Ilkogretim Online - Elementary Education Online, 2020; Vol 19 (Issue 1): pp. 1114-1121
http://ilkogretim-online.org
doi: 10.17051/ilkonline.2020.661931

Computational Analysis Of Heat Transfer & Flow Characteristics Of Swirl Flow Jet Impingement Cooling

[1] V.Chokkalingam , [2]S.Rajendran , [3]T.Paramaguru & [4] K.Sakthivel

[1, 2,] Professor, Department of Mechanical Engineering, Jaya Engineering College Chennai 602024,INDIA.

[3, 4.] Assistant Professor, Department of Mechanical Engineering, Jaya Engineering College Chennai, INDIA

Email: lingamviswakethu@gmail.com

Abstract

Impinging jets have been widely used for increasing heat transfer in engineering applications such as cooling of hot steel plates and turbine blades, tempering of glass, drying of papers and films and cooling of electronic components. The objective of the study is to investigate the characteristics of the heat transfer for swirling jet impingement on a surface which aims to enhance the heat transfer rate and also to compare heat transfer characteristics of straight Impinging jet and swirl flow impinging jet. The present study focuses on the verification of the swirling jet effect on the distribution of the local heat transfer coefficient on the impinged target surface. Studies would be conducted for a wide range of parameters including Reynolds Number, jet impingement angle, swirl Pitch and nozzle to target plane spacing. The motivation of the present work is to explore the efficiency of swirling jet impingement cooling and understand the mechanisms by which heat is removed from a constant heat flux surface.

1. Introduction

Jet impingement systems provide an effective means for the enhancement of convective processes due to the high heat and mass transfer rates that can be achieved. The range of industrial applications that impinging jets are being used in today is wide. In the annealing and tempering of materials, impinging jet systems are finding use in the cooling of hot metal, plastic, or glass sheets as well as in the drying of paper and fabric. Compact heat exchangers, with applications in the aeronautical or the automotive sector, often use multiple impinging jets in dense arrangements. Impingement systems in micro scale applications are commonly used for the cooling of electronic components, particularly electronic chips. In gas turbine applications, jet impingement has been routinely used for a long time. Requirements are being imposed by demands for increased power output and efficiency as well as for reduced emissions. High thermal efficiency can be realized by increasing turbine inlet temperatures and compressor ratios. As a result of this, many gas turbine components, such as rotor disks, turbine vanes and blades, or combustion chamber walls, are operated at temperatures well above highest allowable material limits. In order to assure durability and long operating intervals, effective cooling concepts are required for these highly loaded components.

Three zones can be identified in an impinging jet flow. These are illustrated in figure 1. There is the free jet zone, which is the region that is largely unaffected by the presence of the impingement surface; this exists beyond approximately 1:5 diameters from the impingement surface.

A potential core exists within the free jet region, within which the jet exit velocity is conserved and the turbulence intensity level is relatively low. A shear layer exists between the potential core and the ambient fluid where the turbulence is relatively high and the mean velocity is lower than the jet exit velocity. The shear layer entrains ambient fluid and causes the jet to spread radially. Beyond the potential core the shear layer has spread to the point where it has penetrated to the centre line of the jet. At this stage the centre line velocity decreases and the turbulence intensity increases. Figure 1 also identifies a stagnation zone that extends to a radial location defined by the spread of the jet. The stagnation zone includes the stagnation point where the mean velocity is zero and within this zone the free jet is deflected into the wall jet flow. Finally, the wall jet zone extends beyond the radial limits of the stagnation zone.

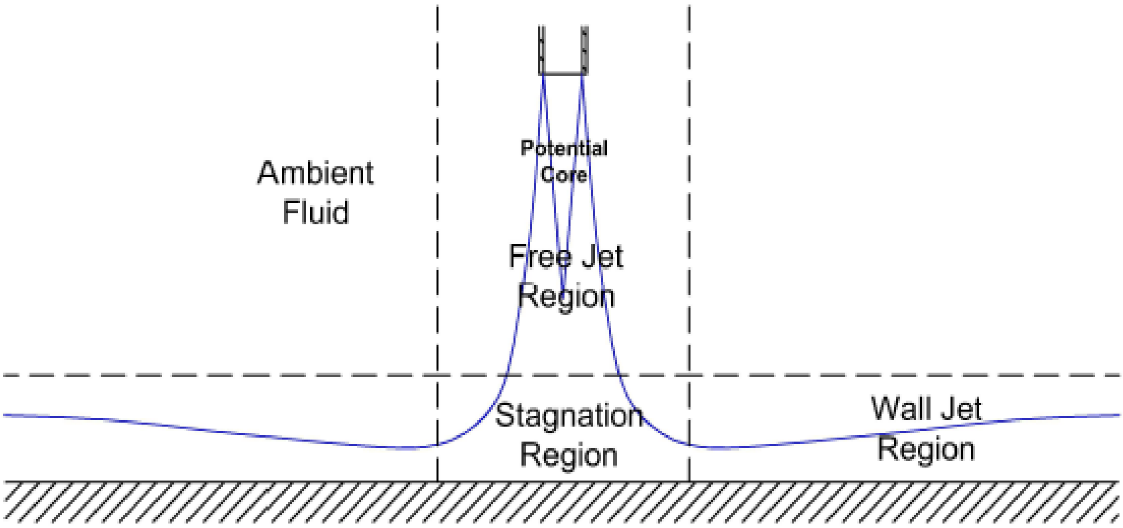


Fig 1. Zones of Impinging Jet Flow

2. Basics of computational fluid dynamics

CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyze a fluid problem requires the following steps. First, the mathematical equations describing the fluid flow are written. These are usually a set of partial differential equations. These equations are then discretized to produce a numerical analogue of the equations. The domain is then divided into small grids or elements. Finally, the initial conditions and the boundary conditions of the specific problem are used to solve these equations. The solution method can be direct or iterative. In addition, certain control parameters are used to control the convergence, stability, and accuracy of the method.

All CFD codes contain three main elements:

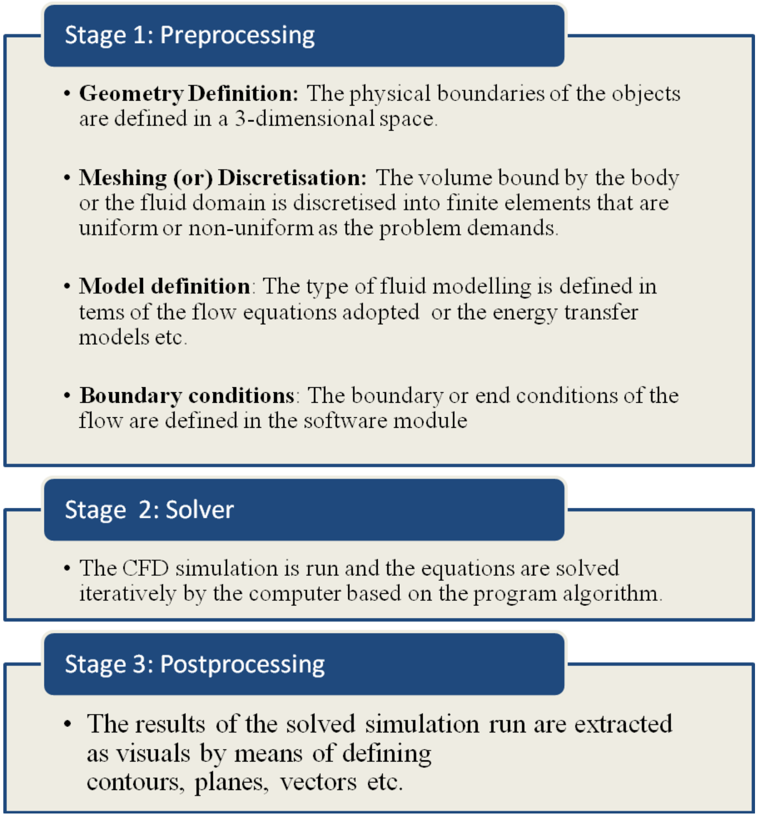
(1) A pre-processor, which is used to input the problem geometry, generate the grid, and define the flow parameter and the boundary conditions to the code.

(2) A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as a flow solver: finite difference method, finite element method,finite volume method, and spectral method.

(3) A post-processor, which is used to massage the data and show the results in graphical and easy to read format.

3. CFD Methodology

Computational Fluid Dynamics software –either manually written code or the commercially available packages go by a particular series of steps that are universal to any method or approach to a CFD problem.



Stage 1: Preprocessing

- **Geometry Definition:** The physical boundaries of the objects are defined in a 3-dimensional space.
- **Meshing (or) Discretisation:** The volume bound by the body or the fluid domain is discretised into finite elements that are uniform or non-uniform as the problem demands.
- **Model definition**: The type of fluid modelling is defined in tems of the flow equations adopted or the energy transfer models etc.
- **Boundary conditions**: The boundary or end conditions of the flow are defined in the software module

Stage 2: Solver

- The CFD simulation is run and the equations are solved iteratively by the computer based on the program algorithm.

Stage 3: Postprocessing

- The results of the solved simulation run are extracted as visuals by means of defining contours, planes, vectors etc.

Fig 2. Flow Chart for CFD Methodology

4. Results and discussion

4.1 Non swirl flow

4.1.1 Effect of flow parameters

Details of exit flow structures in the case of an annular impinging jet with swirl and non swirl have been discussed in this section. Figure 3 shows velocity vector distribution for non swirl jet with varying H/D distances. It is observed from the result that for H/D = 5 recirculation occurs symmetry to nozzle axis. This is due to reverse flow after impingement. This will obviously enhance the heat transfer rate. In the case of H/D = 7.5 no such recirculation zones in the reverse flow. When the H/D is further increased the stream of jet is carried away by fluid particles.

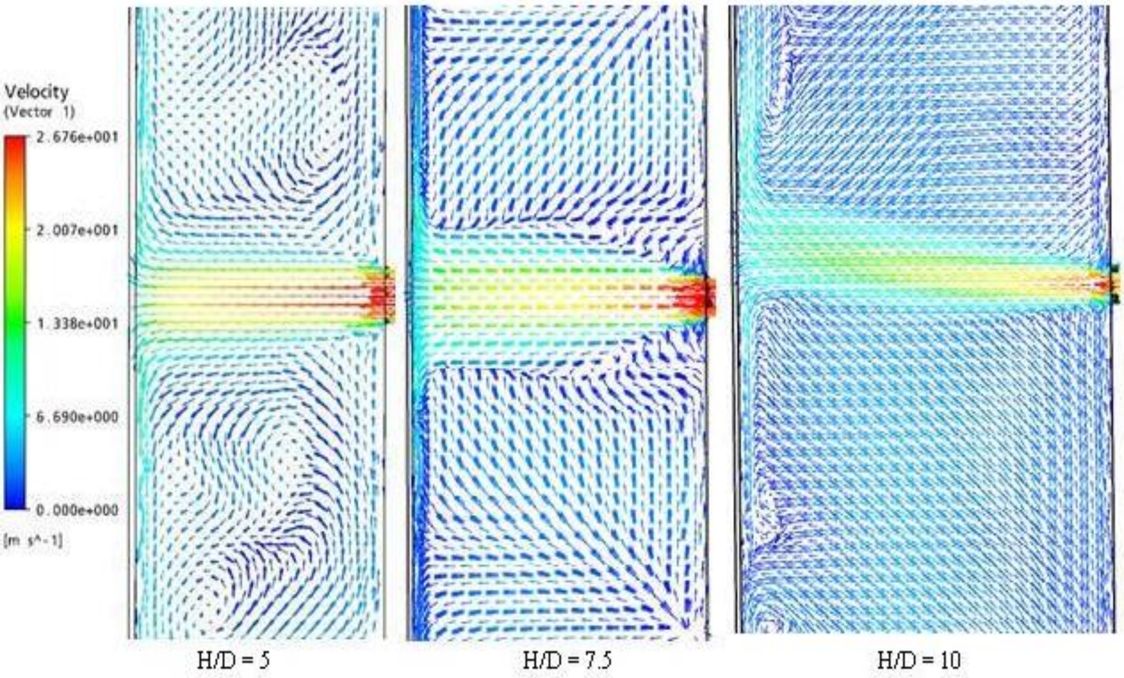


Fig.3 Velocity vector for varying H/D distance

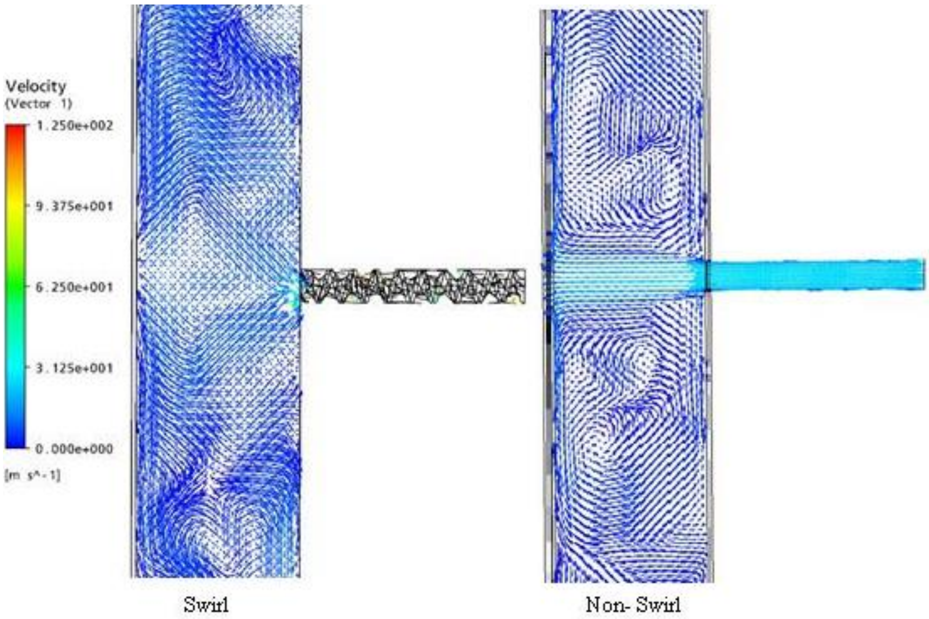


Fig.4 Velocity vectors for Swirl and Non swirl flow

Figure 4 shows exit flow pattern with separation distance of H/D= 5. The result shows the exit flow acquires radial velocity components due to the presence of swirl diverging radially as soon as it exits the annular nozzle. The flow field of swirl jet shows that increased uniformity in the radial velocity component with respect to nozzle axis. Strong recirculation zones have been observed in swirl jet. In the case of non swirl no such radial component of velocity has been observed. Recirculation in flow has been observed for this case about the flow axis due to reverse flow.

4.2 SWIRL FLOW

4.2.1 EFFECT OF FLOW PARAMETERS

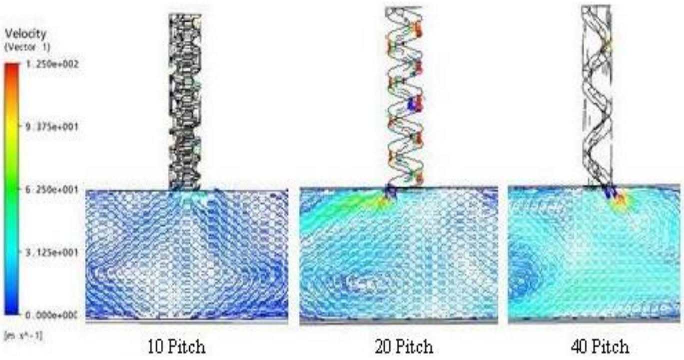


Fig.5 Velocity vector for varying Pitch distance

Details of exit flow structures in the case of an annular impinging jet with swirling have been discussed in this section. Fig.6 shows exit flow pattern with separation distance of H/D=5. The result shows the exit flow acquires radial velocity components due to the presence of swirl diverging radially as soon as it exits the annular nozzle. The result of 10 pitch shows that increased uniformity in the radial velocity component with respect to nozzle axis.

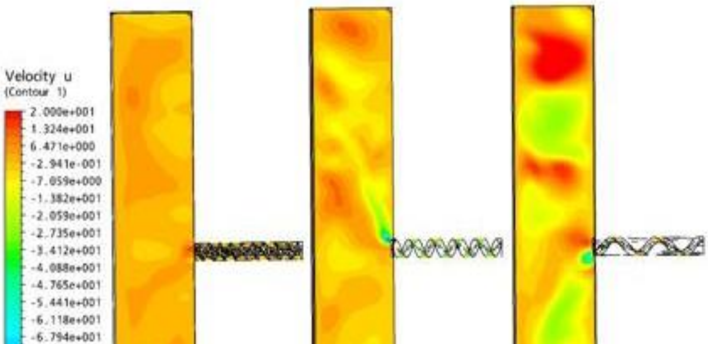


Fig.6 Distribution of axial velocity component

The distribution of axial velocity components for the inserts with varying pitch distances. The result of 40 pitch distance shows that concentrated axial velocity components when the reverse flow occurs.

This is has been reduced in the case of 20 pitch distance. In the case of 10 Pitch insert comparatively reduced axial component of velocity distribution has been observed. This attributes increased radial velocity components in this case.

5. Conclusion

Thus the heat transfer and flow analysis of swirl flow jet impingement cooling was carried out and the following conclusion has been drawn When the separation distance (H/D) increases the radial uniformity of Nusselt number decreases and value of Nusselt number decreases. Increased radial distribution of Nusselt number has been observed in the swirl flow but comparatively Nusselt number decreases slightly. When the Reynolds number increases the radial uniformity of Nusselt number decreases in both cases. Increased Radial velocity components have been observed in the swirl flow and it leads to comparatively stronger recirculation zones. At higher Reynolds number the value of Nusselt number is high.

REFERENCES

[1] C. Nuntadusit and M. Wae-hayee (2012), Visualization of flow and heat transfer for swirling impinging jet, International Communications in Heat and Mass Transfer – Science Direct 39 640–648.

[2] K. Bakirci, K. Bilen (2007), Visualization of heat transfer for impinging swirl flow, Experimental Thermal and Fluid Science- Science Direct 32 182–191.

[3] H.T. Xu, and J.L. Niu (2003), A New Method Of CFD Simulation Of Airflow Characteristics Of Swirling Floor Diffusers, Eighth International IBPSA Conference, Netherlands, August 11-14.

[4] Karl J. Brown and Darina B. Murray (2009), Heat Transfer Characteristics Of Swirling Impinging Jets, Proceedings of the ASME International Mechanical Engineering Congress & Exposition, USA.

[5] Zhou Jie-min, Fu Yi-feng (2006), Study on Compulsive Cooling of Straight Impinging Jet Array and Swirling Jet Impingement, IEEE.

[6] H. Herwig G. Middelberg, The physics of unsteady jet impingement and its heat transfer performance, Institute for Thermo-Fluid Dynamics, Hamburg University of Technology, Hamburg, Germany.

[7] T.A. Shedd, A.G. Pautsch (2005), Spray impingement cooling with single- and multiple-nozzle arrays. Part II: Visualization and empirical models, International Journal of Heat and Mass Transfer 48 3176–3184.

[8] John R. Guarino, Vincent P. Manno, Characterization Of Laminar Jet Impingement Cooling In Portable Computer Applications, Tufts University, Medford.

[9] Koichi Ichimiya and Koji Tsukamoto (2010), Heat Transfer Characteristics of a Swirling Laminar Impinging Jet, Journal of Heat Transfer, ASME.

[10] Z.X. Yuan, Y.Y. Chen (2006), Swirling effect of jet impingement on heat transfer from a flat surface to CO2 stream, Science direct Experimental Thermal and Fluid Science 31 55–60.

[11] Smith Eiamsa-ard and Pongjet Promvonge (2006), A Numerical Study of Gas Jets in Confined Swirling Air Flow, Chiang Mai Journal of Science; 33(3) : 255 – 270.

[12] A. Ianiro G. Cardone (2010), IR wall heat transfer in swirling impinging jets International Conference on Quantitative Infra Red Thermography, Canada.

[13] C. Kinsella, B. Donnelly (2008), Heat transfer enhancement from a horizontal surface by impinging swirl jets, European Thermal-Sciences Conference, The Netherlands.

[14] Tadhg S. O Donovan, Darina B. Murray (2007), Jet impingement heat transfer – Part I: Mean and root-mean-square heat transfer and velocity distributions, Science direct -International Journal of Heat and Mass Transfer 50 3291–3301.

[15] Tadhg S. O Donovan, Darina B. Murray (2007), Jet impingement heat transfer – Part II: A temporal investigation of heat transfer and local fluid velocities, Science direct International Journal of Heat and Mass Transfer 50 3302–3314.

[16] John C. Duda, Francis D. Lagor (2008), A flow visualization study of the development of vortex structures in a round jet impinging on a flat plate and a cylindrical pedestal, Science direct, Experimental Thermal and Fluid Science 32 1754–1758.

[17] D. H. Lee and Y. S. Chung (2007) investigated, Jet Impingement Cooling of Chips Equipped with Multiple Cylindrical Pedestal Fins, ASME, Journal of Electronic Packaging.