International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 3
March 2018

CFD Analysis of Effect of Reynolds Number on Local Heat Transfer Distribution for Jet Impingement on Smooth Plate by Incompressible Chevron Jet

Harpreet Singh
PG Student, Department of Mechanical Engineering,
Defence Institute of Advanced Technology (DU)
Dr. Sunil Chandel
Assistant Professor, Department of Mechanical Engineering
Defence Institute of Advanced Technology (DU)

ABSTRACT

In this paper, numerical simulation was performed to analyse the heat transfer performance of an incompressible hot jet by a chevron nozzle on a flat surface. Influence is studied at the nozzle to plate distance (z/d) 8 for three Reynolds numbers of 28,000, 35,000 and 40,000. Numerical analysis has been done by solving conservation equations of momentum, mass and energy with two equations based RNG k- ε turbulence model to determine the local distribution of Nusselt number on the plate. There is increase in the Nusselt number of plate by impingement of chevron jet as compared to the circular jet due to turbulence generated by chevrons. It was found out that with the increase in Reynolds number the Nusselt number increase at any given nozzle to plate distance (z/d).

Keywords

Chevron Nozzle, Nusselt Number, Heat Transfer, Computational Fluid Dynamics

NOMENCLATURE

a	-	Height of Chevron
b	-	Penetration depth
d	-	Diameter of nozzle
Nu	-	Nusselt number
Re	-	Reynolds number
z	-	nozzle to plate distance

INTRODUCTION

Jet impingement method is most common method used for enhancing heat transfer. These methods are used in steel manufacturing industries, in cooling of Gas Turbine blades, to cool down the material during grinding, in cooling of combustion chamber wall and paper and textile mills. Many researches are done to increase the heat transfer by changing various arrangements like non-circular nozzles (square, triangular, circular and elliptical), fixing the tabs at the nozzle exit (Chevron nozzle) and nozzle profiles (contoured nozzle, pipe nozzle and orifice). A chevron is basically a triangular tab which may be inclined at an angle towards central axis of pipe fixed at periphery of pipe exit. These tabs generate turbulence and hence increases heat transfer. Chevron nozzle is the least explored, although it appears to have more potential in increasing the heat transfer.

Many studies are carried out to study the method to increase the heat transfer. Jambunathan et al. [1] and Viskanta [2] provided comprehensive review on jet impingement heat transfer. They studied the impingement of jet for Reynolds number ranging from 5000-1,24,000 by changing the jet diameter, profile of flow device and nozzle to plate distance. Lytle and Webb (1994)[3] studied the distribution of Nusselt number on flat plate due to impingement of jet for low jet exit to plate spacing and Reynolds number from 3600 to 27600 and observed that there rise in the Nusselt number due to the transition from laminar to turbulent flow at plate. They further decrease the nozzle plate spacing which increases the velocity of the flow in the gap which is responsible for shifting the zone of the transition from laminar to turbulent flow towards the stagnation point.

Vinze et al. [4] conducted experiment and determine the effect of impinging jet temperature on heat transfer distribution on smooth plate. In his experiment, it was observed that heat transfer does not depend much on the temperature of jet. However, jet having ambient temperature does maximum heat transfer than compared to heated air jet. Another experiment is conducted by Vinze et al. [5] to study influence of shape of nozzle on heat transfer distribution. It was observed that the shape of nozzle has great influence on the heat transfer distribution. Circular jet experiences the highest rate of heat transfer compared that to square jet and then to triangular nozzle. It was also observed that the compressibility affects Nusselt number distribution only in the stagnation region and has negligible influence in transition and wall jet region.

Nabadavis et al. [6] did the numerical investigation emphasizes on studying the heat transfer characteristics when a high velocity air jet impinges upon a flat plate having constant heat flux. He observed that the heat transfer rate increases with the increase of Reynolds number of the jet and also there is an optimum value for jet distance to nozzle diameter for maximum heat transfer when all the other parameters were kept fixed.

Vinze et al. [7] carried an experimental study to investigate the influence of the chevrons on local heat transfer distribution for impinging incompressible jets at Reynolds number of 28,000, 35,000 and 40,000. He observed that Nusselt number increases with the increase in the number of chevrons and also with the increase in the tip angle for a given number of chevrons.

Yan et al. [8] numerically studied the heat transfer characteristics of a circular air jet impinging on a flat plate. He numerically studied seven turbulent models for predicting this type of flow and heat transfer is investigated and observed that the (RNG) k-epsilon model, (SST) k-omega model and the large Eddy simulation (LES) produces better results than others turbulent model.

GEOMETRY AND MODELLING

A. Geometric Description

For this simulation geometry is created using ANSYS SPACECLAIM design modeller. Figure 1 shows the physical domain of the modelling. Air flow at high velocity passes through chevron nozzle of diameter 10.8 mm having 4 numbers of triangular tabs. The length of each chevron is a = .43d i.e. 4.644 mm. Jet is impinged on a thin foil of stainless steel having size 150mm x 130mm and thickness of .08mm. This plate is not created in ANSYS SPACE CLAIM program because of its thickness .08mm, which is very less, it is difficult to perform meshing operation on the plate. Shell conduction technique is used in ANSYS FLUENT to specify thickness of the plate rather than creating it in ANSYS SPACECLAIM. The distance between the plate and the nozzle is z = 8d i.e. 86.4mm. An enclosure is created between the plate and nozzle having dimension of 150mm x 130mm x 93mm.

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 3
March 2018

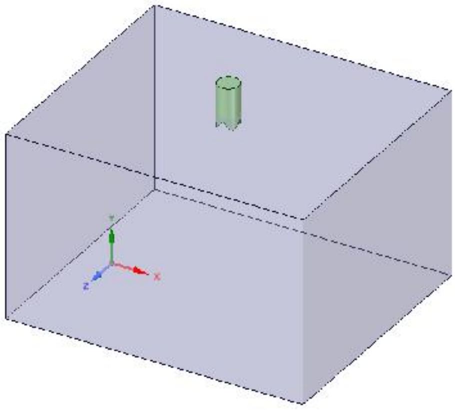

Figure 1: Physical domain

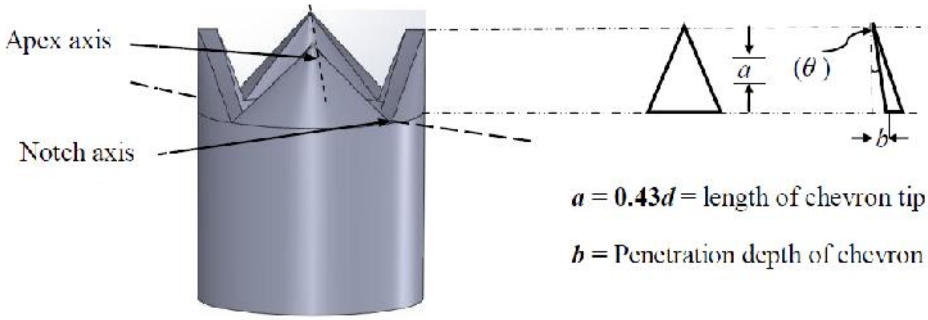

Figure 2: Geometry of chevron nozzle

B. Meshing

Unstructured mesh is created using ANSYS ICEM CFD software. Grid generation is one of the important stage of the simulation. Grid generation of chevron nozzle and whole geometry is shown in figure 3. Fine meshing is done on the nozzle as compared to the enclosure. Mesh independent study is done and it is found out that 5,73,211 number of nodes have 34,10,251 number of elements is the optimum value for the simulation.

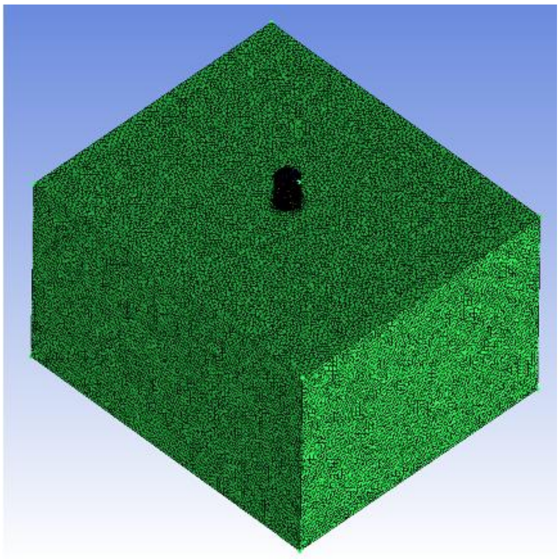

Figure 3: Mesh of the physical domain

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 3
March 2018

C. Turbulent Model

The RNG k-epsilon model was derived using a statistical technique called renormalization group theory. It is similar in form to the standard k-epsilon model which is the most common model used in CFD to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations. The first transported variable determines the energy in the turbulence and the second transported determines the rate of dissipation of the turbulent kinetic energy.

For turbulent kinetic energy k

$$\frac{\partial}{\partial}(\rho \ ) + \frac{\partial}{\partial x_i}(\rho \ u_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_e \ \ \frac{\partial}{\partial x_j}\right) + G_k + G_b - \rho \ \ - Y_M + S_K \qquad (1)$$

For dissipation ε

$$\frac{\partial}{\partial}(\rho \ ) + \frac{\partial}{\partial x_i}(\rho \ u_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_e \ \ \frac{\partial}{\partial x_j}\right) + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \qquad (2)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively. S_k and S_ε are user-defined source terms.

D. Energy Model

ANSYS Fluent solves the energy equation in the following form:

$$\frac{\partial[\rho \ ]}{\partial} + \nabla.\left[\vec{v}\,[\rho \ \ + p]\right] = \nabla.\left[K_e \ \ \nabla T - \sum_j h_j \vec{J_j} + (\bar{\bar{\tau}}_e \ \ . \vec{V})\right] + S_h \qquad (3)$$

where K_{eff} is the effective conductivity ($k+k_t$, where k_t is the turbulent thermal conductivity, defined according to the turbulence model being used), and J_j is the diffusion flux of species j. The first three terms on the right-hand side of represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. S_h includes the heat of chemical reaction, and any other volumetric heat sources defined.

E. Boundary conditions

Table 1 shows the mass flow rate of inlet of pipe for different Reynolds number having turbulence intensity of 5% and hydraulic diameter of .0108 m. The direction of flow of air is at negative-y direction. The temperature at inlet of the nozzle is set to 350 K. The wall of the nozzle is thermally insulated, so heat flux through the nozzle wall is set to 0. All the sides of enclosure except the plate side of the enclosure are set to pressure - outlet. Thickness is given to plate side of enclosure using shell conduction technique of .08 mm and temperature of 300 K.

Table 1: Mass flow rate for different Reynolds number

Reynolds Number	**Mass flow rate (Kg/sec)**
28,000	0.00454
35,000	0.00559
40,000	0.00643

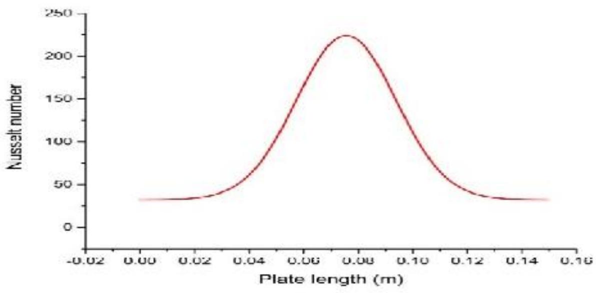

Figure 4: Local distribution of Nusselt number for Reynolds number 28,000

International Journal of Engineering Technology Science and Research
IJETSR
www.ijetsr.com
ISSN 2394 – 3386
Volume 5, Issue 3
March 2018

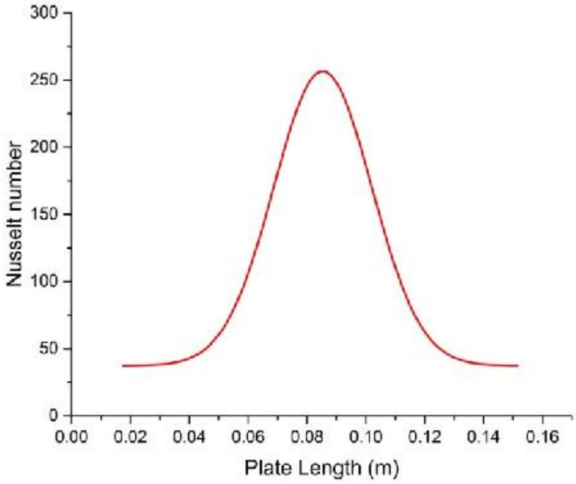

Figure 5: Local distribution of Nusselt number for Reynolds number 35,000

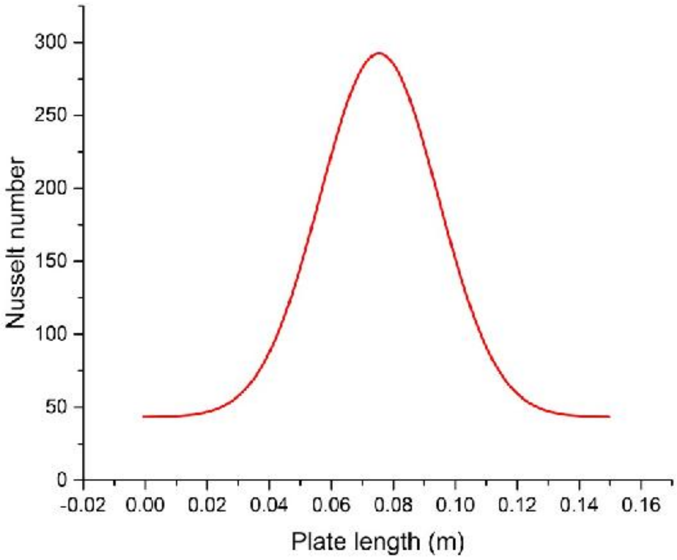

Figure 6: Local distribution of Nusselt number for Reynolds number 40,000

RESULTS AND DISCUSSION

In the present study simulation are performed for nozzle having 4 chevrons and at three Reynolds numbers 28,000, 35,000 and 40,000 based on pipe diameter. The Mach number is in the range of 0.014-0.2, hence the jets are incompressible. Figure 4, Figure 5 and Figure 6 shows the local Nusselt number distribution for Reynolds number 28,000, 35,000 and 40,000 respectively.

CONCLUSION

Numerical study is carried out to study the influence of Reynolds number on local heat transfer distribution by chevron jet. It is concluded that with the increase in Reynolds number the Nusselt number increases but there is no effect observed on local distribution of Nusselt number by change in Reynolds number.

REFERENCES

[1] Viskanta R.,"Heat transfer to impinging isothermal gas and flame jets", Experimental Thermal and Fluid Science (1993), Vol. 6, 111-134

[2] Jambunathan K., Lai E., "Moss M.A. and Button B.L., A review of heat transfer data for single circular jet impingement", International journal of Heat and Fluid Flow (1992).

[3] Lytle D. and Webb B.W., "Air jet impingement heat transfer at low nozzle-plate spacing", International Journal for Heat and Mass Transfer (1994), Vol. 37, 1687-1697.

[4] Ravish Vinze, S. Chandel, M.D. Limaye, S.V. Prabhu, "Influence of jet temperature and nozzle shape on the heat transfer distribution between a smooth plate and impinging air jets", International Journal of Thermal Sciences (2016), 136-151.

[5] Ravish Vinze, S. Chandel, M.D. Limaye, S.V. Prabhu, "Effect of compressibility and nozzle configuration on heat transfer by impinging air jet over a smooth plate", International Journal of Thermal Sciences (2016), 136-151.

[6] Asem Nabadavis and Dipti Prasad Mishra, "Numerical Investigation of Jet Impingement Heat Transfer on a Flat plate", Carbon – Science and Technology (2016), 1-12.

[7] Ravish Vinze, S. Chandel, M.D. Limaye, S.V. Prabhu, "Local heat transfer distribution between smooth flat surface and impinging incompressible air jet from a chevron nozzle", International Journal of Thermal Sciences (2016), 136-151.

[8] Y. Y. Yan, J. Maltson, "Numerical Study on Stagnation Point Heat Transfer by Jet Impingement in a Confined Narrow Gap", University of Nottingham.