IARJSET

ISSN (Online) 2393-8021
ISSN (Print) 2394-1588

International Advanced Research Journal in Science, Engineering and Technology

Vol. 6, Issue 9, September 2019

Fluid Flow and Heat Transfer Analysis of Circular Multi Jets Impact on Concave Surface

C. Chandrakalavathi[1], Y. Uday Kumar[2], M. Murali Mohan Naik[3]

P.G Student, AITS College, Kadapa, AP[1]

Assistant Professor, Mechanical Department, AITS College, Kadapa, AP[2,3]

Abstract: The computational investigation is carried out to study the fluid flow and heat transfer analysis of a circular Multi jets impingement on concave surface. The computational technique simulates the impingement of jet in a gas turbine guide vane. In the present study the effect of different parameters like Jet Reynolds number (Re_d=800, 2000, 4000), inter-jet distance to jet diameter ratio(c/d) and target plate distance to jet diameter ratio (H/d=1,2,3) on flow field and heat transfer are analysed. The analysis is carried out by using ANSYS Fluent 14.5. The maximum pressure and maximum Nu numbers are 85.27 Pa and 54.35 and obtained at H/d=1, c/d=3 and Re_d=4000.

Keywords: Jet impingement, Turbine Guide vane Reynolds number, Nusselt number, Fluent

I.INTRODUCTION

Jet Impingement cooling is the most common method used to cool the leading edge of the gas turbine blade vane. Impingement is usually carried out by either slot or circular jets configured with singular or multiple arrangements. A single circular jet impingement results localized high heat transfer rate , whereas multiple jets provides uniform heat transfer rate over the impinging area.

A review of literature on the various methods of cooling in a gas turbine blade is available in [1]. In general, a flat plate is used as the target surface, reflecting the mid chord of the turbine blade/vane. The Experimental setup for flow visualization and Heat transfer of Jet impingement on flat and curved plate are carried out by Fleischer et al.[2] and Vadiraj V Katti[3]. However, the leading edge portion of the turbine vane, where the impingement cooling is more commonly employed location, is a concave surface. Chupp et al.[4] reported probably the first work on the impingement of a row of circular jets on concave surface. They reported correlations for heat transfer over the leading edge portion of target surface. These correlations are given by

$$Nu_{st} = 0.44 Re_d^{0.7} (d/c)^{0.8} \times \exp\left[-0.85 (H/d)(d/c)(d/D)^{0.4}\right] \qquad (1)$$

$$Nu_{avg} = 0.63 Re_d^{0.7} (d/c)^{0.5} (d/D)^{0.6} \times \exp\left[-12.7 (H/d)(d/c)^{0.5} (d/D)^{1.2}\right] \qquad (2)$$

Equation 1 and 2 are the theoretical calculations for stagnation Nusselt number and average Nusselt number for 4<c/d<16, 1<H/d<10 and 1.5<D/d<16, and 3,000<Red<15,000.C.V Tu and D.H.Wood [5] reported the pressure distribution on a plate beneath the impinging jet is nearly Gaussian, independent on Reynolds number. Fleischer et.al.[2] presented the dynamics of a jet impingement on a convex surface, they gave the vortex initiation and separation through flow visualization technique. Bunker and Metzger[6] used temperature sensitive coatings in their experimental study. While all the investigations are experimental in nature, Numerical investigation on jet impingement heat transfer are available up to 1989 reviewed by Polat et al[7]. They recommended that a semi confined, single turbulent jet can be used as a test configuration for evaluation of newly developed turbulent models.

The heat transfer dynamics of a slot jet impinging of a nonlinear fluid flow on a slightly curved concave surface was presented by Haydar Eren et al.[8].They reported the correlations for slightly curved concave surface are reported for stagnation and average Nusselt number. The mathematical model and computational procedure for impinging jets on concave surface was explained by N.Souris et al.[9] and they also explained about the potential core region as the length of the region where the local velocity is maintained at not less than 95% of the jet velocity at the nozzle exit. Effect of nozzle configuration and curvature on heat transfer characteristics of slot jet impingement was experimentally studied by Geunyoung Yang et al[10]. Round shape, rectangular and 2D contoured nozzles are the different types of nozzles used and they are compared in [10]. Measurement of impinging jet flow and heat transfer on a semi - circular concave surface was done by Mansoo Choi et al. [11] and explained about the secondary peaks and potential length in

DOI 10.17148/IARJSET.2019.6909

impinging jets. Kayansayan and Kucuka [12] studied single two dimensional slot jet with symmetry condition. They presented numerical results for Reynolds number up to 600.Jia et al.[13] and Souris et al. [9] also reported results on slot jet impingement with high Reynolds number of 1700-20000 by using different turbulence models. They recommend k-ϵ model for low computational requirements. Radial slot jet impingement and heat transfer on a cylindrical target with large Reynolds number (80,000), lower curvature with different number of jets was studied by Neil Zuckerman et.al[14], and they resulted that the secondary peak could appear near the stagnation region as well as one in the fountain region for lower curvature. Flow structure of turbulent multiple circular jets impinging on a flat plat was visualized by lamp black and studied by K.Durga Prasad et.al[15]. Array of circular jets impinging on concave surface related to anti-icing system of aircraft wings was recently studied by Frageau et al.[16]. The effect of target plates curvature on the heat transfer in laminar confined impinging jet flows was analyzed by A.M. Tashini et al.[17]. They reported the heat transfer characteristics for low Reynolds number below 500.Effect of Reynolds number on pressure distribution and Nusselt number was reported by B.V.N Ramkumar et al[18]. They recommended K-ω SST turbulent model is better predictive model for impingement studies on concave surface.

II.PROBLEM FORMULATION AND COMPUTATIONAL METHOD

Row of circular jets impinging on a concave surface with three dimensional steady state system is considered in the present investigation. The physical model simulates the impingement cooling of leading edge region of a gas turbine nozzle guide vane. The purpose of the taking circular profile is, it is the best configuration for the characterizing the geometric parameters although the leading edge of the turbine blade is not circular.

2.1 Geometry Details:

The physical model of the present problem (Fig1) is chosen in line with the experimental set-up used by Bunker and Metzger[6]. The target concave plate is heated at required temperature and air at room temperature is used for impingement. The number of jets and the inter jet distance are varied in such a way that the total mass flow rate through these jets remain constant at a particular Reynolds number. Table 1 gives the geometrical dimensions of different jet configurations considered.

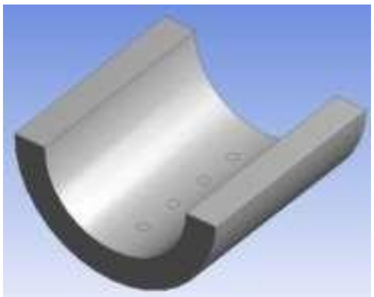


Fig 1: Computational Geometry

Table 1: Geometric Details of Jet Configurations

Dimension/ratio	Values
Diameter of orifice (d)	5 mm
Concave plate diameter (D)	100mm
H/d	1, 2 and 3
c/d	3,4 and 5
Number of jets	4
Length (span) of target plate (L)	125mm

2.2 Numerical simulation:

A three dimensional computational domain is created in the models consisting of circular nozzle , target plate and the flow confinement region, shown in Fig. 2. In the computational domain, jet plenum is not included to reduce the computational cost and to increase the flexibility . The free boundaries of the target plate are extended in order to avoid solution fluctuations such as those noted in [19]. Mass flow condition is imposed at the inlet. The value of mass flow rate is so selected that the corresponding Reynolds number matches laminar, transition and turbulent flows. In other cases, it is varied independently. A turbulence intensity of 5% is chosen as exact value was not available in Ref. [6]. It is so selected that the system used in the experiments is expected to yield low turbulence intensity. Constant heat flux of 658W/m2 is applied to target surface. No slip with adiabatic wall boundary condition is imposed along the solid surfaces of the nozzle. A constant temperature of 296 K is applied to the target surface wall. Constant pressure outlet boundary condition is applied to all open boundaries (side and bottom faces in Fig. 2). Atmospheric pressure of zero gage and 296 K is applied to these outlets. To evaluate the effect of boundary condition of these open boundaries, a case

DOI 10.17148/IARJSET.2019.6909

with wall boundary at 296 K on side faces is also tested. Figure 3 shows the mesh used for a typical case. Boundary layer mesh is used on the concave surface. Unstructured tetrahedral mesh is used for outer region. The values of maximum y^+ used in the present computations are around 1.5.

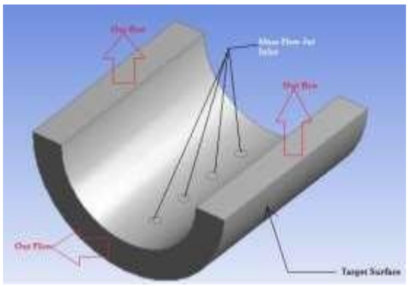


Fig 2: Boundary Conditions

2.3 Parameters investigated:

The parameters investigated in the present study include Reynolds number (Re), jet to target distance (H) and inter jet distance (c). Totally 18 different cases are simulated in the present study. Details of all the cases are given in Table 2.

Table 2: Details of the Different Cases

case	Nozzle diameter (mm)	Reynolds number Re	H/d	c/d	Number of jets
1	5	800	1	3	4
2	5	2000	1	3	4
3	5	4000	1	3	4
4	5	800	2	3	4
5	5	2000	2	3	4
6	5	4000	2	3	4
7	5	800	3	3	4
8	5	2000	3	3	4
9	5	4000	3	3	4
10	5	800	1	4	4
11	5	2000	1	4	4
12	5	4000	1	4	4
13	5	800	2	4	4
14	5	2000	2	4	4
15	5	4000	2	4	4
16	5	800	3	4	4
17	5	2000	3	4	4
18	5	4000	3	4	4

2.4 Mesh sensitivity study

Fig 3 shows the meshed component of the computational domain.

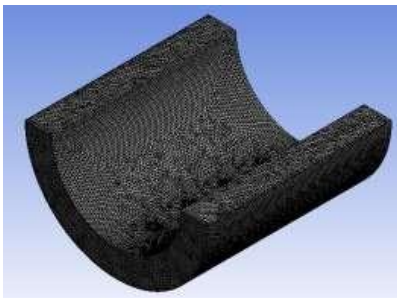


Fig 3: Meshed Geometry

DOI 10.17148/IARJSET.2019.6909

A Grid independence test is carried out to ascertain the accuracy of the numerical results. The mesh sensitivity study is carried out by analyzing the variation of static pressure distribution on the target plate at stagnation point initially, a model with 0.35 million cells is analysed. Figure 4 shows the static pressure distribution at stagnation point with different mesh sizes. It is clear from the figure that the difference in the values of pressure with 0.537 and 0.555 million is not significant.. To get the advantage of computation time, mesh with 0.5537 million is used for this case. Similar mesh sensitivity studies are carried out for all the cases.

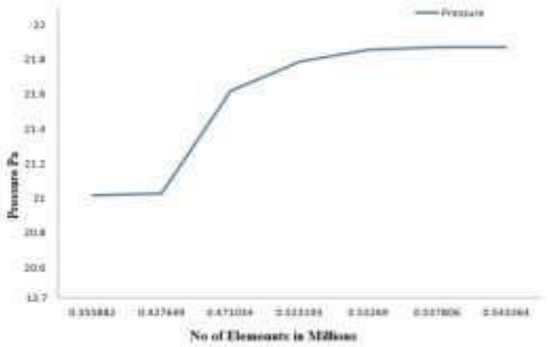


Fig 4 : Grid Independence Test for H/d=1, Re=2000, C/d=4

2.5 Numerical solution

The Present study is simulated in a finite volume based CFD package FLUENT, which is used for solving the governing continuity, momentum, energy and turbulence model equations. Power law scheme is used for Descritization and the well known K-ω SST turbulence models are used as turbulence model as it gives minimum error with experimental data as compared to K-є model as in [18]. SIMPLE algorithm is used for pressure velocity coupling. Flow is considered Incompressible and constant properties are used because of the small variation in temperature and pressure values. The solution is considered to be converged when the sum of the normalized residuals was on the order of 10^{-4} for continuity, momentum, turbulence quantities and 10^{-8} for energy. In some cases it was difficult to get these residual levels. In those cases, the convergence up to 10^{-3} is achieved generally in 1,000 iterations with the use of under relaxation parameters. Any change of relaxation factors to speed up convergence is possible but not desirable at that stage of computation. Therefore, the total surface heat flux is monitored continuously and it is treated that the solution is converged when there was no variation (less than 0.1%) in the monitored flux for 500 consecutive iterations.

III.RESULTS AND DISCUSSION

All calculations are performed on the dimensional form of the governing equations. However, the results are presented in a non-dimensional form, in terms of pressure coefficient and Nusselt number for the sake of generality.

3.1 Effect of Jet Impingement on Heat Transfer:

Fig 5 shows the Velocity and pressure contours of fluid flow with Red=4000, at H/d=3 on a mid section plane of the target surface. It is clearly showing how the velocity(Kinetic energy) is converting into pressure energy. Also it shows the potential core region and formation of secondary fluid where mixing of fluid occurring. The Nusselt number which is related to heat transfer is mainly depends on Re number, H/d ratio, and inter jet to jet spacing.

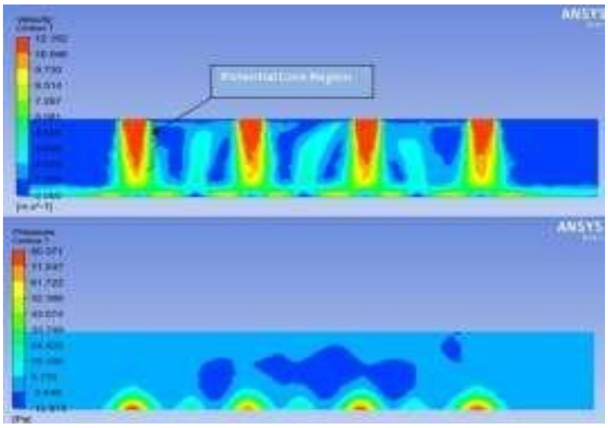
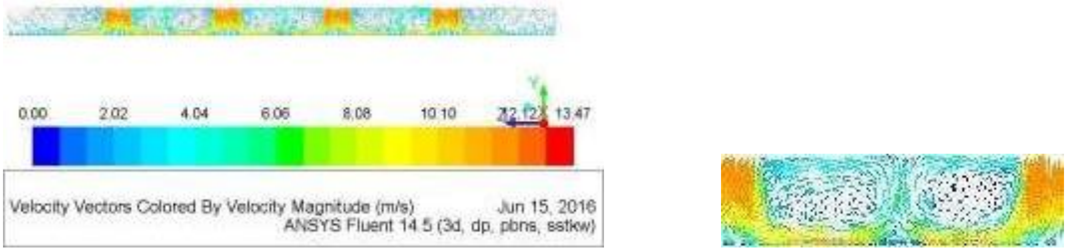


Fig 5: Contours of Velocity and Pressure at H/d=3 and Re_d=4000.

DOI 10.17148/IARJSET.2019.6909



(a) (b)

Fig 6: Velocity Vectors of jet at H/d=1, c/d=4, Re_d=4000 (a) General View, (b) Exaggerated View

The velocity vectors of fluid flow after impingement over the target surface has been shown in Fig 6 . The total velocity of the fluid is converted in to pressure energy at the stagnation point. Due to mixing region between the adjacent two jets and also pressure difference between the stagnation region and free flow region the fluid tries to retain back and that forms vortex region. Due to the vortices the heat will trapped in to the fluid and increases convection and thereby heat transfer.

3.1.1 Effect of Red:

The three types of fluid flows analysed here are Re_d =800, 2000 and 4000, which indicates Laminar, Transition and Turbulent region. Fig 7 shows the Nu number distribution with respect to Z/d at H/d=1, c/d=4 for different Re number. The Nu number will be more for Turbulent flow Re_d=4000 compared to laminar flow. The regions of secondary peaks are observed which represents vortex region of the fluid flow which makes uniform heat transfer on the target surface. Fig 8 shows the Nu number distribution on the target surface for Re_d number=4000. Nu number is distributed along the surface throughout the target surface and is maximum at stagnation point where pressure is maximum.

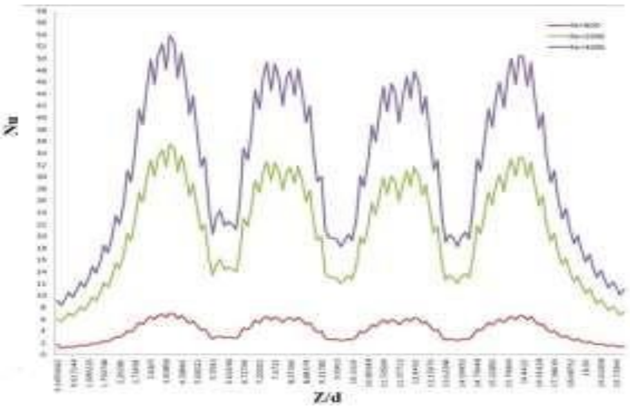


Fig 7: Nu number distribution with respect to Z/d at H/d=1, c/d=4 for different Re

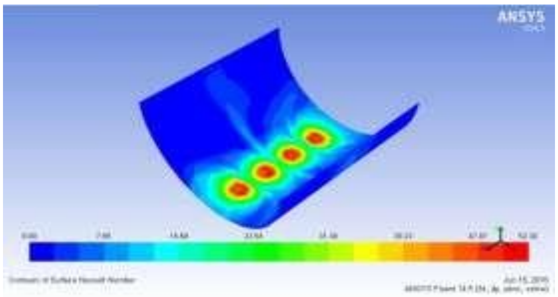


Fig 8: Nu number distribution for Re_d=4000

3.1.2 Effect of H/d:

There is a effect of Jet to target surface distance which is represented as H/d on Nu number. H/d=1, 2 and 3 are investigated for the same Re number and the same interject distance c/d. Fig 9 shows the Nu number distribution of jet at Re_d= 4000 and for different H/d ratios. The effect of H/d on heat transfer is less when compared to effect of Reynolds number. Though the effect is comparatively less, the heat transfer is high for lower H/d compared to higher H/d.

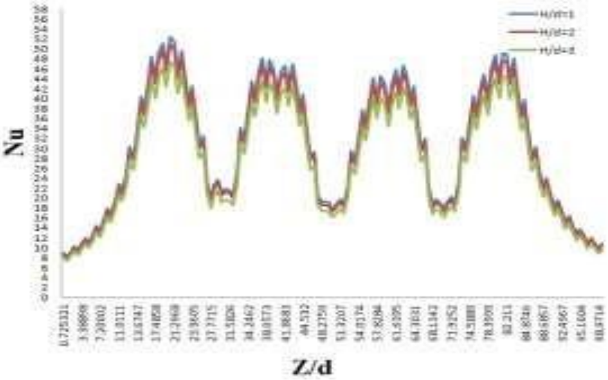
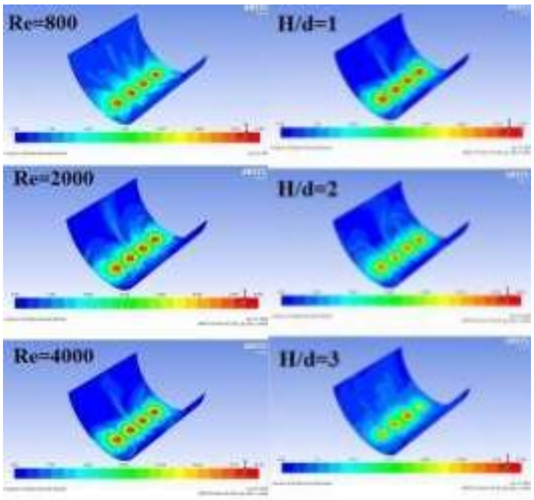


Fig 9: Nu number Distribution for different H/d for Re_d=4000.

But lowering the H/d value may results to reversing the flow in to the jet inlet for higher Re number which effects the total jet flow. Dimensionless parameter Z/d is chosen as abscissa to evaluate and analyse the results accurately.



(a) (b)

Fig 10: (a) Contours of Nu number same H/d and different Re_d number
(b) Contours of Nu number for Same Re_d number and different H/d

Fig 10(a), (b) shows the contours of Nu number for different Re number and different H/d ratios, Here as H/d ratio increases, the stagnation Nu number may decrease but, the uniformity of Nu number on the surface of the target may increase, due to the fluid moves through out plate, than at particular stagnation point.

3.1.3 Effect of c/d:

The effect of inter jet spacing to diameter ratio (c/d) is shown in Fig 11. The jet which have less jet to jet spacing will have more heat transfer characteristics compared to high c/d.

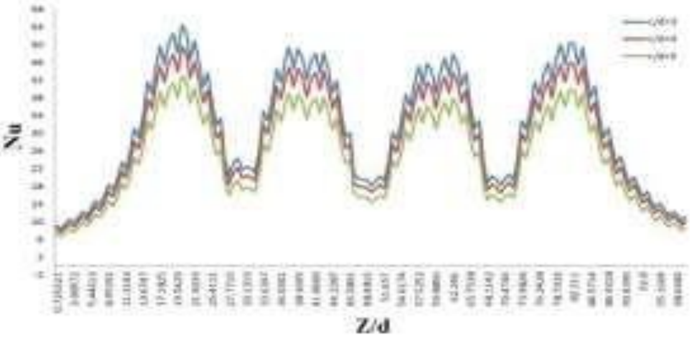
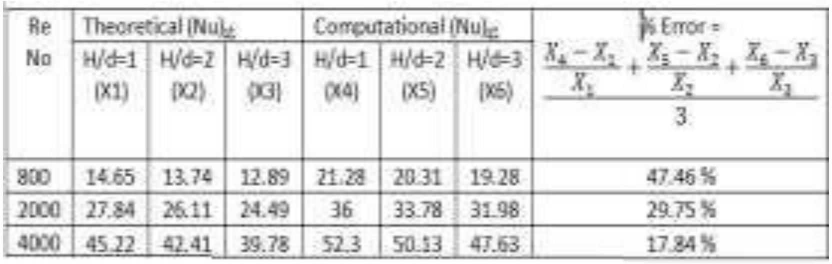


Fig 11: Nu number distribution for H/d=2, Re_d=4000 and for different c/d ratios

3.1.4: Validation of present work:

The present work is validated with theoretical calculations given by [4], and results are matching about 17 % error for c/d=3, Re_d=4000. from the Fig 12, it has been confirmed that the theoretical equations are valid for Re_d>3000 only as resulted from Chupp et al. [4]



Re No	Theoretical $(Nu)_{sc}$			Computational $(Nu)_{sc}$			% Error = $\frac{\frac{X_4-X_1}{X_1}+\frac{X_5-X_2}{X_2}+\frac{X_6-X_3}{X_3}}{3}$
	H/d=1 (X1)	H/d=2 (X2)	H/d=3 (X3)	H/d=1 (X4)	H/d=2 (X5)	H/d=3 (X6)	
800	14.65	13.74	12.89	21.28	20.31	19.28	47.46 %
2000	27.84	26.11	24.49	36	33.78	31.98	29.75 %
4000	45.22	42.41	39.78	52.3	50.13	47.63	17.84 %

Fig 12: Theoretical and Computational error calculations for c/d=3,a and all Reynolds number

IV.CONCLUSION

The following are the conclusions made from the present investigation. The total kinetic energy of the jet is converted to pressure energy. The Pressure on the target plate is maximum at stagnation region where the velocity becomes zero. The lower pressure zones between the mixing regions of the jet forms the vortices which may cause secondary peak pressure regions. The Nusselt number is maximum at stagnation region and also forms secondary peak regions as pressure forms. The maximum pressure and maximum Nu numbers are 85.27 Pa and 54.35 and obtained at H/d=1, c/d=3 and Re_d=4000. The effect of Reynolds number is more when compared to H/d ratio. The Computational results are validated with theoretical results with 17 % error.

REFERENCES

[1]. Chenglong Wang, 2013, "Internal Cooling of Blades and Vanes on Gas Turbine". Heat and Mass Transport

[2]. A.S Fleischer, K.Kramer, R.J. Goldstein, 2001, "Dynamics of The Vortex Structure of A Jet Impinging on A Convex Surface", Experimental and Thermal Fluid Science 24(169-175)

[3]. Vadiraj V.Katti & S.V.Prabhu, 2010, "Local Heat Transfer Distribution from a Row of Impinging Jets to a Cylindrical Concave Surface", Proceedings of the 37th National & 4th International conference on Fluid Mechanics and Fluid Power December 16-18, IIT Madras, India

[4]. Chupp RE, Helms HE, McFadden PW, Brown TR (1969) Evaluation of internal heat-transfer coefficients for impingement cooled turbine airfoils. J Aircraft 6:203–208

[5]. C.V Tu, D.H. Wood, 1996, "Wall Pressure and Shear Stress Measurements Beneath an Impinging Jet", Experimental Thermal and Fluid Science; 13:364-373

[6]. Bunker RS, Metzger DE (1990) Local heat transfer in internally cooled turbine airfoil leading edge regions: part I: impingement cooling without film coolant extraction. ASME J Turbo 12:451–458

[7]. Polat S, Huang B, Majumdar AS, Douglas WJM (1989) "Numerical flow and heat transfer under impinging jets: a review". Annu Rev Numer Fluid Mech Heat Transf 2:157–197

[8]. Haydar Eren, Nevin Celik, Bulent Yesilata, " Nonlinear Flow and Heat Transfer Dynamics of a Slot Jet Impinging on Slightly Curved Concave Surface". International Communications in Heat and Mass Transfer 33(2006) 364-371.

[9]. Souris N, Liakos H, Founti M (2004) Impinging jet cooling on concave surfaces. AIChE J 50(8):1672–1683

[10]. Geunyoung Yang, Mansoo Choi, Joon Sik Lee,1999, " An Experimental Study of Slot Jet Impingment Cooling on Concave Surface: Effects of Nozzle Configuration and Curvature", International Journal of Heat and Mass Transfer 42, 2199-2209.

[11]. Mansoo Choi, Han Seoung Yoo, Geunyoung Yang, Joon Sik Lee, Dong Kee Sohn, 2000, "Measurements of Impinging Jet Flow and Heat Transfer on a Semi-Circular Concave Surface", International Journal of Heat and Mass Transfer 43, 1811-1822

[12]. Kayansayan N, Kucuka S (2001) Impingement cooling of a semi cylindrical concave channel by confined slot-air-jet. Exp Thermal and Fluid Science 25:383–396

[13]. Jia R, Rokni M, Sunden B (2002) Numerical assessment of different turbulence models for jet impinging on flat and concave surfaces. GT-2002-30449, Proceedings of ASME Turbo Expo 2002, Amsterdam, The Netherlands, June 3–6

[14]. Neil Zuckerman and Noam Lior, 2007, " Radial Slot Jet Impingement Flow and Heat Transfer on a Cylindrical Target", Journal of Thermophysics And Heat Transfer, Vol. 21, No.3, July-September .

[15]. K. Durga Prasad, K. Ravi Kumar, M.R.CH.Sastry, 2012," Fluid Flow and Heat Transfer Analysis of Turbulent Multiple Circular Jets Impinging on a Flat Plate",International Journal of Engineering Research & Technology(IJERT) ISSN:2278-0181, Vol.1 Issue 7

[16]. Frageau M, Saeed F, Paraschivoiu (2005) Numerical heat transfer correlation for array of hot-air jets impinging on 3-dimensional concave surface. J Aircraft 42(3):665–670

[17]. A.M. Tahsini and S.Tadayon Mousavi," Laminar Impinging Jet Heat Transfer for Curved Plates", World a Academy of Science, Engineering and Technology, Vol:6 2012-12-26

[18]. B. V. N. Rama Kumar and B. V. S. S. S. Prasad, " Computational Flow and Heat Transfer of a Row of Circular Jets Impinging on a Concave Surface", Heat Mass Transfer DOI 10.1007/s00231-007-0274-3

[19]. Parneix S, Behnia M, Durbin PA (1999) "Predictions of turbulent heat transfer in an axisymmetric jet impinging on a heated pedestal". ASME J Heat Trans 121:43–49.