IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

CFD Analysis of Investigation of Heat Transfer Using Single Swirling Jet Air Impingement with Twisted Tapes

Mr.Jaspreet Singh[1], Mr.Rakesh Kumar[2], Mr.Parampreet Singh[3]

Student, Mechanical, NWIET, Moga, India [1]

Assistant Professor, Mechanical, NWIET, Moga, India [2]

Assistant Professor, Mechanical, LLRIET, Moga, India[3]

Abstract: This paper reports the computational fluid dynamics (CFD) modelling of heat transfer analysis from a heated target surface, using single swirling jet air impingement at different Reynolds numbers. Swirl is produced by using a nozzle body with half-length downstream twisted tape insert (HLDI). The nozzle to plate (target surface) distance and twist ratio (y) of twisted tape is taken to be constant as 21mm (H/D = 1) and y = 2.93 respectively. Different Reynolds numbers as 12000, 17000, 22000 and 27000 are used to investigate the heat transfer characteristics on heated surface.

Keywords: CFD, Swirling jet, Reynolds number, Heat transfer.

I. INTRODUCTION

Jet impingement is one of the most active techniques for heat transfer enhancement. Because of high heat transfer rate and other advantages, the technique has been widely used in many engineering and other areas for example cooling of turbine blades, chemical vapour deposition, drying of food products, cooling of hot steel plate, tempering of glass, papers, textiles and films, cooling of electronic equipment, and cooling of outer wall of the combustion chamber. Large quantity of literature is available on different configurations as single, row, and array of jets with co-relations developed for heat and mass transfer characteristics related to this field. The major disadvantage of conventional jet impingement is that the local heat flux can be highly non-uniform (Viskanta, 1993) [1]. To overcome this problem of radial non-uniformity several attempts have been made with different configurations. The introduction of swirl in the jet solved this problem up to much extent. Ward and Mahmood (1982)[2] showed that a swirling jet provided more uniform radial distribution of heat transfer as compared to non-swirling jet. A number of experimental, empirical and numerical studies have been conducted. Excellent review papers (Viskanta, (1987)[3] and (Zuckerman and Lior, (2006)[4] were published highlighting different issues. Carefully controlled experiments are required to improve the understanding of heat transfer characteristics from the impingement plate. Huan and Genk (1996)[5] conducted experiments to investigate and compare the performance of swirling and multi-channel impinging jets with that of a conventional impinging jet (CIJ), having the same diameter at the same conditions. Salce and Simon (1991)[6] also tried to visualize the swirl air flow inside a cylindrical cavity using the smoke wires technique. The images of the flow field, however, were not clear and didnot show the whole flow field as compared to the Shlienand Hussain (1983)[7] Different techniques are used by different 0heat transfer coefficient. One of the experimental techniques is thermo chromic liquid crystal (TLC).

This method is used by Nuntadusit, and Wae-hayee (2012)[8]. Flow and heat transfer characteristics of swirling impinging jet (SIJ) were studied experimentally at constant nozzle-to-plate distance of L=4D. Hunag et al. (1998)[9] used a swirl generator made of cylindrical plug with four narrow channels to provide swirl to single and multiple air impinging jets. It has been observed that swirling impinging jet (SIJ) demonstrated large increase in Nusselt number and significant improvement in radial uniformity of heat transfer as compared to multi-channel impinging jets (MCIJ) and conventional impinging jet (CIJ) which is contrary to results observed by Ward and Mahmood (1982)[10] and Lee et al. (2002)[11]. This was attributed to the fact that turbulence generated due to sudden

II. NUMERICAL SETUP

The FLUENT-6.3.26 commercial CFD package was used to perform 3D calculations of the convective heat transfer on the impinged plate, which is associated with swirling impinging jets with different Reynolds numbers. The computational domain consisted of the entire pipe-plate assembly as shown in fig.1. The geometry of the problem was created for air flowing in a tube of 21 mm diameter (D) with twisted tape as swirler (HLDI) and length (L) 300 mm impinging onto a flat plate heated at uniform heat flux of 3000 W/m^2. The geometry was tested for different Reynolds numbers as 12000, 17000, 22000, and 27000. The configuration chosen for study is shown in fig.2,

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

where HLDI stands for half-length downstream insert configuration.

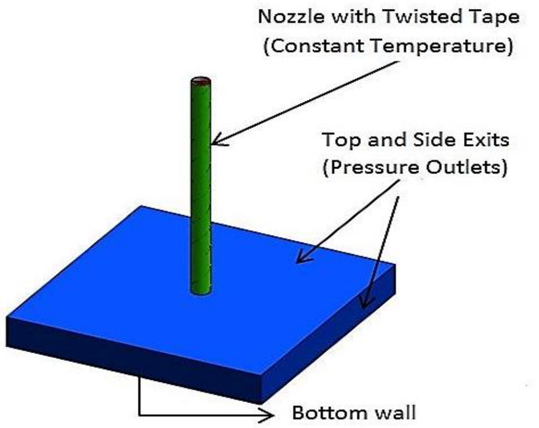

Fig.1Entire pipe-plate assembly

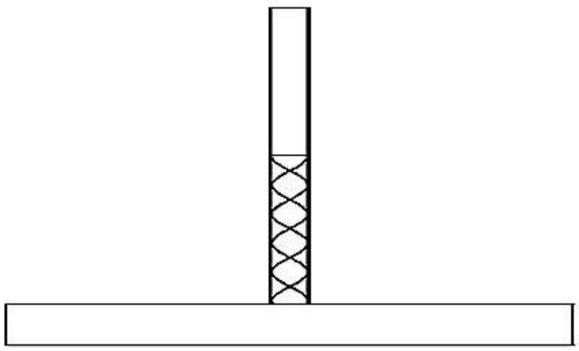

Fig2The configuration chosen for study

III. NUMERICAL SIMULATION

Fig.3 shows the diagram of computational domain used for numerical simulation in FLUENT 6.3.26 and Table 1 explains the boundary conditions applied at various regions of the computational domain used for numerical simulation.

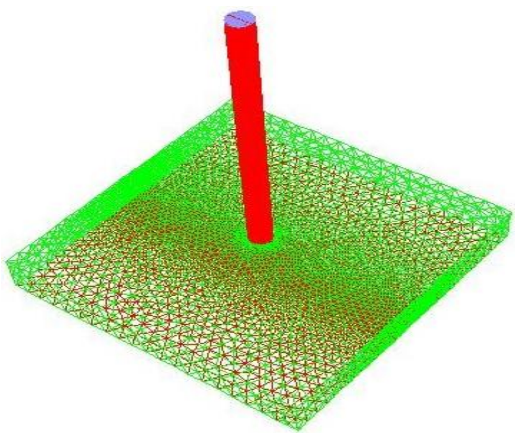

Fig. 3 Computational domain used for numerical simulation

Numbers of cells used for the domain are around 12, 50,808 which are sufficient to resolve the fluid flow. The flow and turbulence fields have to be accurately solved to obtain reasonable heat transfer predictions. Second order scheme is used for all terms that affect heat transfer. Second order discretization scheme is used for the pressure; second order upwind discretization scheme is used for momentum, turbulence kinetic energy, specific dissipation rate, and the energy. Flow, turbulence. The standard SIMPLE algorithm is adopted for the pressure velocity coupling. The simulation type is steady state condition, convergence criteria is specified as 10E-05 residuals and convergence control is set at maximum 1000 iterations which can be changed if convergence is not achieved. The three dimensional Navier-Stokes and energy equations with the standard turbulence model are solved using CFD software (FLUENT 6.3.26) which are combined with continuity and momentum equations to simulate thermal and turbulence flow fields. The turbulence model used is shear stress transport (SST) ‘k-ω’ model which is found to work the bestamong the available turbulence models for this flow configuration and is also chosen due to its simplicity, computational economy and wide acceptability.

Table 1.Boundary condition

Physical Location/ Geometrical Identity	**Boundary Type**	**Colour**
Top Surface/Opening/Outlet	Pressure Outlet	Green
Bottom Surface/ Impingement Plate	Wall	Red
Side/ Opening/ Outlet	Pressure Outlet	Green
Centre/Twisted Tape	Wall	Red
Inlet/Nozzle Inlet	Velocity Inlet	Blue
Nozzle Wall	Wall	Red

IV. GOVERNING EQUATIONS FOR HEAT TRANSFER

Heat transfer from solid boundary to moving fluids can be described using Newton’s law of cooling:

$$q = h(T_\infty - T_S) \qquad (1)$$

where, ‘q’is the heat flux from the solid to the fluid, ‘h’ is the heat transfer coefficient, ‘T_∞’ is the temperature of the bulk of the fluid and ‘T_S’ is the temperature of the solid surface. The temperature difference between the solid surface and the bulk of the fluid is the driving force for convection.

The heat transfer coefficient ‘h’ is often non-dimensionalized using the thermal conductivity ‘k_t’ and length scale ‘D’ which characterizes the geometry. This

IARJSET
ISSN (Online) 2393-8021
ISSN (Print) 2394-1588
International Advanced Research Journal in Science, Engineering and Technology
Vol. 3, Issue 6, June 2016

results in a non-dimensional number called Nusselt's number (Nu):

$$Nu = \frac{hD}{k_t} \quad (2)$$

The Nusselt's number (Nu) is often a complex function of the geometry, the flow velocity and the physical properties of the fluid. Usually, correlations are derived between the Nusselt's number (Nu), Reynolds number (Re), Prandtl number (Pr) and a non-dimensional function of geometry.

$$Pr = \frac{\mu c_p}{k_t} \quad (3)$$

So,

$$Nu = \frac{q}{(T_\infty - T_S)} \cdot \frac{Pr}{\mu c_p} . D \quad (4)$$

V. GRID DEPENDENCY TEST

The dependency of the grid affects the results of simulation directly as shown in table 2. Grid dependency tests were performed comparing the variation in maximum values of Nusselt's number. The results were calculated in case of without twisted tape insert at Reynolds number 12000 and jet to plate distance 2 (H/D=2) .The results of comparison are listed in the following table. From the table it is clear that 12, 50,808 cells case will give reliable results at minimum computation time. Numbers of cells used for the domain are around 12, 50,808 which are sufficient to resolve the fluid flow

Table 2.Grid dependency results

Sr. No.	Number of cells	Nusselt's number
1	08,20,750	88.3
2	09,10,626	70.4
3	10,13,629	102.6
4	11,26,327	125.6
5	12,50,808	133.5
6	13,40,936	136.4
7	14,35,629	134.3
8	15,16,535	132.2

V. SWIRLING IMPINGING JET (SIJ) SIMULATION AND RESULTS

Effect of Reynolds number on the heat transfer characteristics can be studied by simulating the fluid flow for different Reynolds number at same separation distance and twist ratio.

The separation distances (H/D) used for the geometry considered is 'H/D' = 1 with twist ratio of twisted tape insert 'y' = 2.93.Reynolds number is varied as 'Re' = 12000, 17000, 22000 and 27000. The effect of Reynolds number on Nusselt's number and heat transfer coefficient is shown in fig. 4 and 5 respectively.

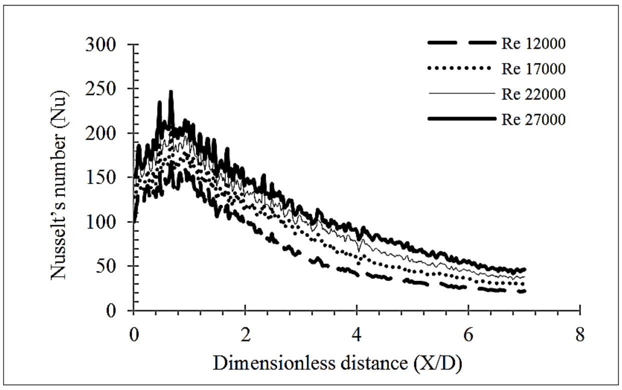

Fig. 4: Variation of Nusselt's number w.r.t. 'X/D' for various values of Reynolds number at 'H/D' = 1 and for 'y' = 2.93

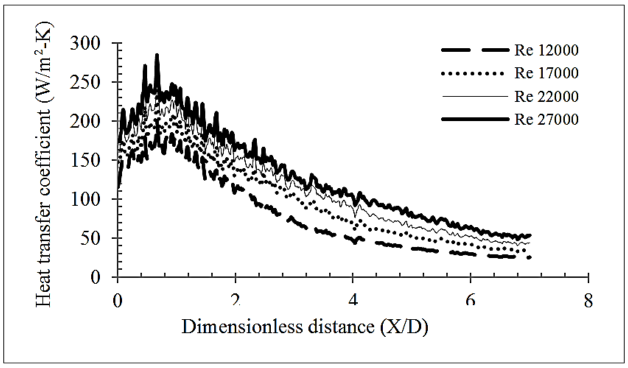

Fig. 5: Variation of heat transfer coefficient w.r.t. 'X/D' for various values of Reynolds number at 'H/D' = 1 and for 'y' = 2.93

Effect on Nusselt's number:

[1] Nusselt's number variation with Reynolds number shows that the values of 'Nu' increase with the increase in 'Re' as anticipated by the physics of the problem. The values of 'Nu' at various Reynolds number for same separation distance and configuration are given as under:

[2] At 'H/D' = 1, and for the of configuration half-length downstream twisted tape insert with twist ratio of 2.93 the value of Nusselt's number at stagnation point i.e. 'X/D' = 0 is 100.21 for 'Re' = 12000 this value increased to 127.86 for 'Re' = 17000, 141.51 for 'Re' =22000 and 150.34 for 'Re" = 27000. The value of Nusselt's number is found to be 174.18 for 'Re' = 12000 at 'X/D' = 0.66 and at this 'X/D' value of Nusselt's number is increased to 207.86 for 'Re' = 17000, 229.17 for 'Re' = 22000 and 247.05 for 'Re' = 27000 which is the maximum value of Nusselt's number in this configuration.

Effect on heat transfer co-efficient:

[3] Effect on heat transfer co-efficient variation with Reynolds number shows that the values of Effect on heat

DOI 10.17148/IARJSET.2016.3633

transfer co-efficient increase with the increase in 'Re' as anticipated by the physics of the problem. The values of Effect on heat transfer co-efficient at various Reynolds number for same separation distance and configuration are given as under:

At 'H/D' = 1, and for the configuration of half-length downstream twisted tape insert with twist ratio of 2.93 the value of heat transfer co-efficient at stagnation point i.e. 'X/D' = 0 is 115.49 for 'Re' = 12000 this value increased to 147.35 for 'Re' = 17000, 163.08 for 'Re' =22000 and 173.25 for 'Re" = 27000.

The value of heat transfer co-efficient is found to be 200.73 for 'Re' = 12000 at 'X/D' = 0.66 and at this 'X/D' value of heat transfer co-efficient is increased to 239.53 for 'Re' = 17000, 264.09 for 'Re' = 22000 and 284.70 for 'Re' = 27000 which is the maximum value of heat transfer co-efficient in this configuration. Figure 6 and 7 shows the average Nusselt's number and Heat transfer coefficient values at different Reynolds numbers

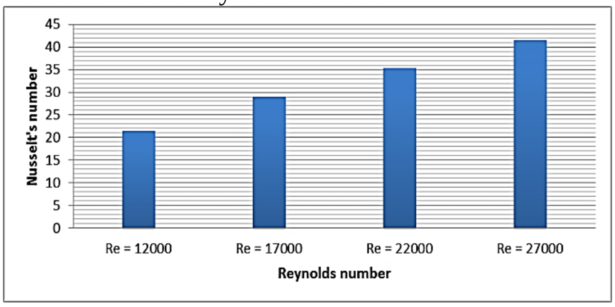

Fig. 5: Variation of average Nusselt's number at different Reynolds numbers

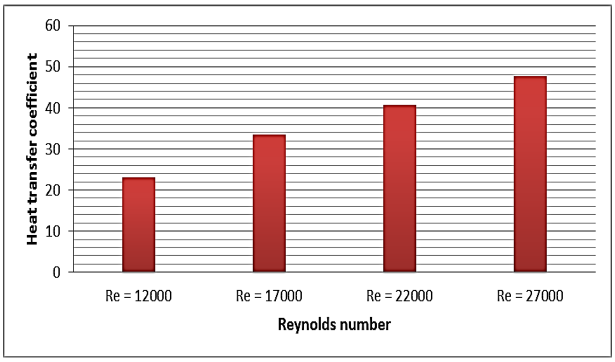

Fig. 6: Variation of average heat transfer coefficient at different Reynolds numbers

V. CONCLUSION

A way to enhance the heat transfer is increasing the turbulence intensity of the jet stream. This is because the mixing in the turbulent flow is on macroscopic scale with groups of particles transported in zigzag path through the fluid. The exchangement mechanism is many times more effective than in laminar flow. Consequently, in turbulent flow, the rates of heat and momentum transfer very large as compared to laminar flow. It is found that enhancements due to turbulence attain maximum values as the turbulence intensity increases for a specific Reynolds number. Though increasing the jet velocity is a natural way to improve the turbulence intensity and this can be done by increasing the Reynolds number. Thus increase in Reynolds number increases the values of Nusselt's number as well as heat transfer rates. The following figure 7 shows the Nusselt's number contours for different values of Reynolds number.

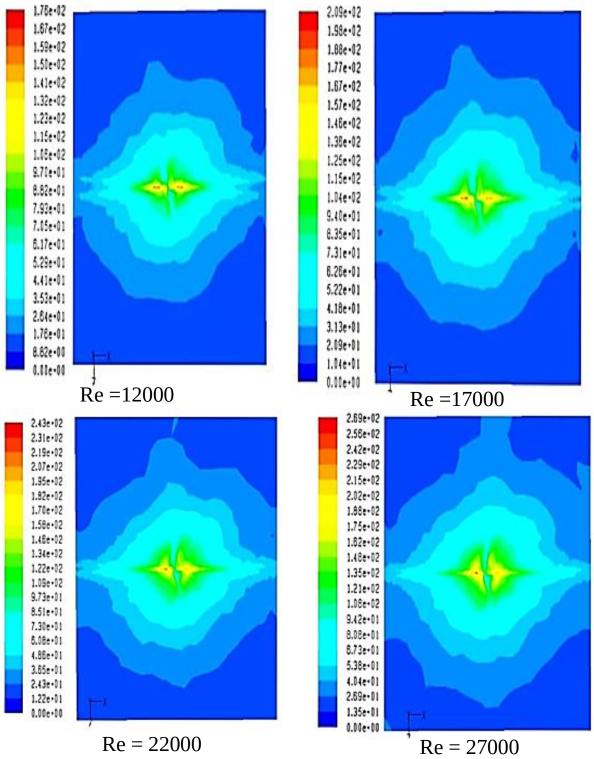

Fig. 7: Map of Nusselt number contours at different Reynolds numbers

Nomenclature

D : Nozzle diameter, m
h: Local heat transfer coefficient, w/m2k
k: Thermal conductivity of air, w/m· k
Nu: Local Nusselt number$q*$: Convective heat flux
r: Radial distance from the stagnation point,
X/D: Dimensionless radial distance
Re: Reynolds number based on the exit diameter, Re = $\rho uD/\mu$
u: Mean velocity at jet exit, m/sρ: Density of air, Kg/m3
μ: Kinetic viscosity of air, Kg/m·s

ACKNOWLEDGMENT

The author sincerely would like to thank the following for helpful discussions: **Mr.Parampreet Singh, Mr.Rakesh Kumar, Mr.Gurpreet Singh Sondh**and**Mr.Gagandeep Singh.**

REFERENCES

[1] Viskanta, R. (1993). "Heat transfer to impinging isothermal gas and flame jets," Experiments on Thermal Fluid Sciences, Vol.6, pp. 111 – 134
[2] Ward, J., Mahmood, M., 1982. "Heat transfer from a turbulent, swirling impinging jet". 7th International Heat Transfer Conference. Vol. HTD-3. DD. 401407.

[3] Viskanta, R. and Beckermann, C. (1987). "Forced convection boundary layer flow and heat transfer along a flat plate embedded in a porous medium," International Journal of Heat and Mass Transfer, Vol.30, pp. 1547 - 1551.
[4] Zuckerman, N. and Lior, N. (2006). "Jet impingement heat transfer: Physics, correlations, and numerical modeling," Advances in Heat Transfer, Vol.39.
[5] Huang, L., El-Genk, MS., 1998. "Heat transfer and flow visualization experiments of swirling, multichannel, and conventional impinging jets". International Journal of Heat and Mass Transfer 41, 583-600.
[6] Salce, A. and Simon, T. W., "Investigation of the effects of flow swirl heat transfer inside cylindrical cavity" Journal of Heat Transfer, 1991, 113, 5.
[7] Shlien, D. J. and Hussain, A. K. M. F., "Visualization of the large-scale motion of a plane jet," flow visualization. Proceedings of the 3rd International Symposium of Flow Visualization, 1983.
[8] C. Nuntadusit a,, M. Wae-hayee a, A. Bunyajitradulya b, S. Eiamsa-ard(2012) "Visualization of flow and heat transfer characteristics for swirling impinging jet" International Communications in Heat and Mass Transfer 39 (2012) 640–648
[9] Hunag. andSaha, S.K. (1998). "Turbulent flow and heat transfer from a slot jet impinging on a moving plate," International Journal of Heat and Fluid Flow, Vol.24, pp. 685 - 697.
[10] Ward, J., Mahmood, M., 1982. "Heat transfer from a turbulent, swirling impinging jet". 7th International Heat Transfer Conference. Vol. HTD-3. DD. 401407.
[11] Lee, J., Lee, S.J. (2002). "Stagnation region heat transfer of a turbulent axisymmetric jet impingement," Experimental Heat Transfer, Vol.12. pp. 139 - 156.