Analysis of Cylindrical Vertical Jet Impingement Heat Transfer On Flat Plate Using CFD

By

K.Siva Satya Mohan[1], S.K.Bhatti[2]

[1]Asst Professor, Department of Mechanical Engineering, GRIET, Bachupally, Hyderabad-500090.

[2] Professor, College of Engineering, Andhra University, Visakhapatnam, India.

Abstract

Impinging jets provide an effective and flexible way to transfer energy or mass in industrial applications. A directed liquid or gaseous flow released against a surface can efficiently transfer large amounts of thermal energy or mass between the surface and the fluid. A numerical simulation and experimental investigation was performed to analyse the heat transfer performance of a hot fluid in a cylindrical impinging jet on a flat surface. The tests were realized for the following ranges of the governing parameters: the jet diameter is 12mm and the distance of Veritical jet to horizontal plate surface was set to be 6 to 18mm. Three different cases with velocities are considered in this analysis. They are H/D = 0.5,1 and 1.5. Reynolds numbers considered are 6000, 8000 and 10000. Fluid of air is considered in this analysis. The plate is considered to be stationery. Horizontal Jet with convergent nozzle is considered in this analysis. Finally, we found out the heat flux generated on the plate.

Keywords: Jet Impingement, Heat transfer, CFD, q , h_w

1.Introduction:

Impinging jets provide an effective and flexible way to transfer energy or mass in industrial applications. A directed liquid or gaseous flow released against a surface can efficiently transfer large amounts of thermal energy or mass between the surface and the fluid.We seek to understand the flow field and mechanisms of impinging jets with the goal of identifying preferred methods of predicting jet performance. Impinging jets provide an effective and flexible way to transfer energy or mass in industrial applications. A directed liquid or gaseous flow released against a surface can efficiently transfer large amounts of thermal energy or mass between the surface and the fluid. Heat transfer applications include cooling of stock material during material forming processes, heat treatment, cooling of electronic components, heating of optical surfaces for defogging, cooling of turbine components, cooling of critical machinery structures, and many other industrial processes. Experimental data for the rate of heat transfer from impinging turbulent jets with nozzle exit Reynolds numbers in the range of 5000 to1,24,000 have been collated and critically reviewed from the considerable body of literature available on the subject. The geometry considered is that of a single circular jet impinging orthogonally onto a plane surface for nozzle to plate distance from 1.2 to 16 and calculated local heat transfer coefficient by k Jambunathan, etal [1]. Liquid crystals are used to investigate the effect of high relative curvature on surface heat transfer for a round air jet impinging perpendicularly on a semi cylindrical convex surface. The relative curvature, (*d/D*), is varied by changing the jet tube diameter for the same surface diameter. Relative curvature varies from 0.18–0.38. The effects of relative curvature, Reynolds number and jet exit-to-surface spacing are described by Cristina cornaro [2]. Detailed heat transfer measurements being made for nozzle to surface spacings of 3,6 and 10 nozzle diameters, with some further measurements being made for nozzle-to-surface spacing of 20,30 and 40 explained by M.Rahimietal [3]. Jet impingement cooling of uniformly heated surfaces is investigated analytically and experimentally for stable, unsubmerged, uniform velocity laminar jets in the absence of phase change is explained by X.Liu, J.H.Lienhard V [4] etal. Convective heat transfer to an impinging air jet is known to yield high local and area averaged heat transfer coefficients. Such Jets are of interest in the cooling of electronic components and of turbine blades and in manufacturing processes such as grinding. The tests include Reynolds numbers, Re, from 10000 to 30000, nozzle to impingement surface distance, H/D, from 0.5 to 8 and angle of impingement, from 30 to 90^0explained by T.S.O Donovan, D.B.Murray[5]. Impinging jets have been used to transfer heat in diverse applications, which include the drying of paper, cooling of turbine blades and the cooling of a griding process, cooling of electronic components explained by Hollworth B.R., and Dublin [6]. Extensive measurements of the near-field pressure provide solid support for the hypothesis that a feedback mechanism is responsible for the sudden change observed in the pressure fluctuations at the onset of response. The feedback loop consists of downstream-convected structures and upstream-propagating pressure waves generated by Impingement of the coherent structures on the plate explained by CHIH-MING HO [7]. Jet impingement is one of the intensive cooling methods to cool hot objects in various industrial processes. The fluid flow and heat transfer characteristics of multi air jet array impinging on a flat plate are investigated both experimentally and numerically by Chougule N.K., [8]. The problem of laminar impingement jet flows of nano fluids has been numerically investigated. Results as obtained for water-Al_2O_3 mixture, show an enhancement of heat transfer rate due to the presence of nanoparticles in the base fluid explained by S.M.Hosseinalipour [9].Spray Cooling is a technique for achieving large heat fluxes at low surface temperatures by impinging a liquid in droplet form on heated surface. Heat is removed by droplets spreading across the surface, thus removing heat by evaporation and by an increase in the convective heat transfer coefficient explained by Christian David Martinez [10].The best cooling performance of five different radiators with different pin fin wave distance explained by Sagot [11]. Comparison of Circular jets wit slot jets wit different nozzle shapes have been explained by [12]. Effect of nozzle geometry on thermal carecterics both experimentally and computationally by [13]. Thermo fluidic behaviour of jets impinging upon an oscillation surface is explained in [14]. Convective heat transfer coefficient for a steady and pulsed impinging jet is explained by[15]. experimental Investigation of effect of te nozzle geometry of square edge and wit chamfered edge on eat transfer characteristics is explained in [16].Experimental study to investigate te effects of separation distance on jet-cylinder impingement is explained in [17]. Influences of various factors on te fluid flow and eat transfer are analyzed both experimentally and simulation is explained in [18]. Understand te eat transfer effects of parameters on cylindrical pedestal is explained in [19]. Numerical modelling of Jet impingement eat transfer is explained in [20]. The main objective of the work is to study the cylindrical jet impingement heat transfer on flat plate using air as fluid. This study is to be compared experimentally with CFD software ANSYS.

2 Methodology

In the present analysis CFD (Creo 2.0, Ansys 14.5) software is used**.** Creois used to design the model of the object. Ansys is used to give the Boundary conditions. cylindrical jet is considered for the analysis. cylindrical Jet and flat plate are drawn in Creo and save the files in .igs format. At the end of jet, kept nozzle for free flow of fluid. Convergent nozzle is taken for the analysis. First Draw the rectangular plate with required dimensions. Then open ansys (fluent) software and open the geometry that will be saved in .igs format. After that mesh the edges and faces separately. The purpose of meshing is for free flow of fluid from jet to plate. After Geometry and meshing go to FLUENT to review results. Three different velocities are considered at inlet. After completion of meshing go to setup edit model and select energy equation and change the option to on and then select the materials to specify the fluids required and apply boundary conditions. Run calculations for 100 iterations and solve the model.

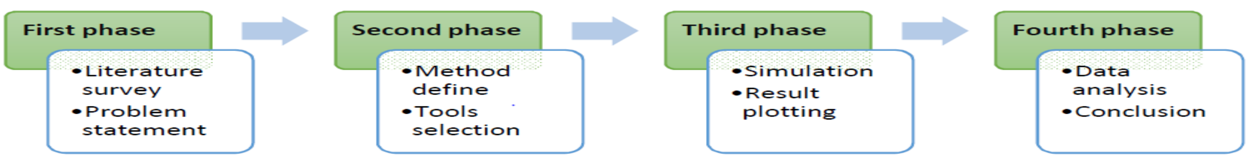


Fig.3.1 METHADOLOGY CHART FOR THE STUDY

Later the tools for solving the model is decided and worked on. After setting suitable schemes, simulation is run till the desired convergence criteria are achieved. There are many tools and techniques using which results can be displayed. These data from the result are categorized and then compared with our basic reference. The result of computer based CFD is then analytically verified.

3. Modelling and Simulation

A model is a simplified representation of a system at some particular point in time or space intended to promote understanding of the real system. A simulation is the manipulation of a model in such a way that it operates on time or space to compress it, thus enabling one to perceive the interactions that would not otherwise be apparent because of their separation in time or space. Modelling and Simulation is a discipline for developing a level of understanding of the interaction of the parts of a system, and of the system as a whole. The level of understanding which may be developed via this discipline is seldom achievable via any other discipline. A simulation generally refers to a computerized version of the model which is run over time to study the implications of the defined interactions. Simulations are generally iterative in there development. One develops a model, simulates it, learns from the simulation, revises the model, and continues the iterations until an adequate level of understanding is developed.The operation of the model can be studied, and, from this, properties concerning the behaviour of the actual system can be inferred.

3.1 EXPERIMENTAL SET UP: The proposed experimental set up is given below

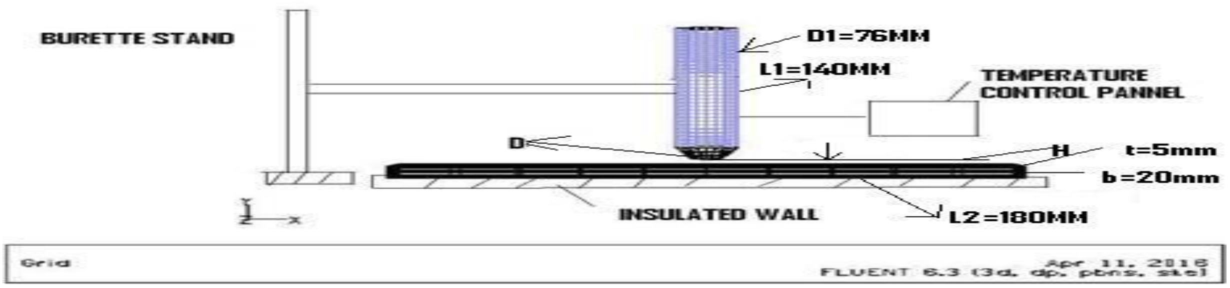


Fig.3.1 Geometry Design of cylindrical jet

3.2 Geometry in CFD

In this research for creating geometry the reference of geometry design of cylindrical jet is taken into consideration of an experimental setup. The grid (mesh) independent study is carried out for every model that is used and made sure that the solution is also independent of the mesh resolution.Cylindrical jet is having 76mm diameter and 140mm height. Convergent nozzle of 12mm diameter is used. Flat plate is considered of length 180mm, width 20mm and thickness of 5mm.

2D sketch **3D modelling**

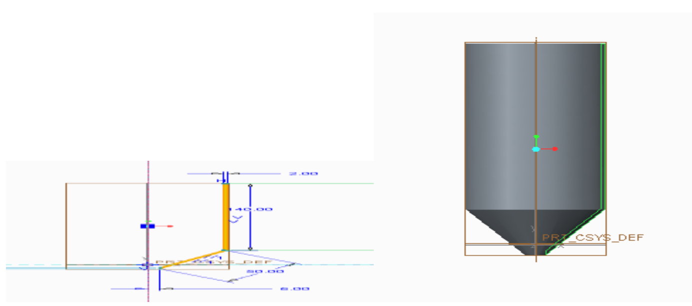


Fig.3.3 Geometry Design of cylindrical jet

4 Meshing

In addition to the automated settings, ANSYS Meshing provides additional control with the option to specify combinations of point controls, edge controls, surface controls and/or body controls. Each one of these has its own options and can be used to influence the mesh in different ways. In this case, the automatic method for mesh shape is selected; however, the sizing for the mesh is selected as fine. It is the meshing section where the sections of the geometry are given name which makes easy while defining domain and giving boundary conditions in the set up section of CFX-PRE

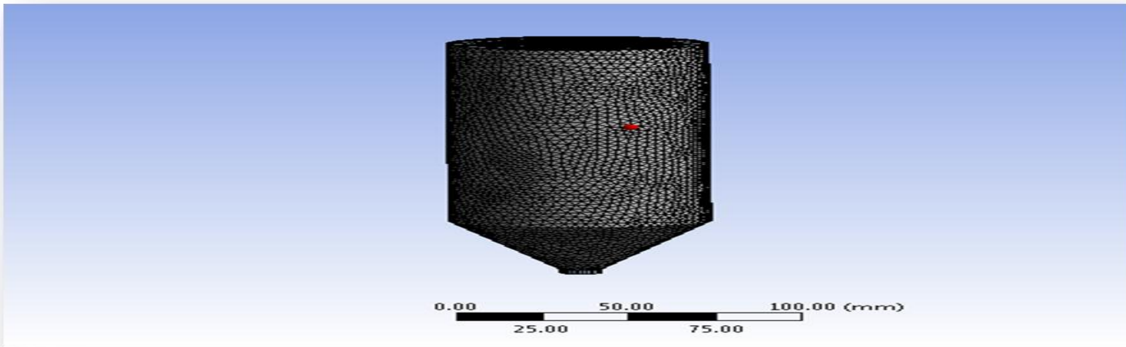


Fig.4.1 Meshing Of Jet in ANSYS CFX (Image 1)

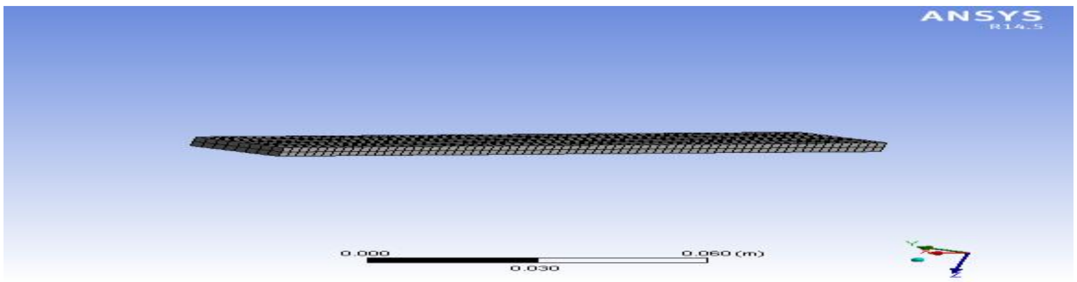


Fig.4.2 Meshing Of plate in ANSYS CFX (Image 2)

5. ANALYSIS SETTING

Steady state analysis is done on CFD. The difference between a steady state simulation and marching a transient solution to steady state is that the steady state simulation ignores many of the cross terms and higher order terms dealing with time. These terms all go to zero in steady state so they don't affect the steady state result. The transient simulation includes all these terms. Usually this means the steady state model has an easier convergence as there are fewer terms to model and some transient non-linearity are removed, but in a few models this non-linearity help convergence (but this is infrequent).

6.1.1 Fluid properties

Fluid	**Density(Kg/m^3)**	**Viscosity(Kg/m-s)**
Air	1.15	20.5735×10^{-6}

6.1.2 Boundary conditions

INLET

V_1	V_2	V_3
1.6842	2.1	2.52

Temperature of fluid - 300K

6.2 Domain Initialization

Initialization in CFX-PRE for the domain is the process by which all unspecified solution field values are assigned at the beginning of a simulation and these values are commonly referred to as initial values. Initialization option for each solved variable to either the Automatic or Automatic with Value needs to be set in CFX-PRE. Only solved (or principal) variables are initialized; other initial field required for other variables is automatically derived from the solved variable initial fields. CFD Analysis on cylindrical jet Using air, nitrogen and carbon dioxide as Working Fluid for steady state simulations, initial values are set automatically in some cases and when good initial guess is not known or is not required.

Material properties of Aluminum 7075: Thermal Conductivity – 173 W/mK, Film coefficient of air – 100W/m^2K

7. Results and Discussion:

7.1 Simulation Results:

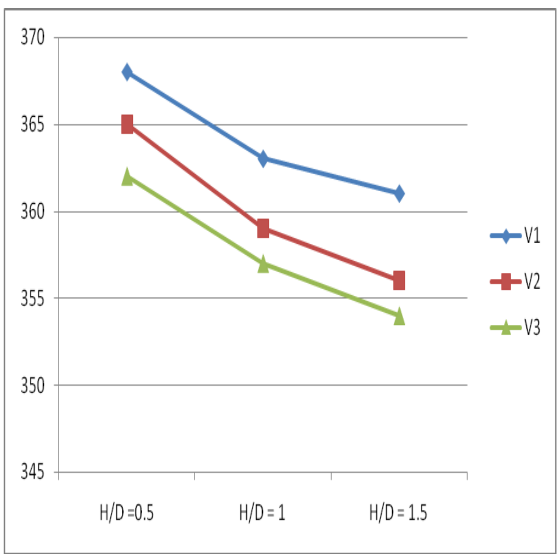
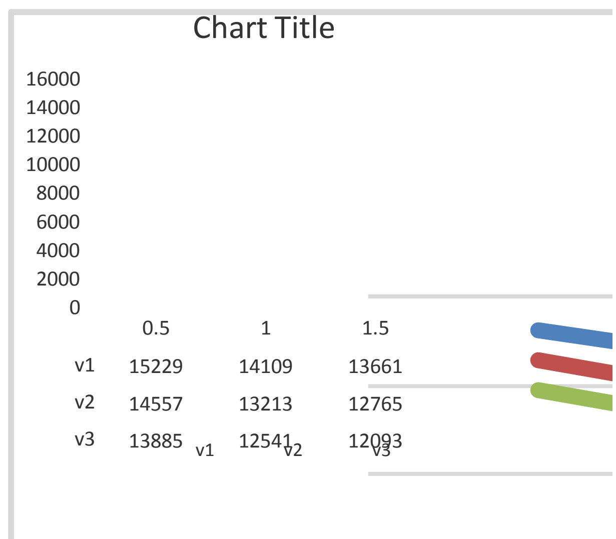


Fig 7.1.1 comparison of temperature for different H/D ratios and velocities for air



	0.5	1	1.5
v1	15229	14109	13661
v2	14557	13213	12765
v3	13885	12541	12093

Fig 7.1.2 comparison of total heat flux for different H/D ratios and velocities for air

From the above simulation results, In Fig.7.1.1 and 7.1.2 as H/D ratio increases Temperature and Heat flux values decreases. Similarly as velocity increases more turbulence is created on the plate implies the decrease in value of Temperature and Heat flux

7.2 Experimental work:

Experimental Investigation has been carried out for different H/D ratios using air and results are calculated using the formulae given below**.** The thermal performance of jet is quantified in terms of total heat flux on plate. Total heat flux depends on heat transfer coefficient and temperature of fluid impinging on plate. Heat flux is calculated by the following equation**,** Heat flux (q) = h(ΔT) = h $(T-T_p)$. Where h = heat transfer coefficient, T = fluid temperature T_p =temperature of plate. Heat transfer coefficient (h) is calculated by using Nusselt number. Heat transfer coefficient (h) = (Nu*K)/D Where K = thermal conductivity of fluid at mean temperature D = diameter of nozzle. Nusselt number (Nu) = $0.023x(Re^{0.8})\ x(Pr^{n})$, Where n = 0.4 for hot fluids, Pr = Prandtl number of fluid at mean temperature Re = Reynolds number = (ρVD)/μ, ρ and μ are fluid properties at mean temperature

7.3 Experimental Results

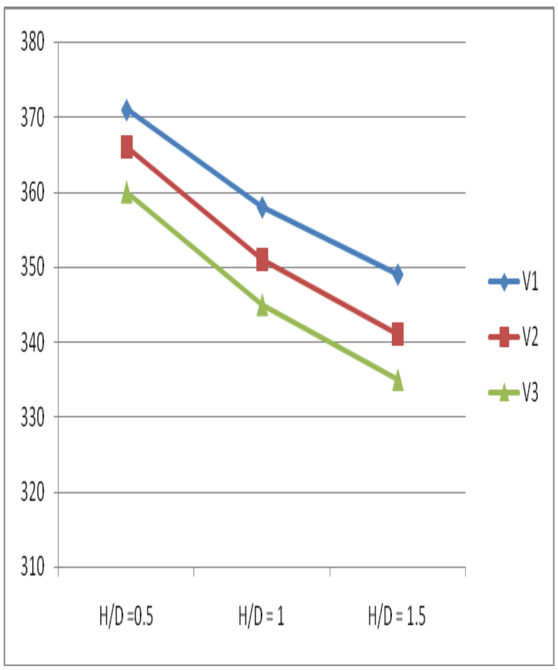
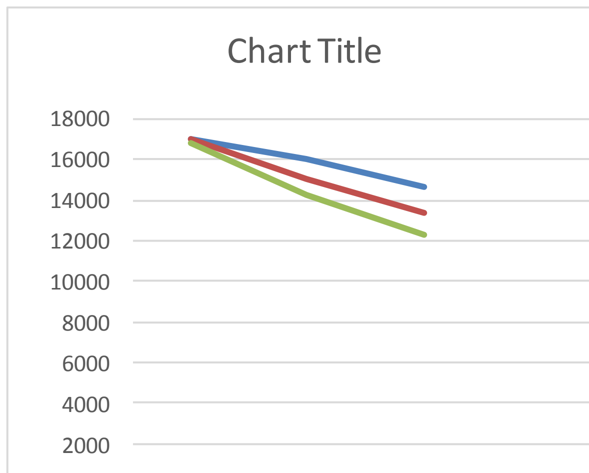


Fig 7.3.1: comparison of Temperatures for air at different H/D ratios and different velocities



	0.5	1	1.5
v1	17064	16042	14657
v2	16990	15028	13396
v3	16856	14281	12320

Fig 7.3.2: comparison of heat flux for air at different H/D ratios and different velocities

From experimental results, from **fig 7.3.1** and **7.3.2** as H/D ratio increases temperature and heat flux values decreases. Similarly as velocity increases temperature and heat flux values decreases.

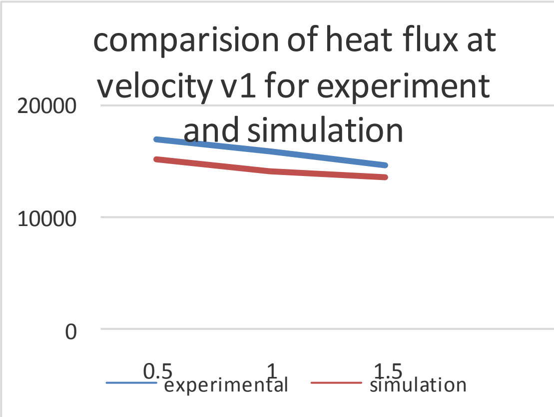


Fig.7.3.3 Heat flux vs H/D ratio at V_1

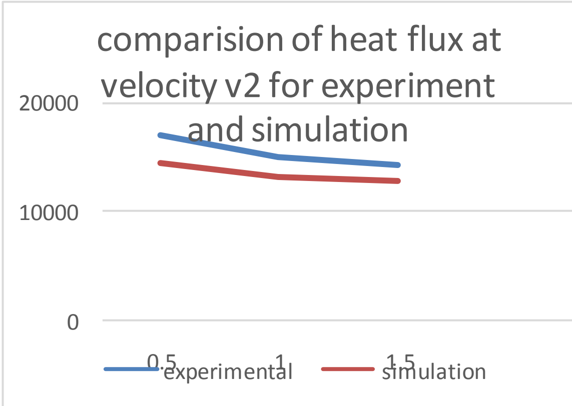


Fig.7.3.4 Heat flux vs H/D ratio at V_2

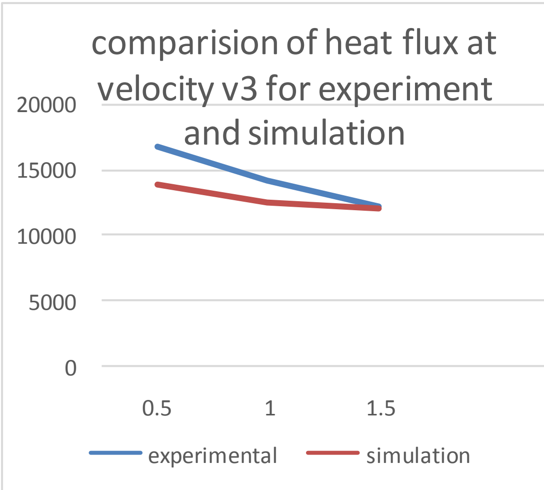


Fig.7.3.5 Heat flux vs H/D ratio at V_3

Fig 7.3.3-7.3.5 shows the comparison of experimental values with Simulation results. Experimental results will give more than simulation values. An error of 10 to 15% occurs between experimental and simulation results.

8. Conclusions and Recommendations

- Impinging jets provide a means of achieving high heat transfer coefficients both locally and on an area averaged basis. The current work forms the first stage of a two-part investigation of heat transfer distributions from a heated flat surface subject to an impinging air jet for Reynolds numbers from 8000, 10000 and 12000 and non-dimensional jet exit to surface spacing, *H*/*D* from 0.5,1and 1.5

- From simulation and experimental results, it is observed that temperature and heat flux are maximum at H/D ratio 0.5.

- From CFD results maximum temperature and heat flux are obtained at Reynolds number of 8000 and H/D ratio of 0.5.

- From thermal analysis Air has obtained maximum heat transfer at Reynolds number of 8000 and H/D ratio of 0.5.

- Finally I conclude air with Reynolds number of 8000 and H/D ratio of 0.5 will give better results than 1 and 1.5
- It is further recommended to increase the inlet velocity of fluid with change in H/D ratios, use of divergent nozzle, use of inclined jet and plate and moving plate.

10. References

1) K Jambunathan, E Lai A review of heat transfer data for single circular jet impingement

2) Cristina cornaro,Michael Rounds Jet impingement cooling of a convex semi-cylindrical Surface

3) M. Rahimi, I Owen, J. Mistry Impingement heat transfer in an under-expanded axisymmetric air jet

4) X. Liu, J.H. Lien hard V, J.S. Lombara Convective heat transfer by Impingement of Circular Liquid Jets

5) T.S. O Donovan, D.B. Murray Effect of Vortices on Jet Impingement Heat Transfer

6) Hollworth B.R. and Dublin Impingement cooling of electronics, ASME Journal of Heat Transfer, 114, pp.607-613.

7) CHIH-MING HO AND NAGY S. NOSSEIR Dynamics of impinging jet. Part 1. The feedback phenomenon

8) Chougule N.K., Parishwad G.V., Gore P.R. Pagnis S., Sapali S.N. CFD Analysis of Multi-Jet Air Impingement on Flat Plate

9) S.M. Hosseinalipour, P.R. Mashaei and K. Esmailpourheat TRANSFER ENHANCEMENT USING NANOFLUIDS IN LAMINAR IMPINGING JET FLOWS

10) Christian David martinezheat Transfer Enhancement of Spray Cooling with Nano fluids

11) B Sagot, G Antonini and F Buron Jet impingement heat transfer on a flat plate at a constant wall temperature

12) Sagar Chirade, Sunil Inole, K.K Sundaram Review of Correlations on Jet Impingement cooling IJSR, 2013

13) Gus Nasif, Ron Barron and Ram Balachandar Jet Impingement heat Transfer: Stationery Disc at IJSMT,2014

14) T natarajan, J.W.Jewkes, R.Narayanaswamy,A.J.King, Y.M.Chung and A.D.Lucey Analysis of Turbulence Statistics and Termofluidic carecteristics under circular jet Impingement

15) Rozli Zulfikli, Ahmed Fadil Ismail, Effect of pulse frequency and pulse shape of single pulsed Air jet Impinement on heat Transfer

16)M.Attalla and M.S.Ahmed Influence the nozzle shape on Local heat Transfer in Impinging Jet
17) Long Jiao and New Tze How Effects of separation distance on the vertical behaviour of jet impingement upon convex cylinders
18) Keyon Cheng, Xiulan Huai, Jun Cai, Zixion guo Numerical simulation of Impinging Cooling on the leading edge of a Turbine blade
19) Moammed Moghiman, Maryam Moein-far, Rasool Elai and Morteza Abdollaian Numerical study of the heat transfer characteristics of a turbulent jet impinging on a cylindrical pedestal
20) N.Zuckerman and Lior Jet Impingement eat transfer: Physics, Correlations and Numerical modelling