It does involve conduction and convection but the convection heat transfer modes is more dominant as presented by Kanade Rahul *et al.* [3] because it usually occurs in both working fluids and conduction through walls of heat exchanger which separates the fluids. Separation of fluids makes full use of both inlet temperature to transfer from high to low temperature and finally results in enhancing the heat transfer. Shell and tube type of heat exchangers are being used in industries whereas plate, spiral and brazed finned type heat exchangers are used for specific purposes. In a double pipe parallel flow heat exchanger concentric tube construction, hot and cold fluid flow in the same direction as presented in Fig. 1. Small size of inside tube diameter makes it possible to use in high pressure applications and also useful in wide range of temperature. The hot fluid flowing from the inlet of the outer tube denoted as $T_{h,in}$ transfers heat to cold water in the inner tube. The system will be in steady state and incompressible fluid flow for the controlled flow rate and inlet temperatures.

The optimization of overall heat transfer coefficient and surface area are the most important parameter in making sure the heat transfer enhancement are able to meet the increasing demand from industry. There are three methods that can enhance heat transfer rate through heat exchanger which are the active, passive and compound methods. These three method works at its best after their own characteristics in enhancing heat transfer. Active method plays its own role with the help of external power to maintain the enhancement; passive method do not need external power to keep enhancing the heat transfer, it just needs to undergo some special surface geometries or fluid additives as mentioned in H. Balla [4]; and compound method does the combination of both active and passive in it to maintain the enhancement. These enhancement has brought researchers into researching more about it since 1977 as per mentioned by Z.S Kareem *et al.* [5] in their research paper of reviewing heat transfer enhancement using all three methods as the technology growing until the latest of 2015. It was a comparison between active, passive and compound method in both experimental and numerical studies, leads to the agreement of using passive method in increasing heat transfer which is always accompanied with pressure drop and increase significantly in turbulent flow.

Reviewing only is not enough without proving it like what M. Omidi [6] did when comparing the application of heat transfer enhancement methods using double pipe heat exchanger (DPHE). It consists of the development procedure that this heat exchanger went through and gives promising results of passive heat transfer enhancement method through corrugation. The secondary flow plays a significant role in increasing the heat transfer rate which is induced by the surface geometries modification [5, 6]. Apart from that, passive heat transfer enhancement especially in DPHE has been frequently cited [7-10] and studies start to concern about it with the new enhancement of adding some nano fluids [11,12] into it. Nano fluids involved in the flow of fluid can increase the heat transfer up to 53.95%.

Therefore, the enhancement in that particular field has brought benefit in human lives, especially in forced convective heat transfer field. The main reason for employing heat transfer enhanced techniques is for cutting the costs as well as for practical purposes. One of the popular heat transfer enhancement is the artificial surface roughness, ribs, grooves and corrugations. Spirally corrugated tube is one of the promising methods in enhancing the heat transfer in tube, and one of the cost-effective enhancement methods. This paper is focusing on verification of a parallel flow of DPHE with spirally corrugated internal tube. It was done for laminar with Reynolds number of 1000. Numerical simulation model using commercial CFD software were run with different mesh size to minimize the grid and to choose which mesh is suitable to use for the whole simulation process. Minimizing and choosing the right mesh were done through Grid Convergence Index (GCI). Solutions of three different grids with their residuals convergence are also presented to fulfil the verification steps. Commercial CFD software is utilized to carry out the flow analysis for the specified inlet and outlet conditions which were set based on what has been done in experimental studies by I. Imani [13]. The estimates outlet temperatures from simulation is used to calculate Nusselt number and denoted as ∅ in GCI. These numerical simulations data will be useful in the design as well as in the performance evaluation of double pipe heat exchanger.

NUMERICAL METHODOLOGY

The analysis of heat transfer enhancement in this study was carried out using commercial CFD software solver based on the finite volume method discretization of the basic fluid flow equations for the conservation of mass, momentum, and energy as shown by Equations (1) to (3) respectively. The discretization equations were then solved through numerical procedures using a workstation with core i7-3770K system of 3.40 GHz CPU and 32GB of RAM. Conservation of mass

$$\nabla \cdot \rho \vec{V} = 0 \tag{1}$$

Momentum equation

$$\nabla \cdot \left(\rho \vec{V}\vec{V}\right) = -\nabla P + \nabla \cdot \left(\mu \nabla^2 \vec{V}\right) \tag{2}$$

Energy equation

$$\nabla \cdot \left(\rho \vec{V} C_P T\right) = \nabla P \cdot \left(k \nabla T\right) \tag{3}$$

Geometry of Simulated Model

The concentric double-pipe heat exchanger simulated has a three-start spirally corrugated tube as the inner tube and a smooth tube as the outer tube. Tubes dimensions follow exactly the same geometrical dimensions of tubes available in market with 1mm thickness for spiral corrugation and 2mm for smooth tube. It was set as 1m of length (L) and Table 1 show the characteristics of each tube used. It involves a bore diameter D_b, envelope diameter D_{en}, nominal diameter D_n, corrugation height e and corrugation pitch p measured in mm. Equation (4) represents the formula of calculating the nominal diameter used to get the equivalent diameter as recommended by [14-17].

$$D_n = \frac{\left(D_b + D_{en}\right)}{2} \tag{4}$$

TABLE 1. Characteristics of tubes used

No	D_b (mm)	D_{en} (mm)	D_n (mm)	e (mm)	p (mm)	Tube Type
1	16	18	17	1	30	Three-start spirally corrugated tube
2	-	-	44	-	-	Smooth tube

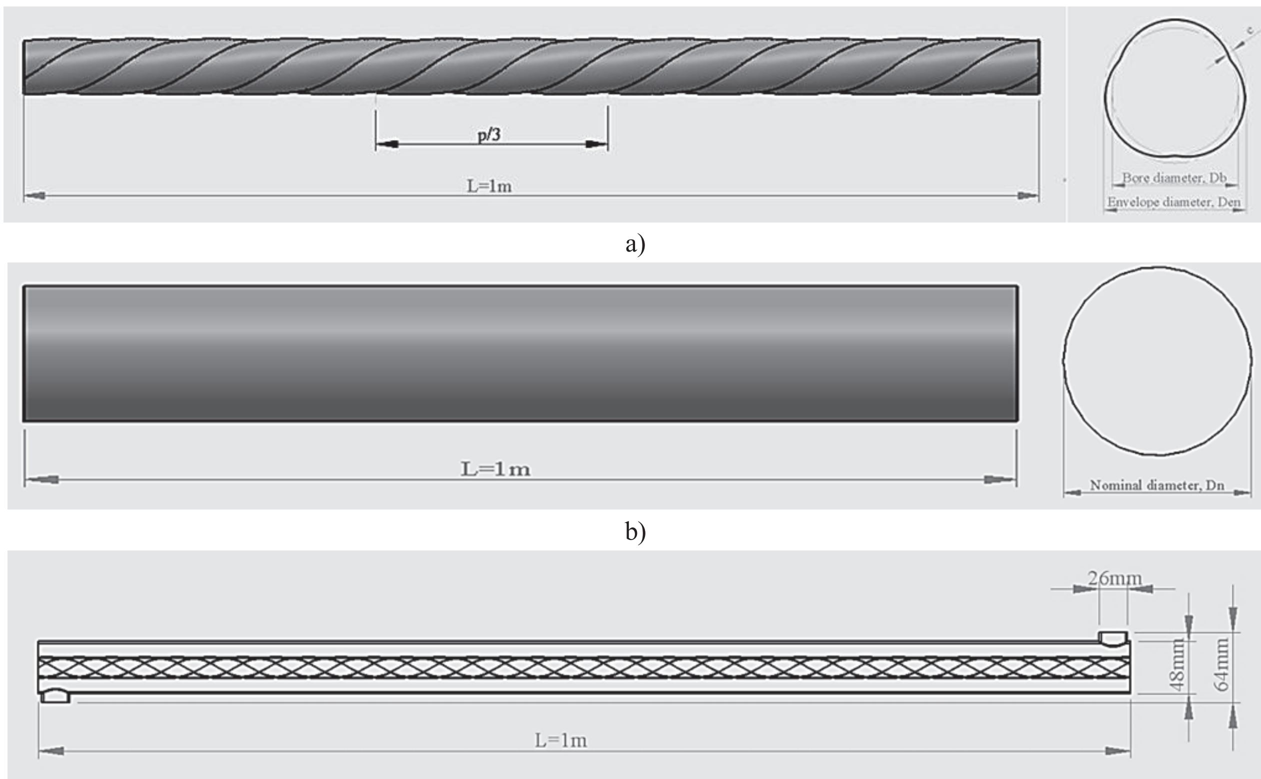
The geometry of three starts spirally corrugated tube and smooth tube as in Fig. 2 were then set up into a double pipe heat exchanger by placing the corrugated tube inside the smooth tube to form a concentric DPHE as presented in Fig. 2(c).

Mesh Parameters

Three gradually refined unstructured tetrahedral meshes with prismatic near wall elements were generated for the model. Meshing process was optimized using three different mesh sizes in order to achieve the most reliable and accurate results, and tetrahedral nodes were selected to further the study. It followed by meshing criteria such as mesh quality and grid density. The stability and accuracy of the computation depends on the quality of meshing which involves the right setting of skewness and aspect ratio of the mesh. These two qualities play a vital role in ensuring that the mesh is good and to avoid error in the simulation. In this case, the skewness and aspect ratio were tightly monitored and it gives 0.8, and 16, respectively.

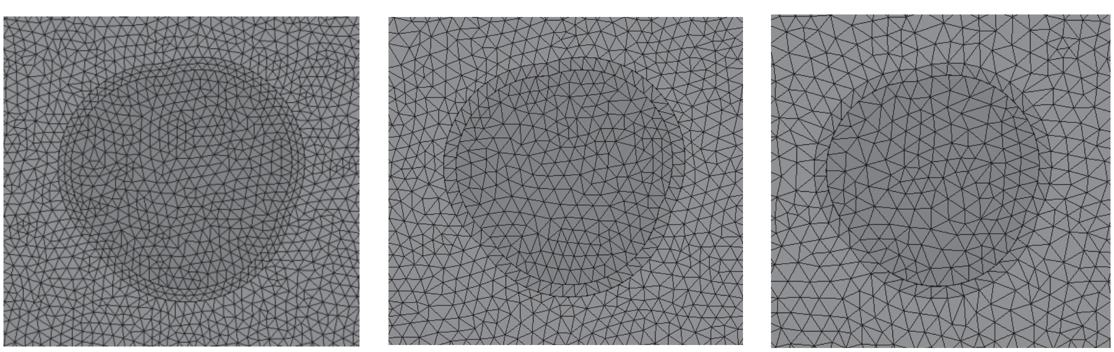
Numerical errors appear between the exact equation and discretization error can be reduced by increasing the spatial grid density. A fine grid is required to capture the swirls and separation occurs in secondary flow regimes. In order to achieve that, mesh refinement studies which included in verification steps were done for different mesh size and it is presented as in Fig. 3 before the choice is finalized. Mesh N_1, N_2 and N_3 represents mesh size of 0.8mm, 1.1mm and 1.4mm, respectively.

Based on the mesh size itself, the number of elements involved would have bigger cell up to trillion and it proves that these meshing has grid range of 1 606 305 to 10 337 326 tetrahedral cells. The adaption of size function was applied from the first cell next to the wall with growth factor of 1.2 set from the wall to the tube core.



(c)

FIGURE 2. Tube geometry of (a) three-start spirally corrugated tube, (b) smooth tube and (c) arrangement of double pipe heat exchanger



(a) (b) (c)

FIGURE 3. Meshing of different grid sizes (a) N_1, (b) N_2 and (c) N_3 for GCI

Boundary Conditions

In the present model, it was assumed that the flow is steady, incompressible and three dimensional (3D); the working fluid is in a single phase and its properties remain constant; gravitational force and radiation are neglected. The boundary conditions of tube were given as follows:

a) Inlet conditions: Velocity and temperature inlet for cold water are $V_{c,in}$ = 0.0591m/s and $T_{c,in}$ = 303K; Velocity and temperature inlet for hot air are $V_{h,in}$ = 0.0156m/s and $T_{h,in}$ = 463K.
b) Outlet conditions: Pressure, P = 0Pa
c) Wall conditions: The constant temperature boundary condition was adopted in the inner tube wall T_w = 393K; coupled boundary conditions on the inside wall of spiral corrugated tube was no slip boundary V_j = V_i = V_k = 0, $T_f = T_w$ and $q_f = q_w$; adiabatic boundary condition was adopted in the outer wall of spiral corrugated tube.

Simulation process only considers hot air flowing in the outside tube and water as the main working fluid flow inside spirally corrugated tube with a Reynolds number of 1000 at constant heat flux at the tube wall of 5000 W/m^2.

CFD VERIFICATION

Solution verification means solving the equations properly mentioned by Roache [18] is actually aim to reduce or minimize source of errors as much as possible in giving better results at the end. Verification start with choosing the solution algorithm, solution methods, residuals convergence and finally getting the most reliable results through GCI presented in the following sub-sections.

CFD software was used to solve the discretized governing Equations 1-3 by generating its temperature, pressure and also velocities vector to the model. Simulations were run as laminar flow following the laminar simulation procedure. Finite volume method (FVM) is a tool that helps in discretizing the governing equation and solved the implicit steady state format. Laminar flow has no model selection in fluent and it only has "laminar" and the optimization will be in spatial difference discretization, and pressure-velocity coupling scheme. To couple the velocity and pressure field, Semi-Implicit Method for Pressure-Linkage Equations (SIMPLE) algorithm is implemented. SIMPLE algorithm was found by Han *et al.* [19] that it gives fast convergence and they also did the second order upwind by that times makes the pressure, temperature and momentum equation in this study are discretized by second order upwind scheme.

Residual convergence is the most fundamental measures as it will directly quantify the error in the solution of the equation system. It will also measure the local imbalance of the variables in each control volume of the CFD analysis. Therefore, each cell in the variable will have individual residual value for each equation solved. The accuracy of the numerical solution is higher when the residual value is lower. Residual is considered to be converged when most of the residuals reach the level of 10^{-4}. In current study, residual monitors for continuity, x-velocity, y-velocity, z-velocity and energy gives $1.1\text{x}10^{-2}$, $4.3\text{x}10^{-4}$, $5.7\text{x}10^{-4}$, $8.4\text{x}10^{-4}$, and $1.1\text{x}10^{-6}$, respectively, as shown in Fig. 4. With that, the simulation is considered as converged.

Grid Convergence Index (GCI)

Grid Convergence Index is one form of the verification process in order to achieve a grid independent solution and it can be done by subsequent refining of the grid. This measure is based on the estimated fractional error that was derived by Richardson extrapolation. The basis of the method is to assume that the results are converging towards the exact solution of the equation system as the mesh is refined with an apparent order of convergence (P) that is in theory proportional to the order of the discretization scheme. The objective of determining GCI value is to determine the expanded uncertainty interval due to the grid for the fine mesh.

Equation 5 and 6 shows the calculations involved in the generated mesh which is the grid refinement ratio r and cell or mesh or grid size h for three-dimensional model.

$$r = \frac{h_{coarse}}{h_{fine}} \tag{5}$$

$$h = \left[\frac{1}{N} \sum_{i=0}^{N} (\Delta V_i) \right]^{\frac{1}{3}} \tag{6}$$

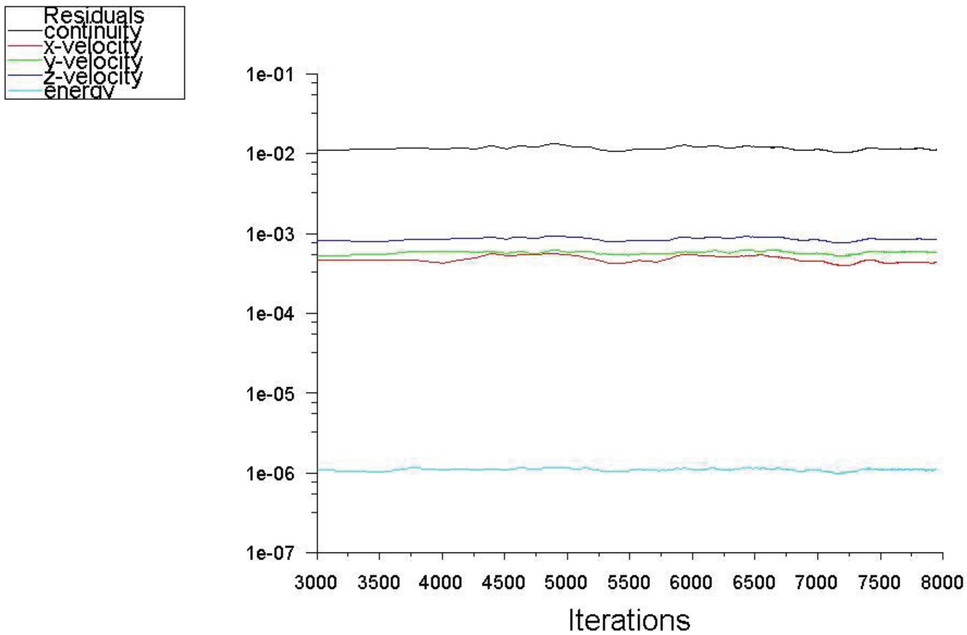


FIGURE 4. Residual convergences.

Where ΔV_i is the volume of the ith cells, and N is the number of cells used for the computations. Considering the representative grid sizes $h_1 < h_2 < h_3$ and grid refinement ratio $r > 1.3$, the apparent order of convergence P can be determined by Equations (7), (8) and (9).

$$P = \frac{1}{\ln(r_{21})} \left| \ln \left| \frac{\varepsilon_{32}}{\varepsilon_{21}} \right| + q(P) \right| \tag{7}$$

$$q(P) = \ln \left(\frac{r_{21}^P - s}{r_{32}^P - s} \right) \tag{8}$$

$$s = 1 \cdot \mathrm{sgn} \left(\frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \tag{9}$$

Where $\varepsilon_{32} = \phi_3 - \phi_2$, $\varepsilon_{21} = \phi_2 - \phi_1$, sgn is the signal function calculated as follows:

$$\mathrm{sgn} = \begin{cases} -1; x < 0 \\ 0 \quad ; x = 0 \\ 1 \quad ; x > 0 \end{cases} \tag{10}$$

With the value of P, the expanded uncertainty GCI can be calculated using Eqs (11) using an empirical Factor of Safety, Fs, equal to 1.25.

$$GCI_{fine}^{21} = \frac{1.25e_a^{21}}{r_{21}^P - 1} \tag{11}$$

TABLE 2. Grid Convergence Index (GCI)

	Nusselt Number
N_1, N_2, N_3	10 337 326, 4 446 746, 1 606 305
h_1, h_2, h_3	0.5288, 0.7006, 0.9837
r_{21}, r_{32}	1.33, 1.40
ϕ_1, ϕ_2, ϕ_3	0.2447, 0.2450, 0.2486
P	8.66
$\phi_{ext}^{21}, \phi_{ext}^{32}$	0.2446, 0.2448
e_a^{21}, e_a^{32}	0.12%, 1.47%
$e_{ext}^{21}, e_{ext}^{32}$	0.01%, 0.08%
$GCI_{fine}^{21}, GCI_{coarse}^{32}$	0.02%, 0.10%

For each mesh size, GCI computed gives 0.02% and 0.1% for fine and coarse mesh, respectively, as shown in Table 2. Based on the GCI results, a reduction in GCI was observed in the three variables which give $GCI_{fine}^{21} < GCI_{coarse}^{32}$. All the three mesh sizes can be used in running the simulation but mesh with 0.8mm size is the most preferable to use because it gives better GCI value and computer used has the ability to simulate using the same number of element of 10 million.

CONCLUSION

Verification of heat transfer enhancement in tube with spiral corrugation is the process of minimizing source of errors in the code and model equation involving spirally corrugated tube. The idea of verification remains the same which is simulating heat exchanger procedure using commercial CFD software. Three mesh sizes of 0.8mm, 1.1mm and 1.4mm were used to find the best results. Following the GCI procedure, it were then denote as fine and coarse GCI give 0.02% and 0.10%, respectively. The reduction between coarse and fine GCI value shows that it can give the most reliable results starts with choosing the right solution to solve governing equation through solution algorithm, solution method and finally choosing the right residual convergence. According to Richardson extrapolation, the independent solution has been achieved with the GCI value of less than 5%. Mesh size of 0.8mm was chosen as the most preferable size and computer used has the ability to simulate using the same number of element.

ACKNOWLEDGMENTS

This research was funded by a grant from Ministry of Higher Education of Malaysia and Universiti Teknologi Malaysia (FRGS Grant R.J130000.7824.4F832). The data presented, the statement made, and views expressed are solely the responsibility of the authors.

REFERENCES

1. Cengel, Y. A., Ghajar, A. J., & Kanoglu, M (2015). Heat and mass transfer: fundamentals and applications *McGraw Hill Education*, New York.
2. Greitzer, E. M., Tan, C. S., & Graf, M. B (2004). Internal flow concepts and applications. *Cambridge University Press*, Cambridge, U.K.
3. Kanade Rahul H., Kailash B A and Gowreesh (2015). Heat transfer enhancement in a double pipe heat exchanger using CFD. *International Research Journal of Engineering and Technology (IRJET),* pp. 419-430.

4. Hyder H. Balla (2015). Enhancement of heat transfer in six-start spirally corrugated tubes. *Case studies in Thermal Engineering*, 9, 79-89.
5. Kareem, Z. S., Jaafar, M. M., Lazim, T. M., Abdullah, S., & Abdulwahid, A. F. (2015). Passive heat transfer enhancement review in corrugation, *Experimental Thermal and Fluid Science*, 68, 22-38.
6. M. Omidi, Mousa Farhadi & M. Jafri (2017). A comprehensive review on double pipe heat exchangers, *Applied Thermal Engineering,* 110: 1075-1090.
7. Ranjith and Shaji K. (2016). Numerical analysis on a double pipe heat exchanger with twisted tape induced swirl flow on both sides. *International Conference of Emerging Trends in Engineering, Science and Technology (ICETST-2015),* 436-443.
8. Chirag Maradiya, Jeetendra Vadher and Ramesh Agarwal (2018). Heat transfer enhancement techniques and their thermal performance factor. *Journal of Basic and Applied Sciences*, Beni-Suef University, 7, 1-21.
9. Hamed Sadighi Dizaji, Samad Jafarmadar and Farokh Mobadersani (2015). Experimental studies on heat transfer and pressure drop characteristics for new arrangements of corrugated tubes in a double pipe heat exchanger. *International Journal of Thermal Sciences*, 96, 211-220.
10. Huai-Zhi Han, Bing-Xi Li, Hao Wu and Wei Shao (2015). Multi-objective shape optimization of double pipe heat exchanger with inner corrugated tube using RSM method. *International Journal of Thermal Sciences*, 90, 173-186.
11. Cong Qi, Yong-Liang Wan, Chun-Yang Li, Dong-Tai Han and Zhong-Hao Rao (2017). Experimental and numerical research on the flow and heat transfer characteristics of TiO_2-water nanofluids in a corrugated tube. *International Conference of Heat and Mass Transfer*, 115, 1072-1084.
12. Mohammad Hussein Bahmani, Ganbarali Sheikhzadeh, Majid Zarringhalam, Omid Ali Akbari, Abdullah A. A. A. Alrashed, Gholamreza Ahmadi Sheikh Shabani and Marjan Goodarzi (2018). Investigation of turbulent heat transfer and nanofluid flow in a double pipe heat exchanger. *Advanced Powder Technology*, 29, 273-282.
13. I. Imani (2018). Heat transfer enhancement in tube with spiral corrugation, Master thesis, University Teknologi Malaysia.
14. Rainieri. S. Farina. A. & Pagliarini. G. (1996). Experimental investigation of heat transfer and pressure drop augmentation for laminar flow in spirally enhanced tubes. *Proceedings of the 2nd European Thermal-Sciences and 14th UIT National Heat Transfer Conference*, 203–209.
15. Rainieri. S. & Pagliarini. G. (2002). Convective heat transfer to temperature dependent property fluids in the entry region of corrugated tube. *International Journal of Heat and Mass Transfer,* 45, 4525-4536.
16. Barba. A., Rainieri. S. & Spiga. M. (2002). Heat transfer enhancement in corrugated tube. *International Communications in Heat and Mass Transfer,* 29, 313-322.
17. Saha. S. K. (2010). Thermohydraulics of laminar flow through rectangular and square ducts with axial corrugation roughness and twisted tapes with oblique teeth. *Journal of Heat Transfer*, 132, 081701.
18. Roache P.J. (1994). Perspective: A Method for Uniform Reporting of Grid Refinement Studies. *Journal of Fluids Engineering,* 116,405-413.
19. Han. H., Li. B., Wu. H. and Shao. W. (2015). Multi-objective shape optimization of double pipe heat exchanger with inner corrugated tube using RSM method. *International Journal of Thermal Sciences*, 90,173-186.