# Numerical Study of Nozzle Diameter Effect in Fluid Impingement Cooling on Cylindrical Surface.

Nurul Farihin binti Mohd Sabri, Nadia binti Kamarrudin

*Faculty of Chemical Engineering, Universiti Teknologi Mara*

*Abstract*—**Jet impingement cooling method is an improved technique used to cool down the equipment and appliances in the industries and plants. This upgraded technique is replacing the old technique method by increasing the heat transfer rate and high efficiency. In this research, air was used as the cooling medium for cooling the equipment that have a curvature surface as most industrial equipment have a cylindrical shape. The objective of this study is to develop computational fluid dynamics model of jet impingement heat transfer on nozzle diameter effect towards curvature surface. The nozzle diameter B/D of 0.013, 0.027, 0.04, 0.053, 0.067 and 0.08 were used with constant Reynold number at 23700. The temperature of the water outlet of the jet is at 298.15 K and the heat flux q″ is at 5633 W/m$^2$. CFD simulation was run and using (k-Ɛ) turbulence model. The result is shown and calculated by using Navier-Stokes equation of energy, continuity and momentum equation. The numerical result shows that the turbulence kinetic energy is higher on the surface wall impingement at the larger diameter B/D = 0.08. This is due to the high momentum results from the high velocity gradient along the wall surface. The velocity increase with the increasing of the nozzle diameter which attributes to the higher Nusselt number and surface cooling rate.**

***Keywords—***
***Curvature surface***
***Nusselt number***
***Reynolds number***
***Computational Fluid Dynamic (CFD)***
***Turbulence kinetic energy***
***Velocity vector***

**Nomenclature**

B nozzle diameter
D curvature surface diameter
H nozzle to target separation distance
Re Reynolds number

## I. INTRODUCTION

Cooling process of equipment is important to achieve the desired temperature and to reduce runaway process that will cause an explosion. There are many applications for these phenomena such as for heating treatment, cooling of electronic components, heating of optical surfaces for defogging, cooling of turbine blade, cooling of critical machinery structures, and other cooling processes involves in plant and industries [1].

The improved cooling technique is important to avoid the sudden temperature rising in the system. Jet impingement technique whether using the air or liquid is one of many methods of cooling that widely used which to provide the desired thermal environment. This research is using the air as the liquid cooling. A directed liquid that strikes from a jet impingement nozzle to a surface enhanced the coefficient for convective heat transfer. There are many types of jet impingement nozzles such as round nozzle, array round nozzles, slot nozzles, array slot nozzles depending whether jet impingement used is single jet or multi jet impingement. This study is focusing on single round nozzle since single jet impingement is being used. Previous researchers did an experiment mainly towards a flat surface. However, since the existing of evolving equipment and cooling process in the industries such as for cooling the turbine blade, the flat plate is no longer suitable for the studies [2]. Therefore, the study of cooling on curvature surface is important to be compared with the cooling on flat surface.

The nozzle geometry and diameter will affect the jet impingement heat transfer efficiency significantly. Garimella et al. [3] did the research on the effect of jet diameters on flat plate. Various of nozzle geometry used to determine the influence of local convective heat transfer rate for jet impingement cooling on flat plate. They found that the heat transfer rate affected by the different structure and geometry of nozzle jet impingement. Zhou et al. [4] in their study of single round jet impingement on concave surface with variation relative surface curvature stated that the heat transfer rate will increase as the jet diameter increase with the same curved surface. Yang et al. [2] did the experimental study of effect of nozzle configuration on concave surface cooling. They used three different nozzle configuration which are round shaped nozzle, rectangular shaped nozzle and 2D contoured nozzle. From the experiment, they concluded that rectangular shaped nozzle has the highest stagnation point Nusselt number as the turbulence intensity increase sharply when the jet flow passes through the sharp nozzle edge.

Yang et al. [5] did the numerical study on slot jet impingement towards semi-circular surface with various diameters, jet to plate spacing and Reynolds numbers. The result came out and concluded that the average heat transfer rate increases more on concave surface than on the flat surface due to the curvature effect. They also did the experiment of the effect of diameter of jet impingement with range of $0.033 \leq B/D \leq 0.05$ and discovered that the highest Nusselt number occurred at the highest diameter which is $B/D = 0.05$. The heat transfer characteristics of jet impingement cooling towards curvature surface is affected from turbulent jet flow structure ejected from different jet diameter and the concave surface may increase the convective heat transfer rate.

Different nozzle diameter of jet impingement indicates the different of flow characteristics as well as heat transfer characteristics. Choo et al. [6] studied experimentally about the effect of various diameter effect on circular hydraulic jump. For various diameter involved, they concluded that by decreasing the jet diameter with constant Reynold number, the dimensionless hydraulic jump will increase as the impingement power is increased. Lee et al. [7] did an experiment based on jet diameter effect on round turbulent jet on flat surface. In that experiment they came up with the result that described as the potential core increase with the increasing of jet diameter. This means that the velocity will increase thus enhance the heat transfer rate. The surface quenching curves were studied to obtain the recorded temperature data during transient cooling of the hot test surface for various downstream spatial locations [8]. It can be concluded that the heat transfer rate at spatial location is enhanced with the increasing of jet diameter [8].

Thus, the parameter of nozzle diameter is important to develop the correlation with the Nusselt's number. To explain convective heat transfer of the cylindrical curvature surface, the continuity, momentum and energy equations from Navier Stokes are required thus are explained in this study.

This study is to determine the effect of cylindrical surface cooling using nozzle impingement to satisfy the demand and protect the curvature surface equipment form high temperature that will cause an explosion. The heat transfer of cooling process is dependent on many parameters. These parameters are important and correlate with each other to calculate the heat transfer rate of the cooling process on surface impingement. This study is focusing on the nozzle diameter as it will determine the Nusselt number and the Reynold number as well as the heat transfer rate.

## II. METHODOLOGY

### A. Numerical computations

Over the last few years, Computational Fluid Dynamics (CFD) using Reynolds-averaged Navier–Stokes equations, coupled with turbulence modeling has become a standard practical simulation tool for the design and analysis of engineering systems [9]. Thus, the present study is using the CFD simulation to obtain the 2-dimension of the liquid flow of the jet impingement cooling on the cylindrical surface. CFD codes are structured around the numerical algorithms that can tackle fluid flow problems. CFD packages have a user interfaces to input problem parameters and to analyze the result for easy access to their solving power. There are 3 classes involves in this code which is pre-processor, solver and post processor.

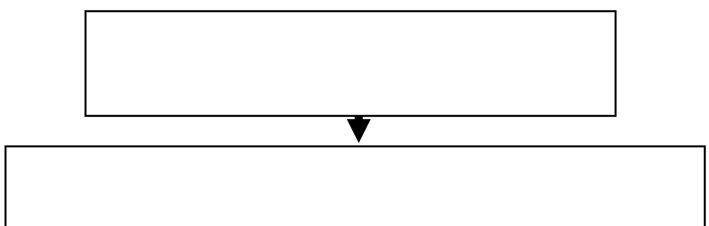


Figure 1 Stepwise CFD procedure

### B. CFD

The overall CFD process have been setup to resolve both upstream and downstream flow around the nozzle as well as to describe the fluid flow. Fluent fluid flow was used in this study to model flow, turbulence, reactions and heat transfer for this study. For further determine how the flow reacts to the impingement surface, steady state and transient state models were applied in the computational domain which predicting both the flow and heat transfer.

Model geometry

The geometry of impingement surface was created using Design Modeler Application and the sketching was done using 2D as suggested by Yang et al. [5]. The diameter of the curvature surface impingement, D is 150 mm. The distance of the nozzle to surface impingement was set to constant which is $H/D = 0.133$ and the diameter of nozzle B/D is set to 0.013, 0.027, 0.033, 0.04, 0.053, 0.067 and 0.08 using the grid. The edges and faces selection were named to inlet, outlet, adiabatic wall, fluid body and surface wall to identify the boundary condition. Figure 2 shows the 2D computational domain of the impingement surface, inlet, outlet and adiabatic wall.

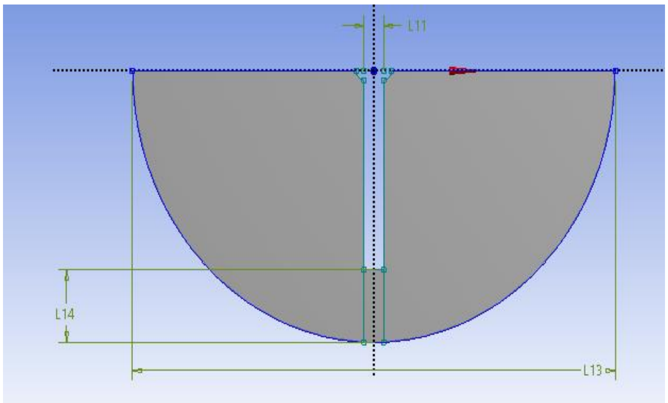



Figure 2 geometry of concave surface in Design Modeler (L11 = 6mm, L13 = 150mm and L14 = 20mm).

Meshing

ANSYS Mesh Module was used in this research to develop the mesh for the computational domain. The element size 5.5 x $10^{-4}$ was applied to all faces of the concave surface to ensure the smoothness of the mesh which will enhance the heat transfer rate. The meshing of the surface can be seen in Figure 3. The surface sizing with smaller mesh size was used to enhance the computational accuracy of the model [10].

The geometry and the mesh were then imported into Fluent in CFD software to solve the governing equation.

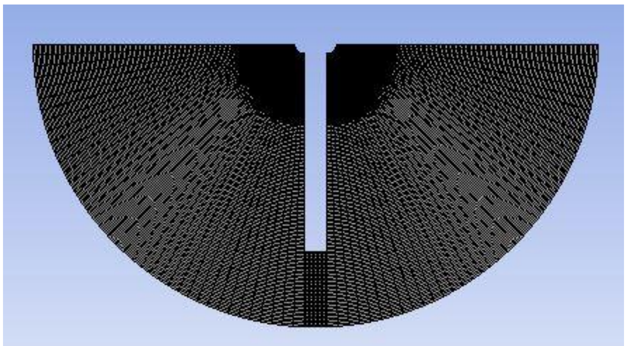


Figure 3 meshing of the surface at element size of 5.5 x $10^{-4}$

Solver

For fluid domain, the boundary condition of inlet flow was set to air. The velocity and temperature of 298.15 K of the inlet flow of the nozzle were specified and the pressure was applied at 1 atm. As for the solid domain, the heat flux of the impingement wall was set at 5663 W/$m^2$ and the wall along the nozzle was specified as adiabatic wall with no slip condition. Second order upwind discretization scheme is used for momentum, turbulence kinetic energy, specific dissipation rate, and the energy. Flow, turbulence, and energy equations have been solved. The standard SIMPLE algorithm is adopted for the pressure-velocity coupling [9].

The simulation firstly done at the steady state for the convergence purpose and further run with transient state. The convergence criteria are specified as 10E-04 residuals for continuity, momentum, turbulence quantities and energy. The iteration was set at 50 maximum which can be changed if convergence is not achieved.

## III. RESULTS AND DISCUSSION

### A. Numerical validation

The numerical validation has been conducted in this study to compare with the experimental [2] and the numerical [5] in the literature. This comparison is to determine the selection of boundary condition in the present study. Various parameter H/B = 0.5, 4, 8 and 12 and Re = 23700 with constant B/D = 0.033 and q″ = 5633 W/$m^2$ were used as shown in the Figure 4. It can be seen that the increase of Nusselt number as H/B increases, is mainly due to increasing axial velocity gradient causes by changing the H/B. This is thought to be due to the fluid acceleration between the nozzle exit and the surface of impingement. This increasing of velocity could be attributed to larger temperature gradient on the curvature surface as well as enhance the heat transfer rate [5].

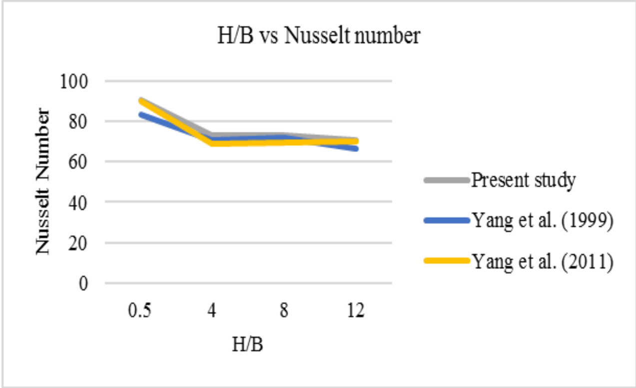
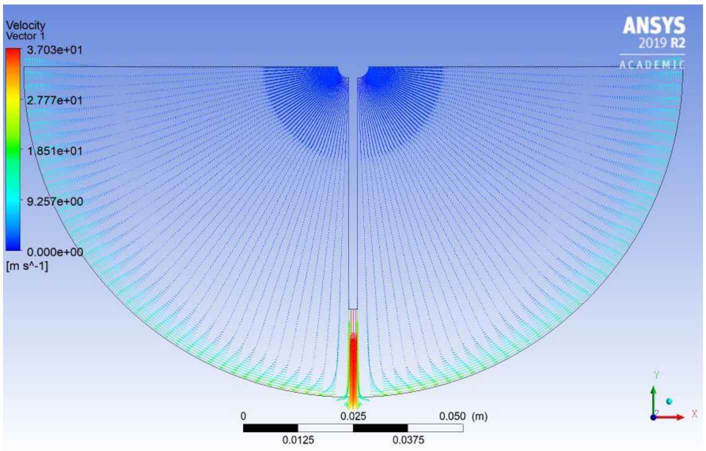


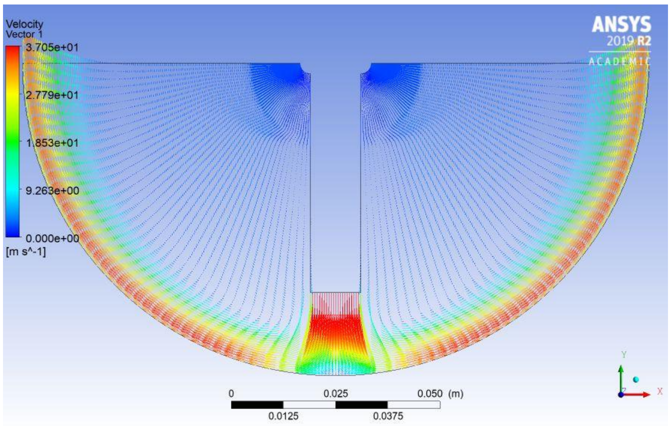

Figure 4 comparison of average Nusselt number distribution with Yang et al.[2] and Yang et al.[5]

### B. Flow characteristics and heat transfer

The jet inlet with velocity and turbulence characteristic are dependent on the upstream flow [11]. Figure 5 shows the computational images of the velocity vectors of B/D = 0.013 and 0.08. It can be seen from the velocity vectors that the shear flow region built up from the edge of the nozzle and extend due to the mixing of jet flow and surrounding air. As the flow approach the impingement surface, the axial velocity is decreases and turns. After turning, the flow expands to the wall jet region where it moves laterally outward parallel to the wall. The wall jet has a shearing layer influenced by the velocity gradient with respect to the stationary fluid at the wall (no-slip condition).[1] The shear strain at the wall reduces gradually but eventually increases again when high velocity fluid mix with the ambient fluid. The velocity at the wall is low due to the viscous friction [12].



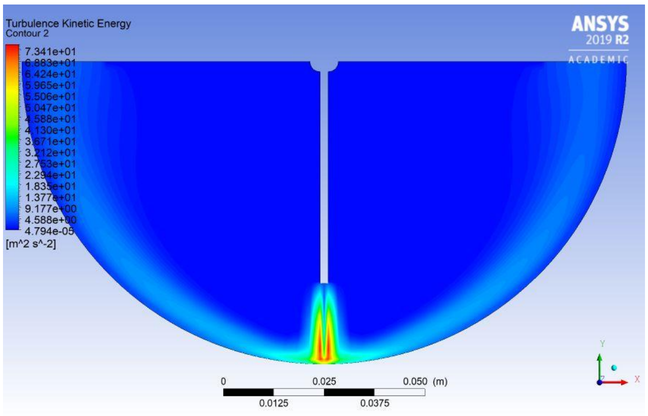

(a)



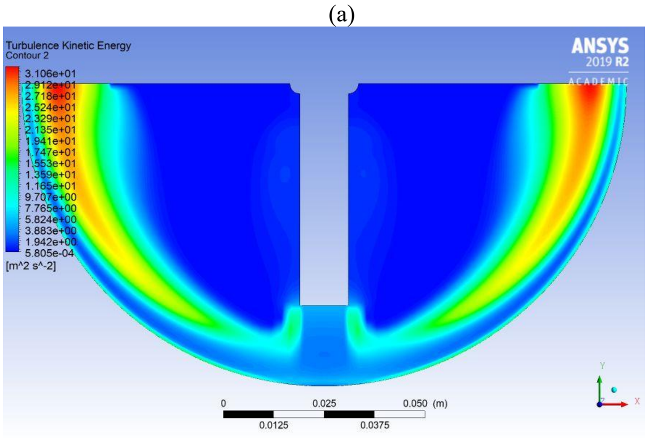

(b)

Figure 5 velocity vector of q″ =5663 W/m2, H/D = 0.13, (a) B/D 0.013; (b) B/D= 0.08

The turbulent kinetic energy of the nozzle exit and the impingement surface can be seen in Figure 6. The turbulence kinetic energy is higher at the shear layer than in the stagnation region as shown in Figure 6. This is due to the existing of the large velocity gradient in the mixing with the shear layer. At B/D = 0.08, as the flow moving upward along the wall jet afar from the stagnation region, the turbulence kinetic energy become the highest due to the high momentum and velocity when jet impinge the surface. Compared to the B/D = 0.08, at B/D 0.013 shows the higher in turbulence kinetic energy at the stagnation region and lower at the wall jet region. This is may cause by the loss of flow characteristics in the process of pre development of the boundary layer when impinge the surface [13].

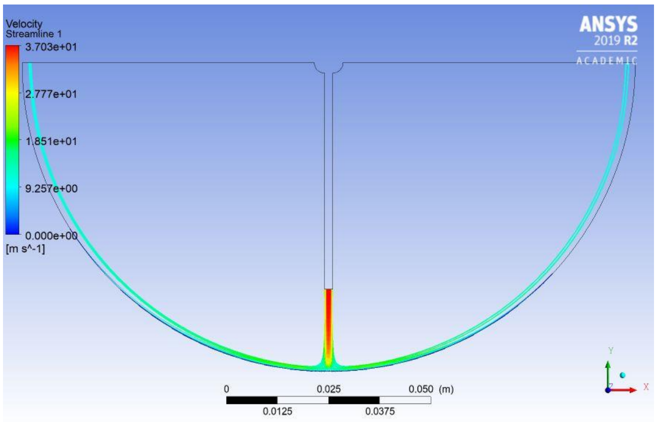



(a)

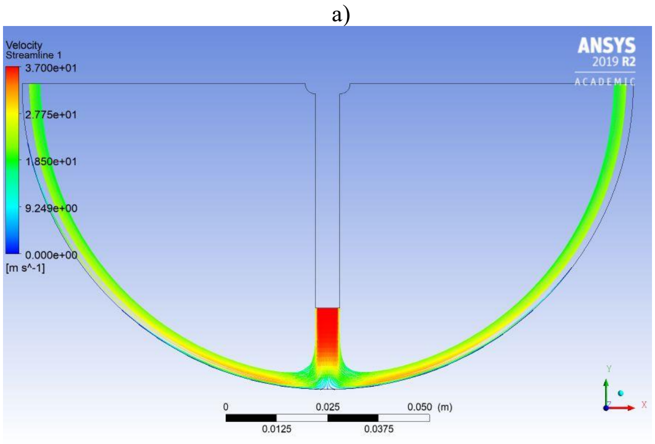



(b)

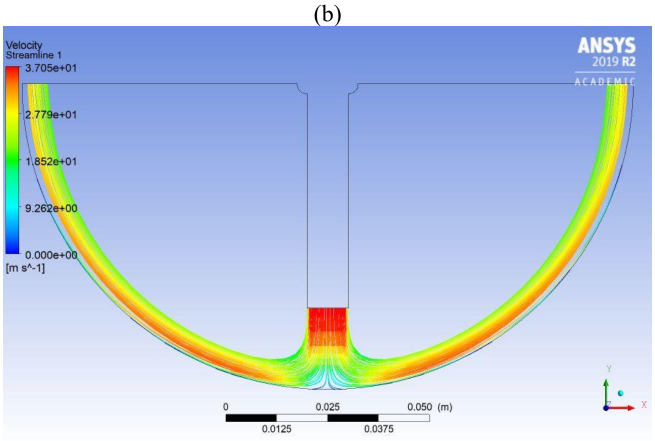
Figure 6 Turbulent kinetic energy of q″= 5663 W/m2, H/D= 0.13, (a) B/D =0.013; (b) B/D =0.08



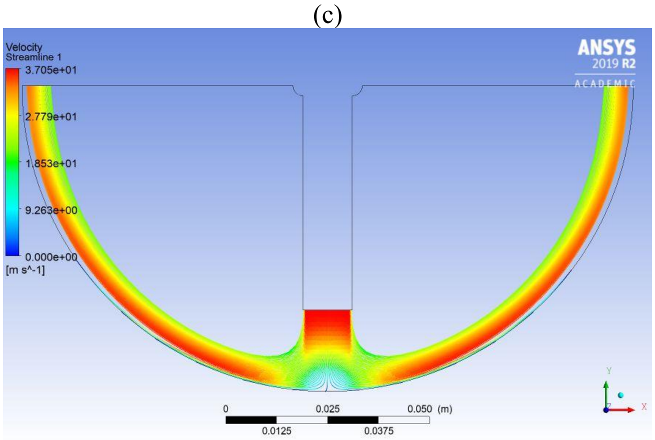

a)




(b)




(c)




(d)

Figure 7 velocity streamline for q″=5663 W/m2, H/D = 0.13: (a) B/D = 0.013; (b) B/D = 0.04; (c) B/D = 0.067; (d) B/D = 0.08

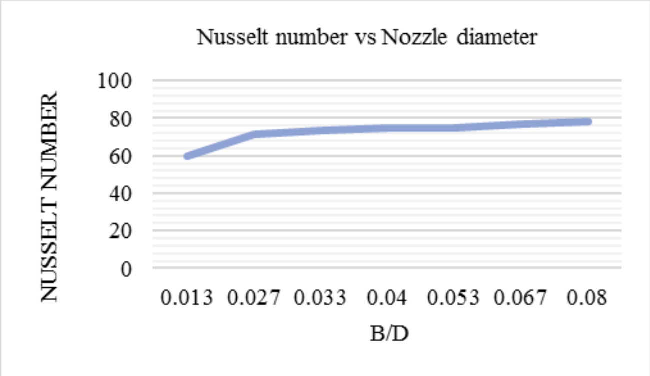



Figure 8 Effect of nozzle diameter on Nusselt number

Figure 7 shows the velocity streamwise of B/D = 0.013, 0.04, 0.067 and 0.08. The large diameter can cause the increasing of the velocity gradient as well as the increase of temperature gradient on the impingement surface. This will enhance the heat transfer rate. The velocity is expected to decrease in magnitude due to the wall jet spread. For the larger nozzle diameter, the velocity streamline is the highest at the potential core region and along the wall region as the broader nozzle inlet gives the large momentum and turbulence intensity which extends over entire surface along the streamwise direction. Surface area also could be attributed to the increasing of Nusselt number. This can be explained by the increasing in diameter will reduce the surface area of ambient fluids which can slow down the heat transfer rate. As shown in Figure 8, the numerical results indicate that the highest Nusselt number is occurs at the highest B/D. The increasing in nozzle diameter will increase the Nusselt number. The results obtained agreed with previous work carried out by Yang et al. [5]. High Nusselt number contributes to the high heat transfer rate.

## IV. Conclusion

Nozzle impingement cooling on the concave surface have been studied numerically. Several diameters were used to study the effect on impingement cylindrical surface. The theoretical model, developed using incompressible turbulent Navier-Stokes equations of motion and energy equation is capable of predicting the flow characteristics and heat transfer of nozzle inlet turbulent on surface impingement. The velocity gradient of the large diameter B/D = 0.08 is higher along the wall surface compared to small diameter B/D = 0.013. This will cause the increase of temperature gradient thus enhancing the cooling transfer rate of inlet fluids and the impingement surface. This study also discovered that the broader nozzle diameter gives the large momentum and turbulence intensity which extends over entire surface along the streamwise direction. Increasing nozzle diameter with constant nozzle to surface spacing will increase the Nusselt number thus enhancing the cooling rate of impingement surface.

## Acknowledgment

Thank you to my supervisor, madam Nadia binti Kamarrudin and Universiti Teknologi Mara for helping me out during this research.

## References

[1] 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.

[2] G. Yang, M. Choi, and J. S. Lee, “An experimental study of slot jet impingement cooling on concave surface: Effects of nozzle configuration and curvature,” *Int. J. Heat Mass Transf.*, vol. 42, no. 12, pp. 2199–2209, 1999.

[3] A. S. Rattner, A. Krishna, S. Garimella, and T. F. Fuller, “International Journal of Heat and Mass Transfer Modeling of a flat plate membrane-distillation system for liquid desiccant regeneration in air-conditioning applications,” *Int. J. Heat Mass Transf.*, vol. 54, no. 15–16, pp. 3650–3660, 2011.

[4] Y. Zhou, G. Lin, X. Bu, L. Bai, and D. Wen, “Experimental study of curvature effects on jet impingement heat transfer on concave surfaces,” *Chinese J. Aeronaut.*, vol. 30, no. 2, pp. 586–594, 2017.

[5] 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.

[6] K. Choo and S. J. Kim, “The influence of nozzle diameter on the circular hydraulic jump of liquid jet impingement,” *Exp. Therm. Fluid Sci.*, vol. 72, pp. 12–17, 2016.

[7] D. H. Lee, J. Song, and M. C. Jo, “The Effects of Nozzle Diameter on Impinging Jet Heat Transfer and Fluid Flow,” *J. Heat Transfer*, vol. 126, no. 4, p. 554, 2004.

[8] C. Agrawal, R. Kumar, a Gupta, and B. Chatterjee, “Effect of Jet Diameter on Surface Quenching at Different Spatial Locations,” vol. 285613, no. 1, pp. 19–22, 2014.

[9] C. Rumsey and T. Beutner, “Introduction: Computational Fluid Dynamics Validation for Synthetic Jets,” *AIAA J.*, vol. 44, no. 2, pp. 193–193, 2006.

[10] H. S, “Jet Diameter Effect on Impingement Jet Cooling on the Leading Edge of a Turbine Blade,” 2015.

[11] N. Zuckerman and N. Lior, “Radial Slot Jet Impingement Flow and Heat Transfer on a Cylindrical Target,” *J. Thermophys. Heat Transf.*, vol. 21, no. 3, pp. 548–561, 2007.

[12] Hayder A, “Numerical study of jet impingement cooling on a smooth curve surface,” no. December, pp. 561–565, 2014.

[13] E. Lai and M. A. Moss, “REVIEW A review of heat transfer data for single circular jet impingement,” vol. 13, no. 2, pp. 106–115, 1992.

[14] A. A. Kendoush, “Theory of stagnation region heat and mass transfer to fluid jets impinging normally on solid surfaces,” vol. 37, pp. 223–228, 1998.