IJNRD.ORG ISSN : 2456-4184
INTERNATIONAL JOURNAL OF NOVEL RESEARCH AND DEVELOPMENT (IJNRD) | IJNRD.ORG
An International Open Access, Peer-reviewed, Refereed Journal

COOLING OF HOT CYLINDER USING JET OF AIR: A REVIEW

[1]ANKUR SACHDEVA, [2]TARUN KUMAR YADAV, [3]VAIBHAV SINGH, [4]SARTHAK CHAUDHARY, [5]RAHUL PAL

[1]Assistant Professor, [2]Student, [3]Student, [4]Student, [5]Student
[1]Mechanical Engineering,
[1]KIET Group of Institutions, Ghaziabad, INDIA

Abstract: For obtaining high rates of heat transfer, when temperature of the impinging fluid is different than the impingement surface, impingement cooling has many of its applications. Impingement cooling includes a jet of high velocity which strikes against an object to be cooled or heated. This review paper sums up the influence of different nozzles of an impingement cooling on the rectangular and cylindrical surfaces, the heat transfer characteristics of a fluid jet impinging on a target surface, and different types of turbulence models. In previous times some experiments were carried out to cool a smooth cylinder, electrically heated, with a rectangular jet of air, at different Reynolds numbers in turbulent flow, where the Reynolds number can be defined with the help of cylinder diameter. This review is done to explain impingement on different types of round or cylindrical surfaces on the basis of different reviews published in previous times, different turbulence models used as there are very few reviews available on curved surfaces compared to flat surfaces and here no personal experimental data is used.

Index Terms **- Jet Impingement, Heat Transfer, Cylinder, Jet height, Turbulence**

I. INTRODUCTION

Review includes many investigations of impingement cooling jets. A variety of geometrical or physical parameters of cooling systems has been studied. Some numerical simulations of the impingement cooling, different CFD turbulence models are presented in the review paper. There have been a number of different studies made to analyse the different dimensionless y/D parameters or different Reynolds number values. Many correlations were introduced to calculate the heat transfer coefficient in the stagnation region, applied for the adequate Reynolds number and nozzle height range.

Jet impingement heat transfer has a wide range of applications in thermal engineering systems like electronic cooling, textiles and turbine blade cooling. If a fluid exits from a nozzle, pipe, rectangular slot, or other opening into a large free space within which there is a target surface whose thermal state is to be managed, a jet is created. By using impinging jets, photovoltaic panel thermal management can be achieved in solar power applications. Jets are used in the heat transfer apparatuses because they can provide an efficient means of cooling. Jet impingement heat transfer with a cylindrical target surface is widely used in heating as well as cooling of continuous casting of circular metal billets, surface cooling of pipes carrying hot air in aircraft cooling systems and cooling of food articles in the food industry. Heat transfer characteristics are dependent on the velocity and the turbulence of the impinging jet.[2]

II. Modelling Flow and Heat Transfer in Air Jet Impingement

Heat transfer and flow of fluid can be shown or calculated by three basic equations called the Navier-Stokes equations. Navier-Stokes equations describe conservation of mass, momentum and energy of a fluid flow. In turbulent flow, velocity magnitude fluctuates with time. This fluctuation is known as turbulence. Velocity in turbulent flow can be divided into average velocity (U) and the turbulent component (u') [3].

$$u_i = U_i + u'$$

When turbulence is also added to the velocity term, Navier Stokes equations written as:

Continuity $$\frac{\partial \bar{U}_i}{\partial x_j} = 0$$

Momentum $$\frac{\partial \bar{U}_i}{\partial t} + \frac{\partial \overline{U_i U_j}}{\partial x} = -\frac{1}{p}\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[v_f \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) - \overline{u_i' u'}_j \right]$$

Energy $$\bar{\rho}_f c_v \frac{\partial \bar{T}}{\partial t} + \rho_f \bar{U}_j c_p \frac{\partial \bar{T}}{\partial x_j} = -\frac{\partial}{\partial x_j}\left[k_f \frac{\partial \bar{T}}{\partial x_i} + \bar{\rho}_f c_p \bar{u}' \bar{T}' \right]$$

Where,

$$\frac{\partial}{\partial x_i} = \frac{\partial}{\partial x}, \frac{\partial}{\partial y} \qquad i.e. \qquad x_1 = x, x_2 = y$$

$$\frac{\partial}{\partial x_j} = \frac{\partial}{\partial x} + \frac{\partial}{\partial y} \quad \text{and} \quad U_j \frac{\partial}{\partial x_j} = U_x \frac{\partial}{\partial x} + U_y \frac{\partial}{\partial y}$$

III. BOUNDARY CONDITIONS:

Performing or doing simulation using Computational Fluid Dynamics (CFD) that solves equations of continuity, momentum and energy using the Reynolds-Averaged Navier-Stokes approach requires well-defined boundary conditions of the grid, in order to simulate the flow and heat transfer. Boundary conditions are shown in the figure given below:

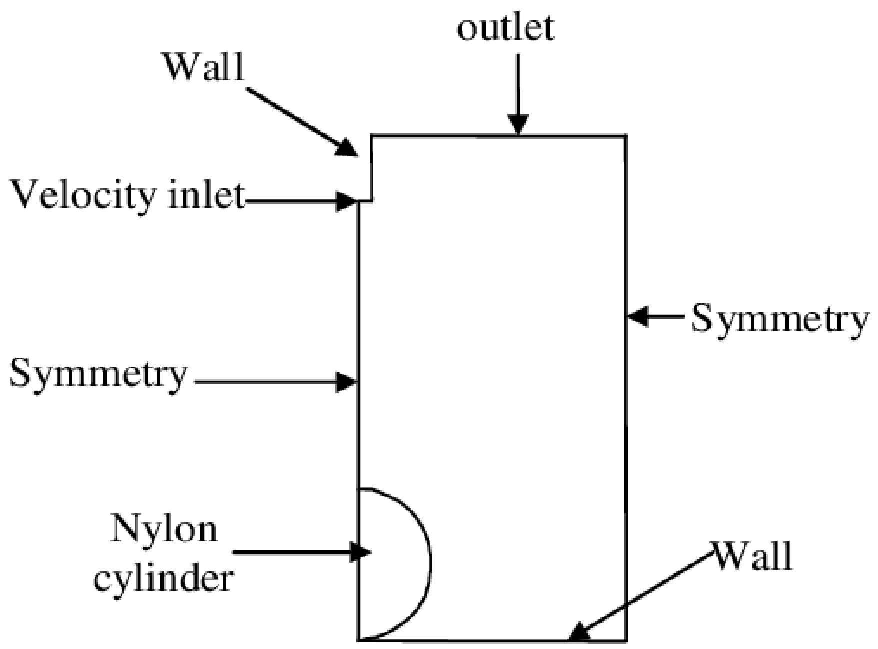

FIG.1: BOUNDARY CONDITIONS USED IN FLUENT FLOW

To simplify the analysis, the steady-state fluid flow is considered and it was assumed that the fluid physical properties are constant and the effect of the gravity and radiation is neglected.

IV. EFFECT OF REYNOLDS NUMBER (RE):

For slot jet impingement, Reynolds number can be defined based on the hydraulic diameter of the jet exit (Yang et al., 1999)

$$Re = \frac{U2B}{\upsilon}$$

Where U is velocity magnitude of the fluid (m/s), B is the width of the slot nozzle (m), and υ is the kinematic viscosity of the fluid (m^2/s).

For a very low Reynolds number, heat transfer characteristics of the impinging jet can be described by velocity alone but at higher Reynolds number, turbulence generated by the jet plays an important role in heat transfer rate and distribution (Gardon and Akfirat, 1965). Heat transfer rate increases with increasing Reynolds number, but the increase is asymptotic. The change in heat transfer coefficient is mainly at the stagnation point, and the effect remains for a very short distance from the stagnation point for impingement on a flat plate and it was observed by Sarkar (2004). The effect of curvature diminishes, which means there is no difference in heat transfer profile around the objects of different curvatures for Re = 6,000.

V. METHODS AND TURBULENCE MODELS:

5.1: Simulations:

Steady state simulations in two dimensions of the heat transfer from a slot air jet impinging on a solid cylindrical product in a semi-confined domain can be made using the CFD software ANSYS. The Nusselt number describes the dimensionless heat transfer; it is a function of the heat transfer coefficient, the thermal conductivity, and a characteristic length, which in this study is the jet width (d). The impinging jet can be assumed to be fully turbulent when exiting the slot pipe. The Reynolds number is based on the average jet velocity and the width of the jet.

5.2: Governing Equations:

The equations which describe the fluid flow and the heat transfer from the jet impinging to the solid cylinder are the transport equations of momentum and energy, which are developed from conservation laws of physics.

$$\frac{\partial U_j}{\partial x_j} = 0$$

$$\rho \frac{\partial U_i}{\partial t} + \rho \frac{\partial (U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} + \tau_{ij}^{\mathrm{turb}}\right);$$

$$\tau_{ij} = \mu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right); \quad \tau_{ij}^{\mathrm{turb}} = -\rho\overline{u_i' u_j'}$$

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p \frac{\partial (U_j T)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(q_j + q_j^{\mathrm{turb}}\right);$$

$$q_j = \frac{\mu c_p}{Pr}\frac{\partial T}{\partial x_j}; \quad q_j^{\mathrm{turb}} = -\rho c_p \overline{u_j' T'}$$

It would be impossible to derive these equations analytically due to the non-linear and stochastic nature of turbulence.

5.3: SST Turbulence Model:

Menter (1994) provided a model of shear stress transport (SST) which combines the advantages of the k–x and the k–e model. The pros and cons of the k–x and k–e models are well known (Menter, 1994). The SST model comes between the k–x model near the surface and the k–e model at the boundary layer edge and outside the boundary layer. Eddy viscosity is modified to rely for the transport of the principal turbulent shear stress in the SST model. The formulation of the turbulent shear stress in two-equation models leads to an over prediction in adverse pressure gradient flows (Menter, 1994). The accuracy of the wall shear stress is determined by the near-wall formation, the wall heat transfer predictions and has an influence on the development of the boundary layer, including the point of separation. According to a literature published by Baughn & Saniei, 1991; Knaabel, McKillop, & Baughn, 1982 there was a comparison done between simulations of a cylinder in cross flow using the k–e, k–x and SST models and measurements of a cylinder in cross flow. The SST and k–x models were used with automatic wall treatment (yþ < 1) as well as with scalable wall functions (9 < yþ < 25) . The heat transfer well on the upper part of the cylinder was predicted by the k–x and the SST models with a low-Re model. They also give a good prediction of the separation point. The k–e model failed to predict the heat transfer correctly, especially in the stagnation region and it also predicted the separation point too late. The k–

x and the SST models with scalable wall functions also performed poorly, which is supported by the result of Kondjoyan and Boisson (1997). It was predicted by the simulations that the heat transfer will continue to increase on the back of the cylinder.

5.4: k - omega Model:

The k - omega model was first created independently by Kolmogorov and later by Wilcox have continually refined and improved the model during the past three decades and demonstrated its accuracy for a wide range of turbulent flows. This model incorporates two key modifications:

1. the addition of a cross-diffusion term
2. a built-in stress-limiter modification

That makes the eddy viscosity a function of k, omega, and the ratio of turbulence-energy production to turbulence-energy dissipation.

In the k-omega model boundary layers and free shear flows are first dissected and analysed using perturbation methods and similarity solutions. All aspects of the model, including boundary conditions for rough surfaces and surfaces with mass injection, were reformulated and validated. Then a series of computations was performed for nearly 100 different applications, including free shear flows, attached boundary layers, backward-facing steps, and separated flows. The test cases cover all Mach-number ranges from incompressible through hypersonic. Wilcox in his paper presented complete details of the model's formulation, including all of the analysis, software, input data, and experimental data used in developing and testing the model.

5.5: k - epsilon Model:

A k- epsilon turbulence model has been formulated to capture the near–wall turbulence and low–Reynolds number effects. The turbulent kinetic energy k and the dissipation-rate epsilon can be evaluated using the T = k/epsilon-transport equation in conjunction with the Bradshaw and other empirical relations. Bradshaw– relation states that the shear stress in the boundary layer is proportional to the turbulent kinetic energy. A k-equation turbulence model in which epsilon is modelled phenomenologically has also been developed by Xu and Rahman. The extension of Bradshaw–relation down to the wall turns out to be of good accuracy, forming a new Reynolds–stress constitutive relation. The Bradshaw–relation has shown that the use of Bradshaw–relation is quite effective for non–equilibrium flows**.**

VI. Jet Flow Characteristics:

A high velocity jet of fluid flowing in a stagnant fluid medium can be categorized in two types i.e., free jet and impinging jet. A jet is called a free jet when there is no target surface. Otherwise, it is called an impinging jet. Also impinging jet flow can be divided into three basic components: free jet, stagnation flow and wall jet regions. The flow immediately following the jet exit is similar to free jet flow; hence it is termed the free jet region. When a free jet impinges in a stagnant fluid, a potential core region is formed where the jet center line velocity is the same as the exit velocity and there is a rapid increase in static pressure as the jet approaches the cylinder. Beyond the stagnation point, the flow accelerates and follows the curvature of the cylinder until the separation point, where the main flow becomes a wall jet. Below the separation point, the flow is reversed because of the separation of the flow, and the interaction with the solid surface under the cylinder. As the flow approaches the surface velocity of the jet decreases and pressure increases. The region where the jet hits the surface orthogonally the velocity of air is zero. This region is called the stagnation region. The high pressure differences between the stagnation region and the surrounding air forces the air to accelerate in the radial direction. Assuming there is no-slip condition, the air velocity is zero at the product surface, forming a boundary layer. This boundary layer resists the surface heat transfer. Boundary layer thickness increases from the stagnation point in a radial direction. The centripetal force due to curvature makes the flow unstable on the concave curvature, and the so-called Taylor-Gortler type vortex is generated (Gau and Chung, 1991). It has its axis parallel to the flow direction and is known to enhance the momentum and energy transfer and the surface heat transfer.

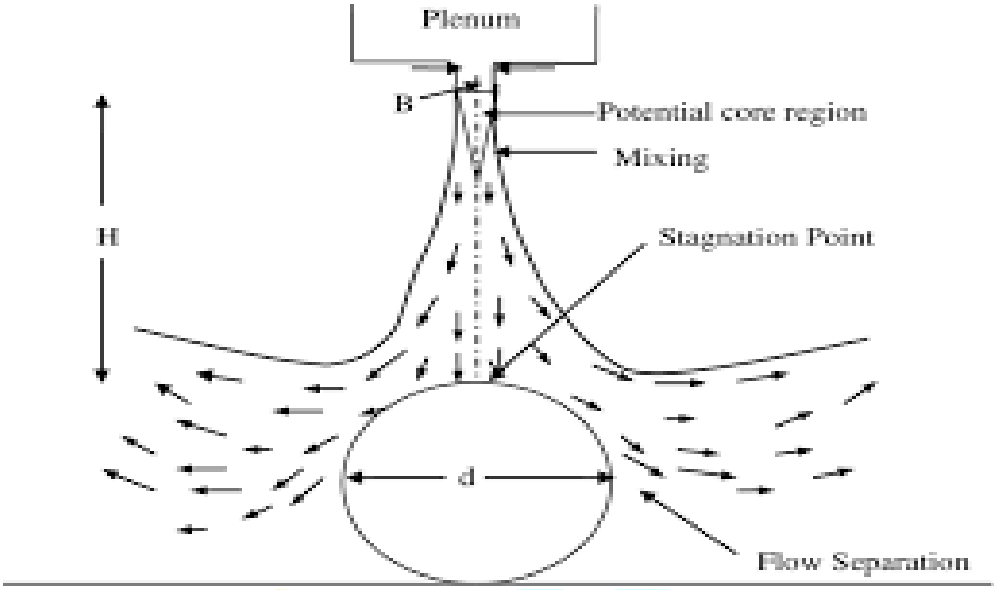

FIG.2: JET FLOW CHARACTERISTICS

VII. SUMMARY OF SOME IMPORTANT COMPUTATIONS ON JET IMPINGEMENT:

Authors	Configuration and important parameters studied	Summary of important observations
Olsson and Fuch	LES of semi-confined circular impinging jet at Re = 10000 with both dynamic and similarity SGS models.	Influence of the SGS model on velocity field was assessed. They reported that the similarity model performed better than the dynamic model at sufficient resolution.
Cziesla et al	Flow field and heat transfer for an impinging jet emanating from a rectangular slot nozzle at Re = 2000 - 10000 with dynamic SGS model.	They observed negative turbulent kinetic energy production rate close to the wall and jet centerline and a better accuracy in the heat transfer predictions at the stagnation region.
Tsubokura et al	Compared 3-D eddy structures for both plane and round impinging jets at Re = 6000 and with dynamic SGS model.	Twin vortices were observed in the impingement region of the plane jet. They also observed non organized structures in the stagnation region for round jet.
Beaubert and Viazzo	Plane jet impingement using the dynamic Smagorinsky model at Re = 3000 to 13500.	The wall shear stress at the impingement wall and effect of Re on the kinematic expansion of the jet were analysed.
Mingzhou et al	Flow field of a semi-confined rectangular exit turbulent impinging jet, on a flat surface at Re = 8500 and with dynamic SGS model.	They observed that the secondary vortices generated in the wall jet region were due to the periodic advancement of the primary vortex structure

Hadžiabdić and Hanjalić	Studied the vortical and turbulence structures with the local heat transfer in impinging flow with Re = 20000 using dynamic SGS model.	Their data provided clarifications for some of the experimentally noticed flow features, e.g., secondary peak in Nu and the negative turbulence energy production in the stagnation region.

VIII. CONCLUSIONS:

The heat transfer characteristics from a slot jet impinging on a circular cylinder placed on a solid surface has been studied using CFD simulations using software like ANSYS with the shear stress transport model (SST). After analysing the SST model through ANSYS using two equation models it was concluded that SST was the best model as reported in different literatures. The simulations predicted the heat transfer well on the upper part of the cylinder, but less correctly in the non-isotropic region in the wake. The effect of Reynolds number and the heat flow characteristics were discussed.
Other two models like k-epsilon and k-omega were also reviewed in the paper to draw conclusions on the basis of previous experiments done by researchers.

Acknowledgments:

Thanks to our professor and mentor Mr. Ankur Sachdeva for helping in preparing this review. We also thank our Head of Department of mechanical engineering Mr. Ashish Karnwal at KIET Group of Institutions for all the help.

REFERENCES

[1] Albertson, M.L., Dai, Y.B., Jensen, R.A., and Rouse H., 1950, Diffusion of Submerged Jets, Trans. Am. Soc. Civil Eng. 115(2409):639–697.

[2] Ashforth, S.F, Jambunathan, K., and Whitney C.F., 1997, Velocity and Turbulence Characteristics of a Semi Confined Orthogonally Impinging Slot Jet, Exp Thermal Fluid Sci. 14:60–67.

[3] Bórquez, R., Wolf, W., Koller, W.D., and Spieb W.E.L., 1999, Impinging Jet Drying of Pressed Fish Cake, J. Food Eng. 40(1-2):113–120.

[4] ANSYS. CFX 5.5 user manual.

[5] Baughn, J., Mesbah, M., & Yan, X. (1993). Measurements of local heat transfer for an impinging jet on a cylindrical pedestal. American Society of Mechanical Engineers, Heat Transfer Division, (Publication) HTD, 239, 57–62.

[6] E.E.M. Olsson & L.M. Ahrne (2003). Heat transfer from a slot air jet impinging on a circular cylinder.

[7] S.K. Singh and R. Paul Singh. Air Impingement Cooling of Cylindrical Objects Using Slot Jets.

[8] M. M. Rahman, V. Vuorinen, R. K. Agarwal, M. Larmi, Timo Siikonen. One-Equation Turbulence Model Based on k/ǫ.

[9] David C. Wilcox, Formulation of the k-! Turbulence Model Revisited.

[10] Anuj K. Shukla, Anupam Dewan, Flow and thermal characteristics of jet impingement: comprehensive review.