Numerical Study of Reynolds Number Effect in Fluid Jet Impingement Cooling Heat Transfer On Concave Surface

Azfar Ruslan[a], and Nadia Kamaruddin[a]

[a]Faculty of Chemical Engineering, Universiti Teknologi MARA, Selangor, Malaysia

Abstract

Generally jet impingement is a technology of providing a medium for heat transfer on surface that is to be cooled by impinging fluid on it. This method of cooling has high heat transfer capability. This technology is applied in micro-electronic industries, gas turbine cooling, quenching process in chemical plant and so forth. However, most of the research of jet impingement are done on flat surface, the knowledge regarding this technology on concave surface are not abundant.

Thus, to increase the viability of this technology on various application and increase industrial process efficiency that relies on rapid cooling on cylindrical surfaces, this study will be carry out. The scope of this study will be on evaluating the effects of Reynolds number on the cooling heat transfer rate on concave surface. The Re= 5000, 10000, 15000, 20000, 25000 and 30000 will be used and the value of Nusselt number will be evaluated to determine the heat transfer. To carry out this study we will use the aid of simulation software. CFD ANSYS FLUENT will be used to simulate the jet impingement system by using standard equation of motion and the Navier-Stokes standard k-Ɛ equation model is used. The results obtained are Nu = 28.74, 44.67, 56.82, 67.30, 76.58 and 84.9 with respective to the increment of Reynolds number. In conclusion it can be said that with higher Reynolds number, Nusselt number will increase which as well indicates high heat transfer.

Article Info

Article history:

Received date: 06 Dec 2019
Accepted date: 06 Dec 2019

Keywords:

Reynolds number
Nusselt number
Jet impingement
Heat transfer
Concave surface

I. Introduction

Jet impingement is a method that provide heat transfer rate. Because of this method capability of high heat and mass transfer on targeted specific location , jet impingement cooling is widely used in the industry that requires temperature control such as in food processing ,drying of paper, cooling gas turbine blades, cooling of reactor, and electrical component [1].

Jet impingement provide effective and flexible way to transfer mass and heat, we need to identify suitable method to predict its performance. The proper and right configuration of jet impingement parameters and application technique can efficiently increase transfer of thermal energy between the jet fluid and targeted location [2].

Generally Jet Impingement is a technology of providing a medium for heat transfer on surface that is to be cooled by impinging fluid on it. This method for cooling has high heat transfer capability normally is applied in micro-electronic industries, gas turbine cooling, quenching process in chemical plant, reactor cooling to prevent temperature runaway and so forth.

Many researches have been done to study jet impingement, mainly on the effects of Reynolds Number, Nusselt Number, the nozzle geometry of the jet, the distance of the nozzle and other parameters on flat surface that could affects the heat transfer rate. However, the knowledge regarding this method of heat transfer and flow characteristics is not abundant, mainly liquid impingement concave surfaces.

According to experimental research done by Cornaro [3]by using smoke wire to visualize impingement on concave surface. Generally, for this type of impingement it has unsteady flow structure when compared to flat and convex surfaces. This is due to the fact that the interaction of the flow and the concave surface generates centrifugal forces that destabilized the flow.Yue Tzu,[1] did a numerical investigation on turbulent slot jet impingement cooling on a semi-circular concave surface. The numerical simulation was conducted with Reynolds number Re= 5,920-23,700, jet to surface distance 0.5 -12 jet width 0.033-0.05 with two different heat flux of 1663-5663 W/m^2. The theoretical model used was Stoke k-Ɛ model equation. The local Nusselt number variation obtained was significantly affected by the jet-exit Reynolds Number. From his study convection intensity and turbulence level increase with increment in Reynolds number which in return enhance heat transfer rate. Other than that, the local Nusselt number is at highest at impingement zone at any value of Reynolds number. The k-Ɛ model predict accurate thermal data [1].

Flow visualization and heat transfer characteristics of liquid jet impingement study conducted by Jafar [4]the numerical model obtained by using finite volume approach together with volume of fluid method. In his study CFD FLUENT used for 2D simulations was implemented. The area of the study was flat and curved surface under parameter of S=40, W=4, D=8, S/D= 5, X/D= 3.125, Reynolds Number= 250. From the study it is found that Nusselt number is higher in the impingement zone located at the centers of both surfaces and are of the same value both cases. This is because the velocity and film thicknesses are equal on both surfaces. On top of that increasing the Reynolds number will in return increases the diameter of the cooling zone around the stagnation point significantly and will decrease its temperature.

According to Singh,[5] study on double circular air jet impingement cooling of a heated circular cylindrical surface both numerically and experimentally for different values of Red, h/d, s/d and Rc/R. Effects of Reynolds number on the cooling of the cylinder without confinement for Re= 10,000 and 25,000, for s/d = 4 and 20 was studied. The difference from both Reynolds number was that the difference in thermal boundary layer on the cylinder, the thermal boundary layer for Re=25,000 is lesser compared to Re=10,000 and lower thermal boundary layer means that heat transfer rate is increased.

The aim of this study is to develop computational fluid dynamics model of jet impingement on concave surface by using CFD ANSYS. The selected range of Reynolds number is from 5,000, 10000, 15000, 20000 25000 and 30,000 which is turbulent flow, thus turbulence equation model is used with constant heat flux of q =5663W/m2.

II. Methodology

Numerical Method

The related equation to be used for turbulence modelling of the motion of the fluid is Navier-Stoke equations which represent the physical laws of mass and momentum conservation. The equation is as follows:

Continuity: $$\frac{\partial U_i}{\partial x_i} = 0 \qquad (1)$$

Momentum: $$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[v\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \overline{u_i u_j} \right] \qquad (2)$$

Temperature: $$U_j \frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\gamma \frac{\partial T}{\partial x_j} - \overline{u_j \theta} \right) \qquad (3)$$

For the simulation software CFD-ANSYS is used, the step flow chart to create successful simulation was taken from Introduction to Computational Fluid Dynamics by Versteeg[6].

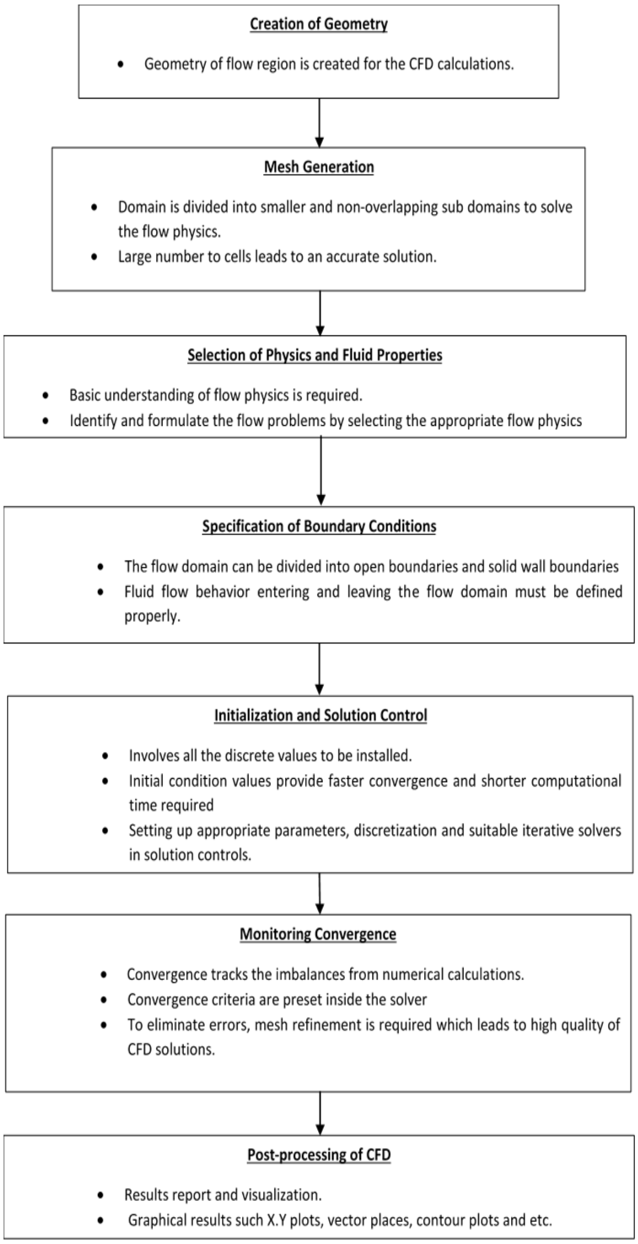

Fig. 1: Flow chart to simulate computational fluid design

Geometry

The numerical computations set up was made with geometry of semi-circular concave wall with diameter of 75mm with nozzle at centre with diameter of 12mm and distance of nozzle from targeted wall surface of H/B = 2. The geometry shape similar to Yue Tzu

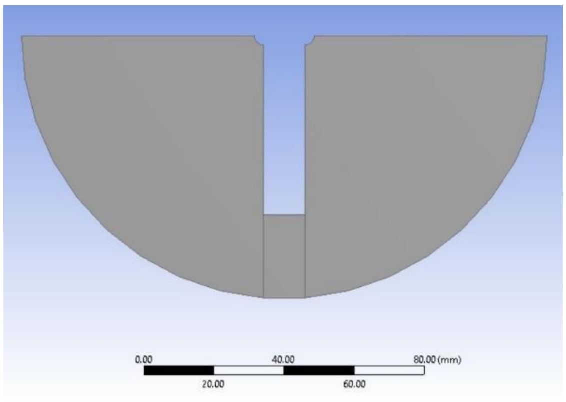

Fig. 2: Geometry of jet impingement of semi-circular concave wall

Meshing

Meshing is done to obtain accurate solutions. Meshing influences the accuracy, convergence and speed of the simulation. Meshing is a set of points and cells connected to form a network. This network can be in many forms of geometry. Each cell holds local equation for the solutions to be calculated. The finer and higher quality mesh will obtain more accurate solution output. The meshing for this present study is done with element size of 0.00055 and the quality check for skewness is to be lower than 0.05.

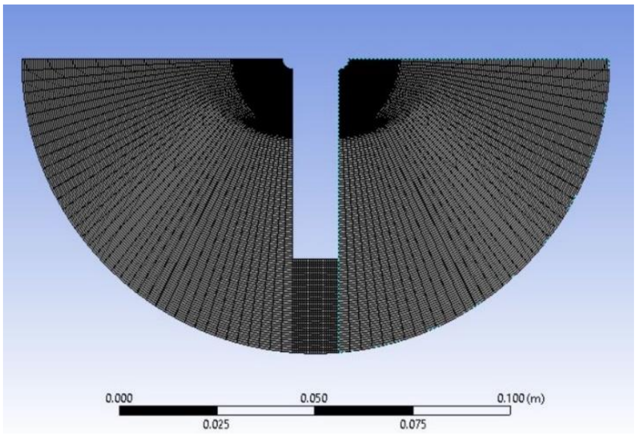

Fig. 3: Meshing of geometry at element size 0.00055

The FLUENT ANSYS setup was using standard equation of motion and the Navier-Stokes standard k-Ɛ equation model. The cooling fluid is air and the targeted wall surface is steel and are set on the geometry boundary conditions the inlet, outlet, wall surface and fluid body. For this present study the solution method used was SIMPLE and the initialization method is hybrid. The iterations results should be lower than $1x10^{-4}$

III. Results and discussion

Validation

Validation is made before proceeding with the simulation process, without validation the simulation made could be totally wrong or out of range of scope. The simulation can be proceeded once the validation shows satisfying match with previous study. The validation is done by using numerical study of Yue Tzu [1] where the working fluid is air with constant heat flux and experimental study of Yang[7]. The same Reynolds number of 23700 and H/B of 0.5, 4, 8 ,12 with diameter of 5mm was taken to mimic their study. The validation result obtain from our simulation was quite similar and agreeable refer figure xx and table xx

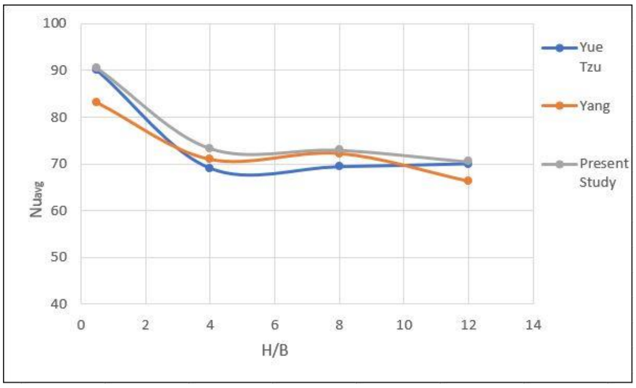

Fig. 4: Comparison of average Nusselt number with Yue Tzu [1] and Yang [7] with Re = 23700

Table 1: Values obtained from the validation.

H/B	Yue Tzu	Yang	Present
0.5	90	83.125	90.53278
4	69.0625	70.9375	73.39298
8	69.375	72.1875	73.02343
12	70	66.25	70.50916

The validation curve deviates slightly from the previous study but the trends are the same. The deviations exist with the numerical study was that could due to the slight different in meshing of Yue Tzu [1]. All in all, the difference was not drastic thus this validation is acceptable and seems to be good enough to proceed with the current parameters study.

Effects of Reynolds Number

Reynolds number impacts significantly on the cooling process of the targeted wall surface. This can be seen that by the increment of Reynolds number from 5000 to 30000 the Nusselt number increase as well almost exponentially Where Nusselt number is the ratio of the thermal energy convected to the fluid to the thermal energy conducted within the fluid and it provides a measure of the convection heat transfer occurring at the surface.

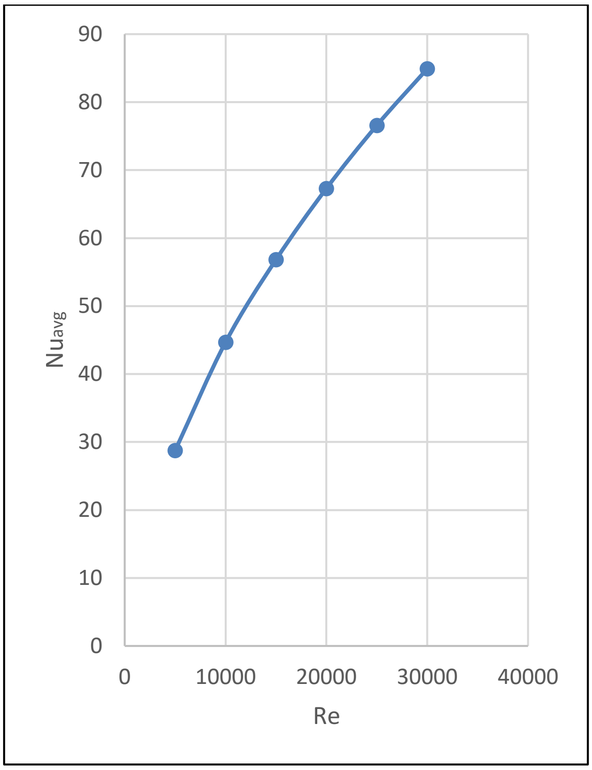

Fig. 5: Correlations between Reynolds Number and Nusselt Number

Table 2: Nusselt number obtained from different Reynolds number

Re	v	Nu
5000	7.46	28.74289
10000	14.91	44.67637
15000	22.37	56.82124
20000	29.82	67.29727
25000	37.28	76.58154

The data also shows that the flow rate will increase the fluid momentum. With the higher Reynolds number, it can also be observed that fluid with high velocity magnitude able to spread wider until the edge of the targeted wall surface compared to the lower Reynolds number. This can be observed in the figure 6 ,7, and 8 where the red region indicates the fluid velocity is high. With higher velocity magnitude fluid spreads through, this will increase the heat transfer rate with corresponding to the increasing Nusselt number. One of the reasons the Nusselt number is high could be the fact that convection strength increase with the turbulence kinetic energy which in return will increase the heat transfer rate. This can be seen from the figure 9, 10 and 11 with high Reynolds number there is significant increment turbulence kinetic energy.

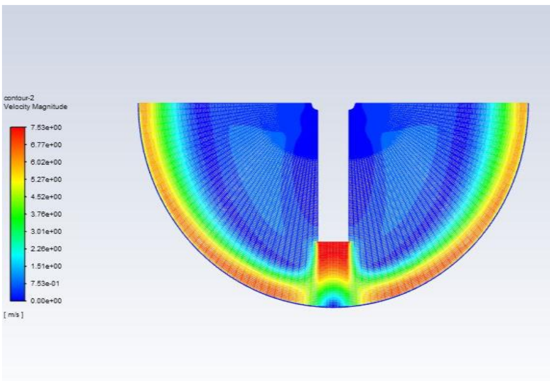

Fig. 6: Highest velocity magnitude of 7.53 m/s at Re=5000

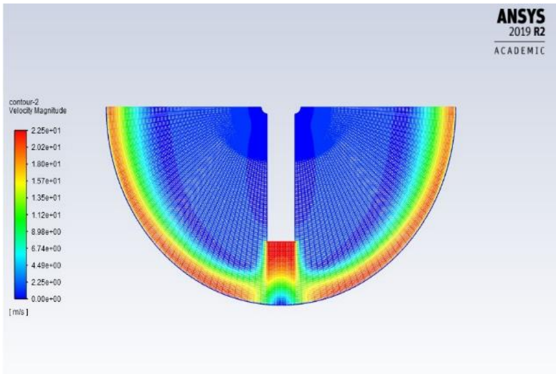

Fig. 7: Highest velocity magnitude of 22.5 at Re=15000

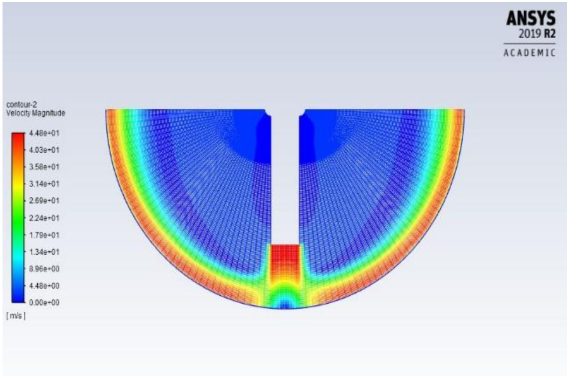

Fig. 8: Highest velocity magnitude of 44.8 m/s at Re=30000

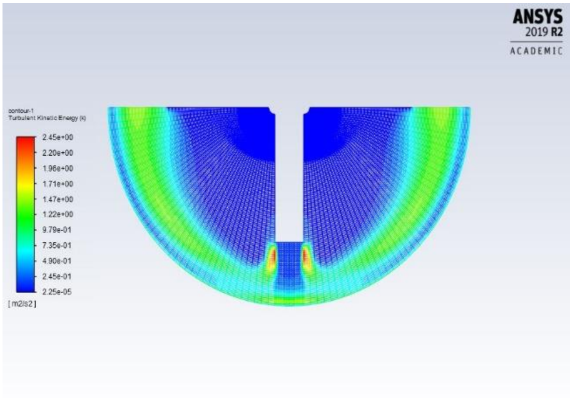

Fig. 9: Highest turbulence kinetic energy of 2.45 m^2/s^2 at Re=5000

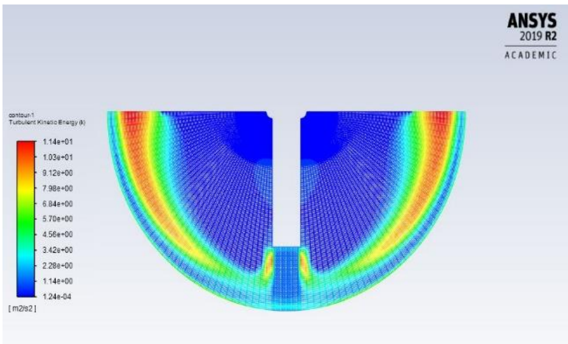

Fig. 10: Highest turbulence kinetic energy of 11.4 m^2/s^2 at Re 15000

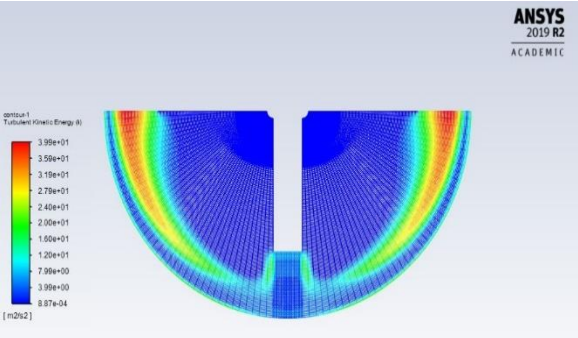

Fig. 11: Highest turbulence kinetic energy of 39.9 m^2/s^2 at Re=30000

IV. Conclusion

The heat transfer properties of the fluid jet impingement on concave surface was conducted numerically using CFD ANSYS with Reynolds number ranging from 5000 to 30000 for validation purpose on Re = 237000 is used. From the present study it can be seen that with the increment of Reynolds number the Nusselt number increases from Nu = 28.74, 44.67, 56.82, 67.30, 76.58 and 84.9 which this indicates that the heat transfer rate increases due to contributions from the fluid velocity magnitude and turbulence kinetic energy increase the convective strength.

Acknowledgement

Thank you, Madam Nadia, and UiTM for helping me out with this research to its completion.

References

[1] Y. T. Yang, T. C. Wei, and Y. H. Wang, “Numerical study of turbulent slot jet impingement cooling on a semi-circular concave surface,” *Int. J. Heat Mass Transf.*, vol. 54, no. 1–3, pp. 482–489, 2011.

[2] N. Zuckerman and N. Lior, “Jet impingement heat transfer: Physics, correlations, and numerical modeling,” *Adv. Heat Transf.*, vol. 39, no. C, pp. 565–631, 2006.

[3] C. Cornaro, A. S. Fleischer, and R. J. Goldstein, “Flow visualization of a round jet impinging on cylindrical surfaces,” *Exp. Therm. Fluid Sci.*, vol. 20, no. 2, pp. 66–78, 1999.

[4] F. A. Jafar, G. R. Thorpe, and Ö. F. Turan, “Flow visualization and heat transfer characteristics of liquid jet impingement,” *Int. J. Comput. Methods Eng. Sci. Mech.*, vol. 13, no. 4, pp. 239–253, 2012.

[5] D. Singh, B. Premachandran, and S. Kohli, “Double circular air jet impingement cooling of a heated circular cylinder,” *International Journal of Heat and Mass Transfer*, vol. 109. pp. 619–646, 2017.

[6] H. K. Versteeg and W. Malaskekera, “An Introduction to Computational Fluid Dynamics: The Finite Volume Method,” *Fluid flow handbook. McGraw-Hill …*. p. 517, 1995.

[7] G. Yang, M. Choi, and J. S. Lee, “An experimental study of slot jet impingement cooling on concave surface] e}ects of nozzle con_guration and curvature.”