Research Progress in Mechanical and Manufacturing Engineering Vol. 2 No. 2 (2021) 997-1002

RPMME

Homepage: http://publisher.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Two-Dimensional Numerical Study of Two Slot Jets Impinging On Flat Surfaces at Various Angles

Muaz Mohd Zahidin [1], Suzairin Md Seri [2,*]

[1,2]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Parit Raja, Batu Pahat, 86400, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.02.108
Received 25 July2021; Accepted 25 Nov.2021; Available online 25 December 2021

Abstract: Jet impingement cooling systems are currently utilized in many sectors. To develop a more effective jet impingement cooling system, numerous ways were examined. This thesis investigates the effect of nozzle angle on jet impingement to determine a heat transfer relationship. This research also investigates the spacing distance between nozzles edge to the impingement surface and Reynolds number at various angles. These investigations are required in addition to the current research efforts for future cooling system development in global industries. Three primary factors were varied in the experiment such as the nozzle angle (30°, 60°, and 90°), the ratio distance between the nozzle's edge and the impinge surface to the width of the nozzle (r/B = 5, 8, and 16) and the Reynolds number (Re = 500, 1000,2000,10000 and 20000). The target surface is at 336 K, or 62.85 ºC, and cooled down by the flow of air from the nozzle. The heat transfer coefficient after cooling is measured and collected. The results show the relationship between the nozzle angle of jets and the heat transfer rate. A higher heat transfer coefficient is observed when the angle of the nozzle approaches the normal line as compared to the lower angles of the nozzle. In addition, a high heat transfer is produced by a high Reynolds number and the heat transfer shows a decrease in rate when the r/B or the ratio of the distance between the nozzle's edge and the impinge surface to the width of the nozzle increases, implying that the systems will be cooled effectively when the nozzle is near to the target surface.

Keywords: Jet Impingement, Numerical Analysis, Nozzle Angle

1. Introduction

Impinging jets are recognized as a method to achieve particularly high heat transfer coefficients and are used in many engineering applications. In a range of uses, impinging jets are used to transfer heat, including paper drying and turbine blade cooling, de-ice aircraft wings in cold weather, and to cool sensitive electronic devices. We attempt to understand the mechanism of the distributed heat on the surface to define preferred methods to predict jet performance. Huge volumes of thermal energy can be transmitted effectively by the directed liquid or gaseous flow emitted on a surface, demonstrating the

**Corresponding author: suzairinr@uthm.edu.my.*

ability to cool high heat flux surfaces. Jet impingement studies aim to measure the average heat transfer coefficient, even though it is understood that heat transfer coefficients change as a function of the distance from the region of effect. High-velocity jet impingement has since become an established technique for surface cooling or heating in a wide variety of thermal management processes and applications. The use of impingement jets is widespread for cooling modern aero-engine components, particularly within the hot stationary components [1]. An extensive study on steady impinging jets has been conducted to consider their heat and mass transfer characteristics. Numerous experiments and reviews on the issue of steady jet heat transfer have been published over the last few decades to further boost thermal efficiency [2].

2. Methods

2.1 CFD simulation

Specifications and properties of materials, equipment, and other resources used in the current study should be described in this section.

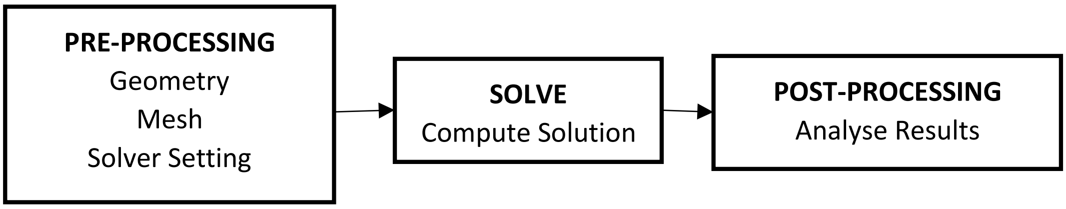

Figure 1: CFD process overview.

2.2 Jet impingement model

The study of laboratory experiments concludes with a section dedicated to impingement array optimization studies. The major objective of the numerical work analysis is on calculating the precision of existing CFD tools in calculating local multi-jet heat transfer rates.

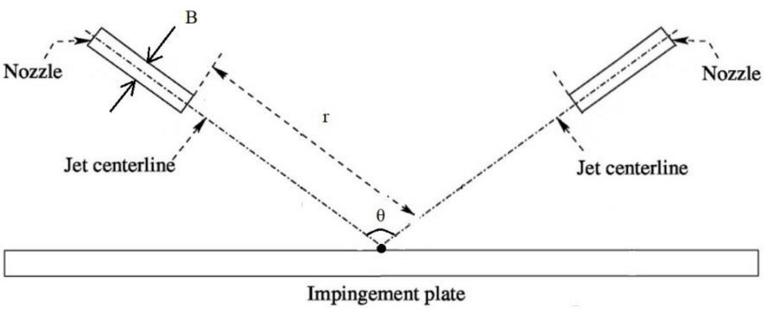

Figure 2: Example of presenting data using a figure

2.3 Geometry and computational domain

Figure 3 shows the computational domain drawn in the Ansys DesignModeler to simulate the jet impingement and study the parameters that will affect the heat transfer of the heated plate. The distance for the heated plate,x is 0.2 m, nozzle width, B is 0.003175 m, and the angle,θ is varied from 30°- 90°. The meshing for the physical domain was obtained by using ANSYS Meshing Software.

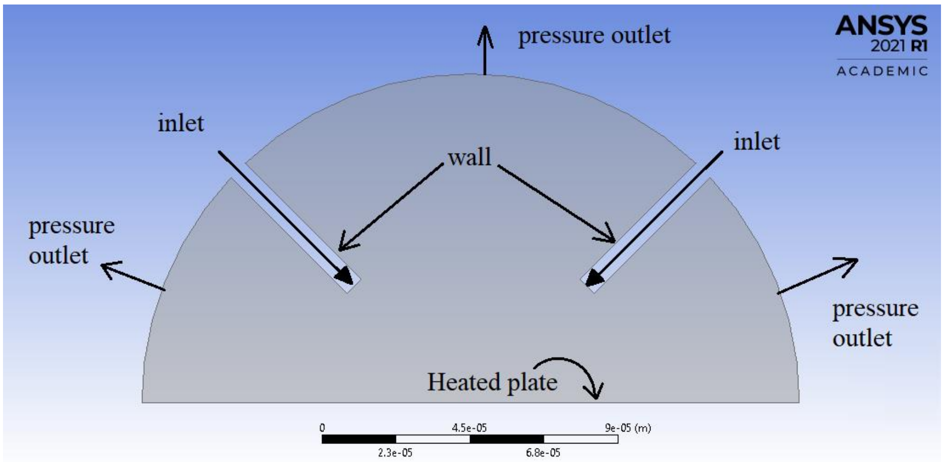

Figure 3: The computational domain

2.4 Meshing

Meshing is a process where it refines (smaller cells) for high solution gradients and fine geometric detail. This process gives accuracy and stability deteriorate as mesh cell deviates from its ideal shape. Global mesh controls are used to make global adjustments in the meshing strategy, which includes sizing functions, inflation, smoothing, defeaturing, parameter inputs, and assembly meshing inputs.

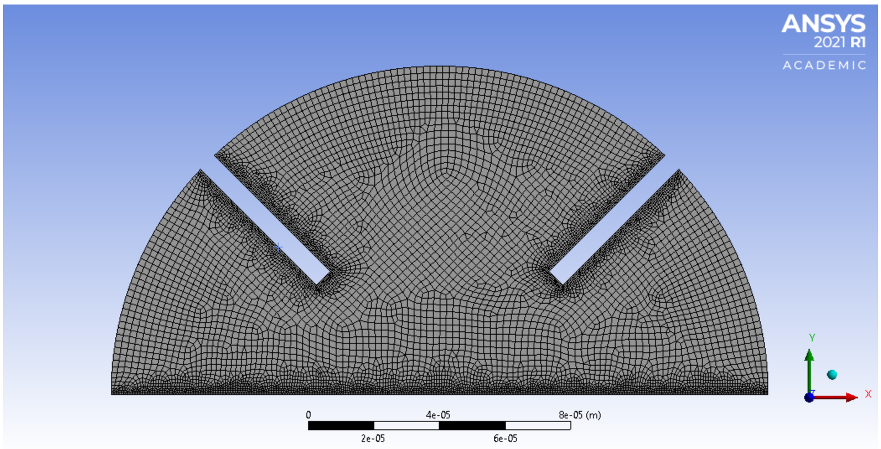

Figure 4: Meshing for jet impingement

3. Results and Discussion

This chapter aims to analyze and discuss explicitly the results obtained from the experimental, numerical, and CFD simulations. Results of ANSYS's simulation were analyzed. The potential performance of the 2-D slanting impingement jet was numerically examined and all designs were analyzed from the model that had been simulated to understand the behavior of the heat transfer. The effects of geometrical and operating parameters on various angle impingement jet performances were numerically assessed.

3.1 Simulation validation

Validation is the most crucial process in simulation to verify the results obtained from the simulation are acceptable and ensure that the project is on the right track. The acquired result is plotted in a graph, and it will be validated against past research. If the graph pattern revealed similarity from the comparison side, the results are said to be reliable. The data utilized to validate this study came from a previous study by Gardon (1966) [3].

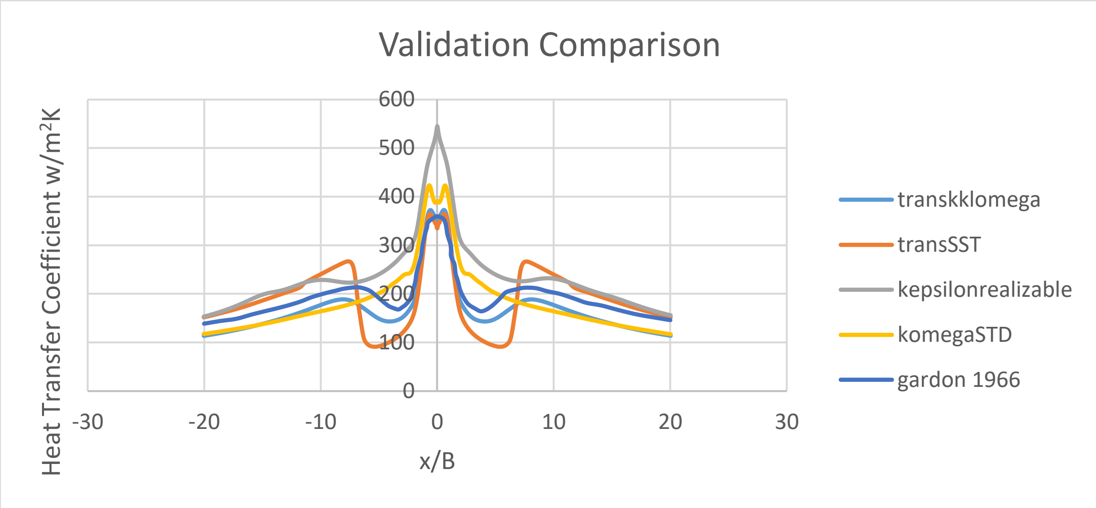

Figure 5: Comparison between Gardon 1966 [3] and simulation results

The Transition k-kl-omega model can predict the trend of the convective heat transfer coefficient with good accuracy with the respect to the experimental measurements, while a notable discrepancy has been found with other turbulent models. For this reason, other models are not suitable for this setup and are less suitable for the slot impingement application, unlike the Trans k-kl-omega. The deviations of the results to the experimental data are reported in Table 1. To evaluate the error, Normalise Root Mean Squared Error, NRMSE, approach has been used.

Table 1: Error concerning Gardon 1966

Models	NRMSE	NRMSE (%)
Trans k-kl-omega	0.126369	12.63694
Trans-SST	0.213326	21.33258
K-epsilon-Realizable	0.27593	27.59304
K-omega-STD	0.171592	17.15917

3.2 Grid independence test

Grid independent test or GIT is a crucial part of the validation process. This step is vital to achieving a precise and valid result in the CFD process by perpetuating the GIT as low as possible without disturbing the result.

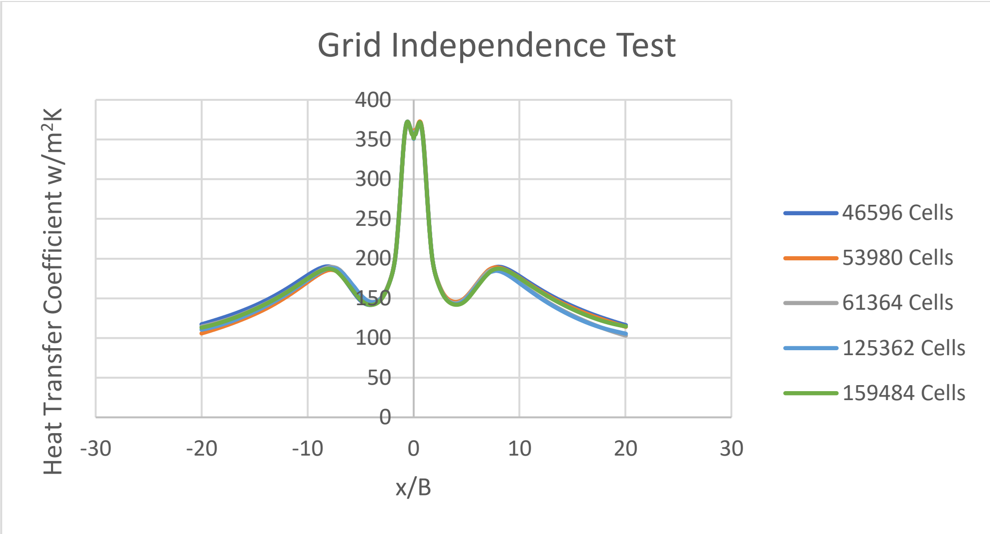

Figure 6: Grid independence test

3.3 Summary of the results

The results obtained from this numerical experiment show that the angle does affect the heat transfer coefficient of the jet impingement. From Figure 7 clearly shows that the heat transfer increase as the angle of the slot nozzle increase.

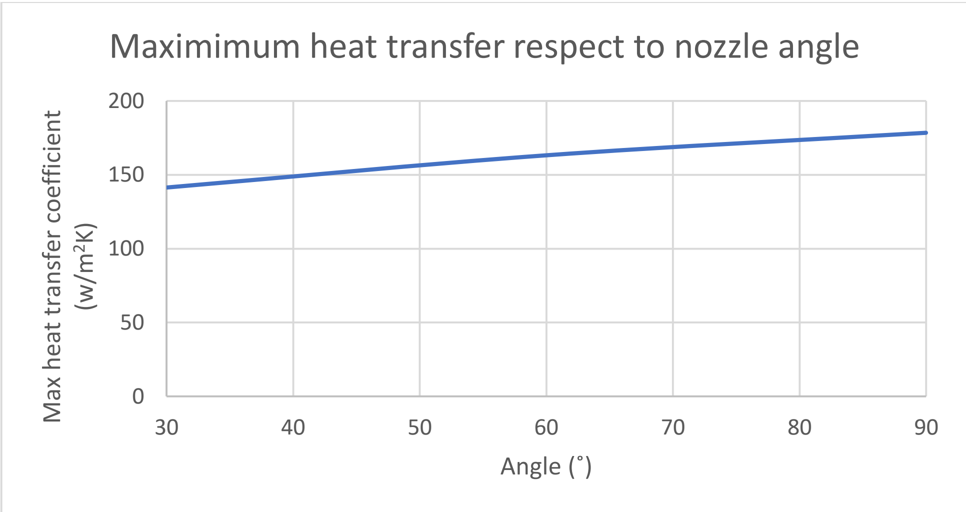

Figure 7: Maximum heat transfer to nozzle angle

4. Conclusion

From the review of heat transfer characteristics on the parameters of jet inclinations, jet Reynolds numbers, and dimensionless jet-to-plate distances, the following remarks can be outlined. A commonly increase trend of heat transfer coefficient with increasing nozzle jet angle, θ about 15 w/m^2K. The heat transfer also shows an increase in trend when the Reynolds number increases, due to the velocity of the jet being relatively high at a higher Reynolds number. Some of the results show that the location of the peak moves away from the center of the impingement area due to an increase in the nozzle jet angle, θ. When the r/B, ratio of the jet-to-plate and the width of the nozzle is getting far from the surface the value of the heat transfer coefficient decrease.

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, University Tun Hussein Onn Malaysia for its support.

References

[1] J. C. Han, J. S. Park, and C. K. Lei, P.P. (1985) "Heat transfer enhancement in channels with turbulence promoters,".JournalofEngineeringforGasTurbines and Power.Vol. 107. 628 – 635

[2] Kondjoyan, A., Peneau, F. and Boisson, H.C.(2002) "Effect of High Free Stream Turbulence on Heat Transfer between Plates and Air Flows".A Review of Existing Experimental Results. Int. Journal of Thermal Science.Vol. 41. 1-16.

[3] Gardon, R.; Akfirat, J.C. Heat transfer characteristics of impinging two-dimensional air jets. J. Heat Transf. 1966, 88, 101–108.

[4] Beitelmal A.H., Saad M.A., Patel C.D., The effect of inclination on the heat transfer between a flat surface and an impinging two-dimensional air jet. International Journal of Heat and Fluid Flow, 2000, 21(2): 156–163.

[5] Attalla M., Salem M., Heat transfer from a flat surface to an inclined impinging jet. Heat and Mass Transfer, 2014,50(7): 915–922.

[6] Jambunathan, K., Lal, E., Moss, M., A., and Button, B. L. (1992). "A Review of Heat Transfer Data for Single Circular Jet Impingement.".Int. J. Heat and Fluid Flow. Vol. 13. 106 - 115