© December 2017 | IJIRT | Volume 4 Issue 7 | ISSN: 2349-6002

Theoretical Investigation for Hot Flat Surface Quenching by Impinging Jet- A Review

Chitranjan Agrawal

Assistant Professor, College of Technology and Engineering, Maharana Pratap University of Agriculture and Technology, Udaipur-313001, Rajasthan, India

Abstract- **The jet impinging surface quenching is predominately used in several industries. Numerous analytical and experimental research works are available for this cooling technique. These investigations have been reviewed several times earlier for steady state cooling condition for air and water. In this manuscript an attempt has been made to review the analytical work published for jet impingement surface quenching with water as coolant. Jet flow structure, jet type and configuration have also been presented in brief for better understanding of the jet impingement quenching process.**

Index Terms- **Heat transfer, Jet Impingement, Jet Exit to Surface Spacing, Potential Core, Stagnation Region, Surface quenching.**

I. INTRODUCTION

Nowadays, rapid cooling of hot surfaces called quenching has become one of the main processes in many industrial applications due to high rates of heat transfer [1]-[3]. The common application areas of impinging jet are cooling of high temperature gas turbines and combustor walls, textile and films drying, annealing and cooling of metals, glass manufacturing, thermal control of high-density electronic equipment, paper industries, manufacturing optical fiber by outside vapor deposition process (OVD) and many more. Jet impingement is one of the methods to achieve quenching with high rate of cooling. There are two ways of study quenching phenomena, one how quenching affects hot surface properties and condition, another how it affects heat transfer characteristics from the surface. To study heat transfer characteristics there are further two areas of investigation, one is to study boiling behavior of fluid and another cooling characteristics. Performance of heat transfer and wetting phenomena of the hot surfaces are major aspects to investigate for study cooling characteristics of hot surfaces. There are many parameters that affect jet impingement quenching phenomena e.g. jet configuration, jet size, fluid flow rate or Reynolds Number, Surface and Fluid temperature, Jet exit to Surface spacing, Surface orientation and roughness etc [1]-[3], [32]-[38], [41]-[45]. The commonly used fluids in jet impingement are air [18]-[20], [25] and water [1]-[3], [21]-[23] but some research have also been reported with fluorocarbons (FC) and Freons [12], [44], [45]. The selection of fluid basically depends on rate of heat removal desired from the hot surfaces.

Field of jet impingement cooling is not in its inception stage, numbers of analytical and experimental work has been published so far that covers wide area of research [1]-[13], [28]-[45]. Numerous attempts have been made to review jet impingement heat transfer by air and water jet impingement heat transfer, especially from the flat hot surfaces [16], [24], [38], [42]. Review available for water jet impingement covers boiling of impinging fluid, however, heat transfer and wetting phenomena of the hot surfaces by jet impingement has not been reviewed extensively [42]. A review on wetting phenomena by jet impingement was reported by Monde et al. [27] based on experimental findings of their coworkers. In this paper an attempt has been made to summarize the analytical and experimental facts related to the circular liquid jet impingement heat transfer over the hot flat plate.

II. JET IMPINGEMENT

The Jet impingement is broadly of three types circular (laminar), planer (Curtain) and spray as shown in Figure 1 [3], [42]. Further these may be single jet or multiple jets also depend on rate of cooling desired. All these three jets have their own advantages and disadvantages. The circular jet is normally preferred because it can penetrate the vapour film on the cooled surface and increase the coolant residence time. In the water curtain system, the surface is cooled by a planer or slot jet, which can cover entire width of the surface. One of the advantages of this system is the improvement of uniform surface cooling in the transverse direction. In a water spray system, the coolant impinges from a row of specially designed nozzles on to the surface.

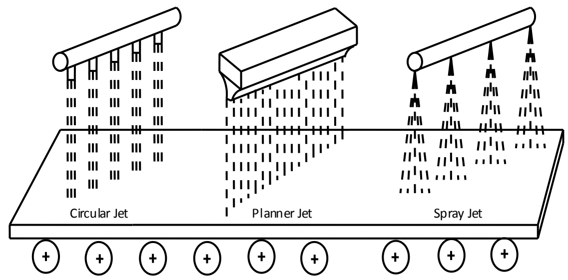

Figure1 Jet Categories [16]

Jet impingement can further configure into different categories these are free surface jets, plunging jets, submerged jets and confined jets, as shown in Figure 2. Impinging jets can also be classified by their impinging direction i.e. whether the jet is oriented normally or obliquely with respect to the impingement surface and on the basis of target surface i.e. flat, convex, or concave. The jet is also called sub cooled jet if the temperature of jet fluid is less than fluid saturation temperature. A free surface jet is the case where liquid jet exposed to a gaseous environment before impinging on an unconstrained flat surface. Unconstrained means liquid can flow off from the edges and does not pool on the surface. On a moving surface there can either be free surface or plunging jets depending on the configuration. When the moving surface is cooled by array of water jets, the first row of jets are free surface jets but further downstream a layer of water is formed on top of the surface and the jets act as plunging jet. As the thickness of the water layer increases the effect of impingement decreases. For maximum heat transfer, all the jets should be free surface types. Submerged jet conditions are those in which jet fluid is same as surrounding fluid, gas and air jet also falls in this category. It was reported that the heat transfer characteristics of submerged liquid jets are similar with gas or air jet impingement [29], however, heat transfer depends on Prandtl number of fluid therefore it should be taken into account to determine heat transfer performance.

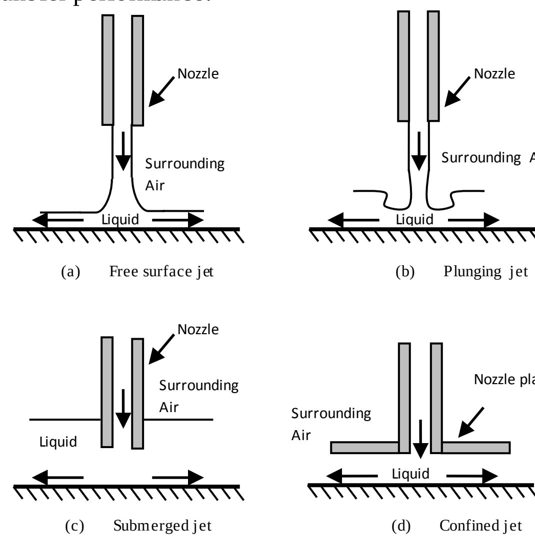

Figure 2 Jet Configurations

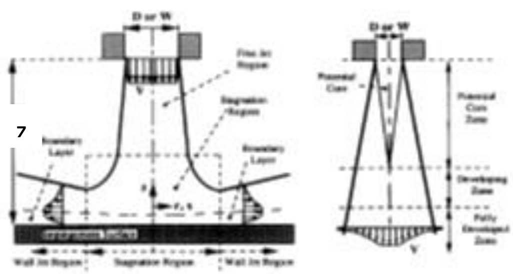

Figure 3 Jet Flow Structure [38]

The flow structure of jet (Figure 3) remains same irrespective of impinging jet configuration, type of fluid and target surface. The flow structure is mainly divided into three regions i.e. the free jet region, the impingement (stagnation) region, and the wall jet region as discussed several times in the literature [16], [21]-[24], [38]. The impingement region is characterized by large pressure gradient, shear stress and heat-transfer rates [37]-[40]. However, many researcher [21]-[23] divides stagnation and wall jet region over the surface into 4-5 different regions for their analysis based on laminar thermal and hydraulic boundary layer. These regions are characterized by different velocity profile due to interaction of mass, momentum and energy within the fluid and with surrounding fluid both in axial and radial direction. These regions are further responsible for different boiling and heat transfer performance by hot surfaces.

III. REVIEW OF ANALYTICAL WORK

A number of analytical investigations are available on circular, planer or slot and annular jet over stationary and moving surface both for the steady and the unsteady state heat transfer analysis. These analyses were based on different laminar and turbulent flow model over the hot surface to categorized different region of flow and heat transfer performances.

Watson [41] studied the fluid mechanics of circular liquid jet using boundary layer theory for laminar and turbulent flow and used viscous similarity and momentum integral solution for the analysis. He divided the flow into stagnation and boundary layer region where surface velocity equals to the jet speed and a region of viscous similarity with decreasing surface velocity. However solution reported was only for boundary layer region. He reported that the film flow terminated by hydraulic jump at a location controlled by downstream condition. Stevens and Webb [36] experimentally determined the location of this hydraulic jump. The film thickness initially decreases than increases downstream over the hot surface due to viscous wall effect which retarded the flow. This was in line with the Steven’s explanation for knee formation in the local heat transfer.

Fujimoto et al. [9] numerically simulate the steady state flow structure in liquid film and the local heat transfer assuming non turbulent free surface flow and constant heat flux between liquid and hot surface. They were able to predict various parameters like liquid film thickness, free surface temperature and pressure profile, velocity and local Nusselt Number over the hot surface in radial direction well within the range of experimental values observed by Stevens and Webb [36].

Liu et al. [22] work was also for constant heat flux condition but used integral method with considering initial laminar and downstream turbulent transition flow in the similarity region. They analyzed the effect of Prandtl Number on heat transfer and predicted critical Prandtl number above which thermal boundary layer does not reaches liquid film free surfaces. They stated that for the lower Prandtl Number, under uniform heat flux condition, the local heat transfer coefficient is 25% higher than uniform wall temperature in the boundary layer region and remain insignificant for the stagnation region. They found integral equations gives about 10 percent lower heat transfer values than differential equations. Liquid flow was divided into four different region based on thermal and hydraulic boundary layer. Separate analytical correlations were proposed for these different region both for laminar and transition region. With the turbulent transition flow analysis they were able to explain secondary maxima at higher Reynolds Number and low z/d as obtained in experimental result by many researchers.

However Ma et al. [23] analytical work for local heat transfer at five different region of flow over the surface both for Pr>1 and Pr<1 was under arbitrary heat flux condition also based on integral method. They proposed analytical correlations for local heat transfer for these different regions. Webb et al [42] theoretical study was for hot surface which temperature was below the boiling of impinging liquid and Chowdhary [8] analysis was for isothermal hot surfaces but for the wall jet region only.

Experimentally it was found that for a fully developed circular jet prior to impingement with spacing (z/d > 8-10), the local surface heat transfer decreases monotonically from stagnation point. However, if the jet is not fully developed, the heat flux exhibits two well defined secondary peaks one located within the impingement region at r = 0.5 d, (inner peak) and the other at about r = 1.6 - 2d, (outer peak). Outer peak is due to the transition of wall boundary layer flow from laminar to turbulent and the inner peak is due to local thinning of the wall boundary layer. This phenomenon was observed experimentally by Gardon et al. for air [10], Stevens and Webb [36], Garimella et al [12] for water and theoretically by Watson et al. [41]. However, such a thinning of wall boundary layer has not been observed by Hrycak er a1. [14], [15] in their experiments. Pamadi et al. [31] theoretically established that the inner peak occurs due to the non-uniform, mixing-induced turbulence in the developing jet. Their analytical result is based on a solution of incompressible flow parabolic equations and a semi-empirical turbulence model for Z/d = 2 and 4 and is consistent with the predictions of Kestin et a1. [17]. Badra et al. [4] using commercial software found that Inner peak moves out ward with increase in Reynolds number and decrease in (z/d).

There are few analytical research is available that can be applied to transient heat transfer or complicated processes. Rao et al. [33] numerical study was for impingement region using Dufort Frankel implicit scheme and obtained Static pressure distribution, shear stress and Nusselt number distribution over the flat surface for Pr=1, Reynolds number=450 and various z/d.

Liu et al. [22] used two phase flow boundary layer equations to calculate the thickness of vapor layer and obtained heat transfer coefficient for the stagnation zone and further modified it based on their experimental values. Hatta et al. [13] developed a numerical model to calculate the relation between temperature and time at a given point of hot plate during cooling. A semi empirical relation was proposed for the heat transfer coefficient and water cooling zone radius, based on the numerical model and the experimental result.

Ouattara et al. [30] gave an analytical model in the cylindrical coordinate system to simulate slow transient 2D conduction during jet impingement. In this model they have assumed that the source term is a piecewise constant space function of radius r and cooling flux is estimated by inverse heat conduction procedure. This model is used to estimate the time dependent cooling heat flux by impact of the water jet as well as the surface temperature.

Monde et al. [26] analytical analysis to obtain surface temperature and heat flux was based on two-dimensional inverse heat conduction equations which were solved by the Laplace transform technique. The inverse solutions are obtained under two simple boundary conditions in a finite rectangular body, with one and two unknowns. The method first approximates the temperature changes measured in the body with a half polynomial power series of time and Fourier series of Eigen function. The expressions for the surface temperature and heat flux are explicitly obtained in the form of power series of time and Fourier series.

Few analyses were able to validate experimental result of heat transfer performance well within range using available commercial software like FLUENT with different turbulent models. Behnia et al. [3] used an elliptic relaxation turbulence model (v2 - f model) to simulate the flow and heat transfer in circular confined and unconfined impinging jet configurations. This model shows good agreement with experimental results compare to low Reynolds number turbulence model (k-ε), which over predict the stagnation Nusselt Number due to high turbulence. A theoretical analysis with different nozzle-to-target distance, Reynolds number, jet confinement and jet-exit profiles shows that confinement leads to a decrease in the average heat transfer rate. However, the stagnation heat transfer coefficient is remains unchanged same has also been experimentally proved by Obot et al. [29], and Garimella and Nenaydykh [11] for water. The effect of confinement is only significant in very low nozzle-to-plate distances (z/d < 0.25) and away from stagnation point (r/d>0.5). However, the Nusselt number at 6 jet diameters away from the stagnation point is nearly independent of nozzle-to-plate spacing that was inline with Jambunathan et al. [16] findings but for air jet. They were also analyzed that the nozzle-exit jet profiles have a strong influence on the heat transfer rates, up to a radial distance of 2d, and at small nozzle-to-plate spacing, for which the free jet has no time to develop.

Morris et al. [28] used a modified (k- ε) model based on renormalization group theory to solve the flow fields in the orifice and the confinement region of a normally impinging, axisymmetric, confined, and submerged liquid jet. They characterized the separation region at the entrance of the orifice and pressure drop across the orifice which was in line with experiments finding of Garimella and Rice [12]. The computed flow patterns in the confinement region of the impinging jet were also same as flow visualizations during the experiments; however, a secondary counter-rotating recirculation zone observed in experiments was not predicted by the models. This counter rotating recirculation was assumed to be responsible for secondary maxima in local heat transfer by Garimella and Rice [12].

Badra et al. [4] were able to validate results for single jet with the experimental data of Stevens and Webb [35] using Fluent and Gambit. They tried six turbulent models and compared average Nusselt number with available correlations. The second peak in Nusselt number at low jet-to-plate distances was well predicted by only the SST k−ω model. Further on the basis of performance of this model for single jet they extended their analysis for the multiple impinging jets for the range of z/d, and Reynolds Number.

Lienhard et. al. [21] modeled splattering of jet and its effect on heat transfer using momentum integral procedure which gives relatively clear and general description of varying characteristics of flow. They characterized the flow over the hot surface by four regions. This model relates the initial turbulence of the jet to the initial surface disturbances in the jet. These disturbances subsequently grow by capillary instability and reaches to the liquid sheet, the initial disturbances are sharply amplified that causes Splattering. Splattering has a strong effect on heat transfer, especially immediately after the breakaway radius. Other researchers were able to explained generation of splattering and its effect on heat transfer performance with the experimental results [20] [43]. Lienhard et al. [21] also mentioned that splattering reduces cooling by jet impingement on hot surfaces. With the jet splattering much of the incoming liquid become airborne within the few jet diameters of the point of jet impact.

IV. CONCLUSIONS

Based on the review of available literature it is found that heat transfer performance under submerged condition are very similar to the heat transfer by air jet. Under submerged condition heat transfer is more than the under free surface jet condition, however confinement reduces heat transfer substantially. Heat transfer is highest at stagnation region and reduces for downstream locations. A secondary maxima in local heat transfer is observed under all condition for high Reynolds number and low z/d spacing only. The effect of z/d spacing on heat transfer performance is significant under submerged condition as compared to free surface jet condition. Maximum heat transfer occurs when spacing is equal to the jet potential core length. Thereafter decreases monotonically with rise in spacing. However, if the jet is not fully developed, the heat flux exhibits two well defined secondary peaks one located within the impingement region and the other at r = 1.6 - 2d, (outer peak). The inner peak is due to local thinning of the wall boundary layer and outer peak is due to the transition of wall boundary layer flow from laminar to turbulent and the. For lower Prandtl Number, in the boundary layer region, under uniform heat flux condition, the local heat transfer coefficient is 25% higher than uniform wall temperature, this difference is insignificant for the stagnation region. Moreover, within stagnation region heat transfer remains unaffected by the change in operating parameters like nozzle exit to surface spacing, jet flow rate, jet fluid temperature and jet type. The splattering reduces jet impingement heat transfer performance limited to breakaway radial distance. This decrease is further extended by reducing the spacing (z/d) and Reynolds number.

REFERENCES

[1] Agarwal C, Kumar R, Gupta A, Chatterjee B, 2014, Effect of Nozzle Geometry on the Rewetting of Hot Surface during Jet Impingement cooling, Experimental Heat Transfer, Vol. 27, 256 – 275.

[2] Agarwal C, Kumar R, Gupta A, Chatterjee B, 2016, Determination of Rewetting Velocity during Jet Impingement Cooling of Hot Vertical Rod, Journal of Thermal Analysis and Calorimetry, Vol. 123, 861-871.

[3] Agrawal C, Gotherwal D, Singh C, Singh Charan, 2017, Effect of Surface Thickness on the Wetting Front Velocity during Jet Impingement Surface Cooling, Heat and Mass Transfer, Vol. 53, 733-741.

[4] Badra J., Masri A. R.,. Freeman M, and. Fitch M, 2007, Enhanced Heat Transfer from Arrays of Jets Impinging on Plates, 16th Australasian Fluid Mechanics Conference, Crown Plaza, Gold Coast, Australia.

[5] Behnia M, Parneix S., Shabany Y, Durbin P.A., 1999, Numerical study of turbulent heat transfer in confined and unconfined impinging jets, International Journal of Heat and Fluid Flow, Vol. 20, 1-9.

[6] Besserman D. L., Incropera F. P. and Ramadhyani S., 1992, Heat Transfer From a Square Source to an Impinging Liquid Jet Confined by an Annular Wall, ASME Journal of Heat Transfer, Vol. 114, 284 – 287.

[7] Chattopadhyay H., 2004, Numerical investigations of heat transfer from impinging annular jet, International Journal of Heat and Mass Transfer, Vol. 47, 3197–3201.

[8] Chaudhury, Z. H., 1964, Heat Transfer in a Radial Liquid Jet, J. Fluid Mech., Vol. 20, 501-511.

[9] Fujimoto H., Hatta N. and Viskanta R., 1999, Numerical simulation of convective heat transfer to a radial free surface jet impinging on a hot solid, Heat and Mass Transfer, Vol. 35, 266-272.

[10] Gardon, R., and Akfirat, J. C., 1965, The Role of Turbulence in Determining the Heat-Transfer Characteristics of Impinging Jets, Int. J. Heat Mass Transfer, Vol 8, 1261-1272.

[11] Garimella S. V. and Nenaydykh B., 1996, Nozzle Geometry Effects in Liquid Jet Impingement Heat Transfer, Int. J. Heat and Mass Transfer, Vol. 39, 2915-2923.

[12] Garimella, S.V., and Rice, R.A., 1995, Confined and Submerged Liquid Jet Impingement Heat Transfer, ASME Journal of Heat Transfer, Vol. 117, 871-877.

[13] Hatta, N., Kokado, J., and Hanasaki, K., 1983, Numerical Analysis of Cooling Characteristics for Water Bar, Transaction ISIJ, Vol 23,5 55-564.

[14] Hrycak P. 1981, Heat transfer from a row of jets impinging on concave semi-cylindrical surfaces, International Journal of Heat Mass Transfer, Vol 28, 175–181.

[15] Hrycak P., 1980, Heat transfer from a row of impinging jets to concave cylindrical surfaces, International Journal Heat mass transfer, Vol 24, 407-419

[16] Jambunathan, K., Lai, E., Moss M.A., and Button B.L, 1992, A Review of Heat Transfer Data for Singular Jet Impingement, Int. J. Heat Fluid Flow, Vol. 13, 106–115.

[17] Kestin J., Maeder P. F., and Wang H. E., 1961, Influence of turbulence on the transfer of heat from plates with and without a pressure gradient, Int. J. Heat Mass Transfer, Vol. 3, 133-154.

[18] Lee J., and Lee S. J., 2000, The effect of nozzle configuration on stagnation region heat transfer enhancement of axisymmetric jet impingement, International Journal of Heat and Mass Transfer, Vol. 43, 3497–3509.

[19] Lee, D. H., Chung, Y. S., and Kim, D. S., 1997, Turbulent Flow and Heat Transfer Measurements on A Curved Surface With A Fully Developed Round Impinging Jet, Int. J. of Heat and Fluid Flow, Vol. 18, 160-169.

[20] Lee, D. H., Chung, Y. S., and Kim, M. G., 1999, Turbulent Heat Transfer From A Convex Hemispherical Surface to A Round Impinging Jet, Int. J. of Heat and Mass Transfer, Vol. 42, 1147-1156

[21] Lienhard V.J. H., Liu H., Xabour L. A., 1992, Splattering and Heat Transfer During Impingement of a Turbulent Liquid Jet", Vol. 114, 362- 372.

[22] Liu X., Lienhard V J. H., Lombara J. S., 1991, Convective Heat Transfer by Impingement of Circular Liquid jets, ASME Journal of Heat Transfer, Vol. 113, 571- 582.

[23] Ma C. F., Zhao Y. H., Masuoka T. and Gomi T., 1996, Analytical Study On Impingement Heat Transfer With Single Phase Free Surface Circular Liquid Jets, J. of Thermal Science, Vol. 5, 271-277.

[24] Martin, H., 1977, Heat and Mass Transfer Between Impinging Gas Jets and Solid Surfaces, Advances in Heat Transfer, Vol. 13, 1-60.

[25] Monde M., Arima H., Liu W., Mitutake Y., Hammad J.A., 2003, An analytical solution for two-dimensional inverse heat conduction problems using Laplace transform, Int. J. of Heat and Mass Transfer, Vol. 46, 2135–2148.

[26] Mohanty, A. K., and Tawfeek, A. A., 1993, Heat Transfer Due to A Round Jet Impinging Normal to A Flat Surface, Int. J. Heat Mass transfer, Vol. 36, 1639-1647.

[27] Monde M., 2008, Heat transfer characteristics during quenching of high temperature solids, Journal of Thermal Science and Technology, Vol. 3, 292- 308.

[28] Morris G. K., and Garimella S. V., 1998, Orifice and Impingement Flow Fields in Confined Jet Impingement, ASME Journal of Heat Transfer, Vol. 120, 68- 72.

[29] Obot N. T., Douglas, W. J. M., and Mujumdar, A. S., 1982, Effect of Semi- Confinement on Impingement Heat Transfer, Procs. 7th International Heat Transfer Conf.,

[30] Ouattara A., Maillet D., Gradeck M., Remy B., and Lebouché M., 2008, Estimation Of Wall Conditions For Transient Water Jet Cooling Of Hot Metals Using An Inverse Heat Conduction Based On An Analycal Model, 5th European Thermal-Sciences Conference, Netherlands.

[31] Pamadi B. N., and Belov I. A., 1980, A Note On The Heat Transfer Characteristics Of Circular Impinging Jet", Int. J. Heat Mass Transfer. Vol. 23, 783 -787.

[32] Pan Y, J. Stevens J and Webb B. W., 1992, Effect of Nozzle Configuration on Transport in the Stagnation Zone of Axisymmetric, Impinging Free- Surface Liquid Jets: Part 2—Local Heat Transfer, ASME Journal of Heat Transfer, Vol. 114, 880- 886.

[33] Rao S. K., 1975, Numerical Study of Normal Impingement of A Plane Jet on A Flat Surface, Vol. 8, 29-43.

[34] Royne, C. J. Dey, 2006, Effect of nozzle geometry on pressure drop and heat transfer in submerged jet arrays, International Journal of Heat and Mass Transfer, Vol. 49, 800–804.

[35] Stevens J, Pan Y, and Webb B. W., 1992, Effect of Nozzle Configuration on Transport in the Stagnation Zone of Axisymmetric, Impinging Free- Surface Liquid Jets: Part 1— Turbulent Flow Structure, ASME Journal of Heat Transfer, Vol. 114, 874- 879.

[36] Stevens, J., and Webb, B. W., 1991, Local Heat Transfer Coefficients Under an Axisymmetric, Single-Phase Liquid Jet, ASME Journal of Heat Transfer, Vol. 113,71-78.

[37] Tawfek, A. A., 1996, Heat transfer and pressure distributions of an impinging jet on a flat surface, Heat and Mass Transfer, Vol. 32, 49-54.

[38] Viskanta R., 1993, Heat transfer to impinging iso thermal gas and flame jets, Exp. Thermal Fluid Sci., Vol 6, 111–134.

[39] Wang X. S., Z. Dagan and L. M. Jiji, 1989, Heat transfer between a circular free impinging jet and a solid surface with non-uniform wall temperature or wall heat flux-l. Solution for the stagnation region, Int. J. Heal Mass Transfer. Vol. 32, 351-360.

[40] Wang XS; Dagan Z; Jiji LM, 1989, Heat transfer between a circular free impinging jet and solid surface with nonuniform wall temperature or wall heat flux-2. Solution for the boundary layer region, Int. J. Heat Mass Transfer, Vol. 32, 361-371

[41] Watson, E. J., 1964, The Radial Spread of a Liquid Over a Horizontal Plane, J. Fluid Mech., Vol. 20, pp. 481-499.

[42] Webb B.W., and Ma C. F., 1995, Single-phase liquid jet impingement heat transfer. Advances in Heat Transfer, Vol. 26, 105-217.

[43] Whelan, B.P., and Robinson, A.J., 2009, Nozzle geometry effects in liquid jet array impingement, Applied Thermal Engineering, Vol. 29, 2211-2221.

[44] Womac, J., Ramadhyani, S., and Incropera, F.P., 1993, Correlating Equations for Impingement Cooling of Small Heat Sources With Single Circular Liquid Jets, ASME J. of heat transfer, Vol. 115, 106-115.

[45] Womac D. J., Incropera F.P., and S. Ramadhyani, 1994, Correlating equations for impingement cooling of small heat sources with multiple circular liquid jets, Journal of Heat Transfer, Vol. 116, 482–486.