ISSN 2320-9984

Volume 7 Issue 4 | December 2019 ISSN: 2320-9984 (Online)

International Journal of Modern Engineering & Management Research

Website: www.ijmemr.org

Computational Investigation of Side Edge Closure for Multiple Jets Impinging on a Concave Surface

Anand Bisen

Department of Mechanical Engineering,
Kalaniketan Polytechnic College Jabalpur,
Jabalpur, (M.P.) [INDIA]
Email: anandbisen.knpc@gmail.com

Abstract—*A numerical simulation of the leading edge cooling of gas turbine vane for staggered array of jets impinging on a semi-circular concave surface is investigated. Two configurations of spent air discharge to ambient are considered: a) side-edge open (SEO) and b) side-edge close (SEC). In the present study an effort has been initiated to investigate the effect of side-edge closure on the behaviour of flow and heat transfer characteristics of jets emanating out of impingement orifices and impinging on the hot concave target surface. A computational model with fixed plenum size and varying target surface size to obtain gap ratios of 0.5, 1, 2, 3 and 5 have been used. The parameters varied include: 1) jet Reynolds number (Re) ranging from 2127 to 10,633 and 2) jet-to-target spacing to orifice diameter ratio (h/d) of 0.5, 1, 2, 3 and 5 for a fixed longitudinal pitch to orifice diameter ratio (c/d) of 5.4. The k-e turbulence model with non-equilibrium wall functions is selected based on the close prediction of the model with the experimental results and the maximum deviation being 5%.*

Keywords:— *Reynolds number, Turbulence model, Jet impingement, Gas turbine vane*

1. INRODUCTION

High thermal loading of the leading edge gas turbine vane necessitates most efficient cooling method which can only be offered by jet impingement cooling. The reason behind its prime importance is just simply because of its wide range of applications besides gas turbine blade cooling viz. cooling of electronic components, hot steel plates, tempering glass, drying papers and films and the like. The study and the research on jet impingement cooling are vast, with almost all the studies converge to the point of importance of higher heat transfer coefficient. As the jet impingement cooling involves single or multiple orifices, the discharge coefficient of those orifices also becomes an influencing factor. Previous studies have been done on the subject area, but in isolation, either on an impinging surface (Lbrahim et al. (2005)) or on a film cooled surface (Lin and Shih (2001)). Hollworth and Dangan (1980) and Hollworth and Lehmann (1983) have investigated jet impinging on flat surfaces. Many studies (Lichtrarowicz et al., 1965,Rohde et al.,1969 and Florschuetz et al., 1983) have focused on the importance of such discharge coefficient of orifices for different geometrical and flow configurations. The effect of other parameters such as approach Mach number, pressure drop, inlet edge radius, thickness to diameter ratio, and the angle between the approaching flow and the axis of the orifice gained importance from the studies of Rohde et al. (1969) and Florschuetz et al. (1983) have introduced a concept of cross flow effect on the discharge coefficient of multiple jets impinging on the flat target surface for various h/d ratios.

Cho and Rhee (2001) have investigated the effect of jet Reynolds number, height to diameter ratio (h/d) and arrangement of holes (staggered and inline) in impingement cum effusion system. The major conclusions of their work were: (a) secondary stagnation zone would form a peak in the Nusselt number in the up-wash region, (b) secondary vortices were strengthened and the flow was accelerated owing to the presence of the effusion holes for the staggered hole arrangement resulting in increased heat transfer, (c) the RSM model (with standard wall functions for near wall region) used by them was unable to capture the secondary vortices and the secondary stagnation zone, and (d) the average heat transfer reduced as the h/d increased. Rhee et al. (2003a and 2003b) have studied the influence of cross flow. Xu et al. (2006a and 2006b) have studied the single row of jet impinging with external cross flow on the flat surface with effusion holes on it. Their conclusions were: (a) the local heat transfer near effusion holes increased with increase in Reynolds number, (b) the heat transfer characteristics were more affected by cross flow to jet flow mass flux ratio and least affected by film hole flow to jet flow ratio, and (c) the values of Nusselt number was maximum near film holes with h/d=1.5. From the foregoing surveys, it may be concluded that no work is reported on combined impingement with effusion cooling on concave surface with more than one row of holes each and with different exit configurations.

Similarly Hay and Lampard (1998) and Hay et al. (1994) have reported the decrease in discharge coefficient with increase in angle of orientation, and increase with increase in inlet radiusing and cross flow Mach number for angle of orientation less than 45°. Brignoni et al. reported the effect of changing nozzle geometry on the pressure drop and local heat transfer distribution in confined air jet impingement on a small heat source for different Reynolds number, h/d, chamfer angles and chamfer length. Dano et al. (2005) reported for flat target surface a decline in the discharge coefficient for decrease in h/d due to increase in the cross flow velocity. Li et al. (2006) in their study developed correlations for discharge coefficient of single jet on a flat target surface considering the effect of inlet cross flow and Reynolds number. Royne et al. (2006) made a comparative study on pressure drop and heat transfer of submerged jets for different nozzle geometry and concluded pressure drop for countersunk nozzle to be the highest, sharp-edged nozzle to be the lowest and straight nozzles to be in the intermediate range. Recently Tang and Chang (2007) have studied numerically the effect of h/d on the discharge coefficient of single circular hole for various l/d using k-ε turbulence model with non equilibrium wall function. Also correlation relating discharge coefficient to h/d reported. The major conclusion from the study showed decreasing discharge coefficient for h/d < 0.5 whereas unaffected for larger h/d. Hwang and Liu (1989) have showed increase in the discharge coefficient of about 33% for h/d of 1 and not much influence as h/d increase from 3 to 5. Also reported that decrease in the discharge coefficient for h/d<1 due to the increase in the flow resistance caused by the wall shear stress between the target plate and the test plate.

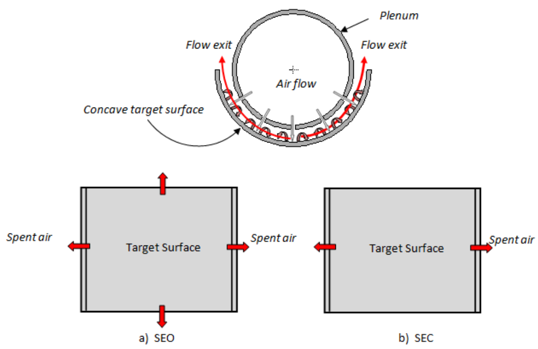

Figure 1: Schematic illustration of spent air discharge configurations

The present study focuses mainly on these two influencing factors of discharge coefficient and heat transfer coefficient for the staggered array of jets under various geometrical arrangements. The concept of side-edge closure which is clearly shown in

Figure 1 is rarely found so far in the open literature for the jets impinging on the concave surface. The side-edge closure in the present case signifies the discharge of air in the circumferential direction only and flow of air in the longitudinal direction is restricted. So the present investigation will contribute largely in terms of the valuable information on the discharge coefficient and the heat transfer coefficient.

2. COMPUTATIONAL METHODOLOGY

The second order discretisation scheme was used for the analysis as it gives accurate prediction of the results. The pressure velocity coupling was performed with the simple algorithm.

3. COMPUTATIONAL DETAILS

3.1 Computational domain and mesh

A three dimensional computational model is build using Gambit, a modelling software used as the pre-processing stage of the CFD analysis. The computational domain used consists of a plenum, target surface and a number of jet holes. The complete specifications of the computational domain are shown in table 1. The proposed model is meshed using gambit with both structured and unstructured mesh whose complete detail is shown in figure 2. For a better prediction of the simulation, a boundary layer mesh is attached to the concave target surface.

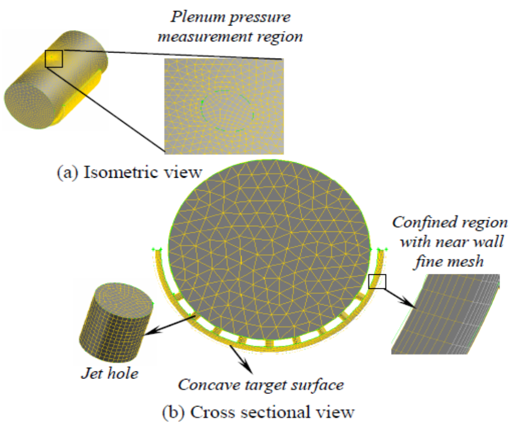

Figure 2: Computational domain with mesh

Table 1: Specifications of test section

Parameters	Symbol	Values
Outer diameter of plenum	D_0	140mm
Thickness of plenum	T	5mm
Length of plenum	L	236mm
Diameter of jet orifice	D	5mm
Longitudinal pitch of hole	c	27mm
No of rows of jets	N_R	9
Angular pitch of hole	θ	15° , 30°
Jet orifice-to-target surface spacing	h	0.5, 1, 2, 3, 5
Diameter of the concave target surface	D	150mm,170mm,190mm

The boundary conditions employed in the present model is detailed in table 2. The mass flow inlet condition is specified at the inlet while the pressure outlet imposed at all the exit conditions of the flow. The input conditions imposed for the two spent air discharge configurations of side-edge open and closed is separately shown in table 3. Where ever the wall condition is specified it is imposed with no slip wall condition. A constant heat flux of 1500 w/m2 condition is specified on the target surface.

Table 2: Boundary condition details

Inlet	Mass flow inlet
Plenum, End wall	Wall
Exit ① ② ③ ④	Pressure outlet
Target surface	Wall
Jet holes	Interior

Table 3: Boundary condition details for SEO & SEC

Case	Pressure outlet condition	Wall condition
Side-edge open	① ② ③ ④	-----
Side-edge closed	③ ④	① ②

3.2 Grid Independence study

The grid independence study is carried out to optimise the size of the mesh used for the analysis so as to reduce the computational time and resource. Various grid sizes of 1.05, 1.43, 2.1 and 2.6 million have been used in the present study as shown in Table 4. Finally a grid size of 2.1million is adapted for further investigation as it ensured the grid independent solution.

Table 4: Grid independence study through Static Pressure Comparison for various mesh sizes

Mesh Size (in millions)	Inlet Plenum Pressure (Pascal)
1.0	261.8845
1.5	252.4383
2.1	244.7386
2.6	240.6234

3.3 Turbulence model and convergence criteria

Figure 3 shows the comparison of various turbulence models for the staggered array of jets. Since the Realizable k-ε turbulence model with the non-equilibrium wall functions as near wall treatment predicts close to the experimental data compared to the other turbulence models used, it is solely adapted for further investigation.

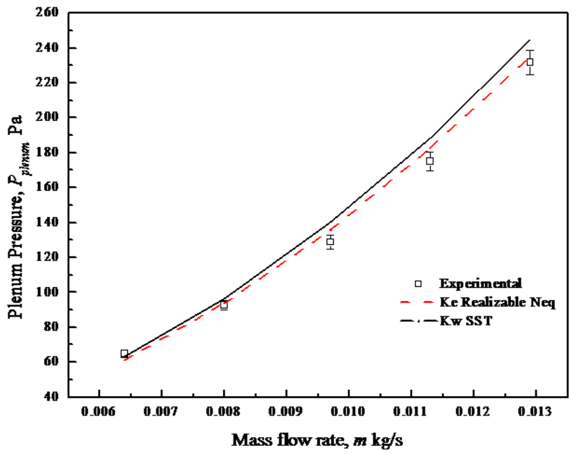

Figure 3 Comparisons of numerical results with experiments for various turbulence models for staggered array of jets

The solution of the problem is regarded as converged when the residuals specified falls below 10^{-6} for continuity, momentum, turbulence model and 10^{-8} for the energy equations respectively. But since the convergence in the present case is not reached, the mass balance and the static wall pressure are monitored. The Iteration is terminated once the net mass balance reaches less than 0.2% and the static pressure monitored remains constant continuously for nearly 500 iterations.

4. EXPERIMENTAL DETAILS

The present experimental study is limited to measurement of discharge coefficient of impingement holes. The schematic layout of such experimental facility is shown in Figure 4. The high pressure air produced by the twin cylinder reciprocating compressor enters the test section through the main supply line by operating the appropriate control valves. The Bourdon tube pressure gauge mounted on the reciprocating compressor indicates the pressure at the exit of the air tank. The air drier and air filter incorporated in the main supply line ensures moisture free and clean air enters the test section. The rotameter with the range of 5 to 60 m^3/hr used gives directly the measure of actual discharge. The control valve 1 is used to control the supply of air from the air receiver to the test section. The control valve

3 adjacent to the rotameter is used to control the actual discharge to the test section. The control valve 2 is used to bypass air, which is the representative of the flow path in the actual gas turbine engine. Digital micro manometer (Furness Controls make) with range of 0 to 1999 mm WC is used to measure the static pressure inside the plenum.

The test section, fabricated out of mild steel, houses plenum, diffuser, end pipe and the flange, where the purpose of the diffuser is to reduce the incoming velocity and increase the pressure. The specification of the test section is the same as shown in table 1. The plenum consists of circular impingement holes with length to diameter ratio (l/d) of one, drilled at its bottom portion wherein the coolant flows through the orifices and impinges on the concave surface. The plenum is designed with staggered arrays of jet orifice. The spent air after impingement exits along the specified directions for the two cases shown in Figure 4.

The smooth concave target plate is made out of aluminum sheet of dimension 165 × 250 × 5 mm. The plate is made into semi-circular shape of diameters, D = 150, 170, and 190 mm respectively in order to obtain the gap ratios of 1, 3, and 5. Two rings were used to close the side-edges thereby confining the flow only in the circumferential direction. A fixture as shown in Figure 4 is designed to support the target plate firmly.

5. EXPERIMENTAL METHODOLOGY

The actual flow rate of air to the test section is controlled and varied from 20-40 m^3/hr by adjusting the required control valves. The value of discharge in the present case is limited to 40 m^3/hr due to fluctuations in the flow at higher discharge. The pressure tap positioned at the centre of the plenum wall measures the static pressure drop across the impingement holes by means of a digital micro manometer. All the pressure measurements are made after the steady state is reached. The proposed experiment is repeated for various jet-to-target spacing obtained by adjusting the target plate fixture. The exit pressure is taken at atmospheric as the air after impingement on the target surface discharges directly into the atmosphere. Air tight gaskets are used to avoid any leakage of air that occurs into the plenum.

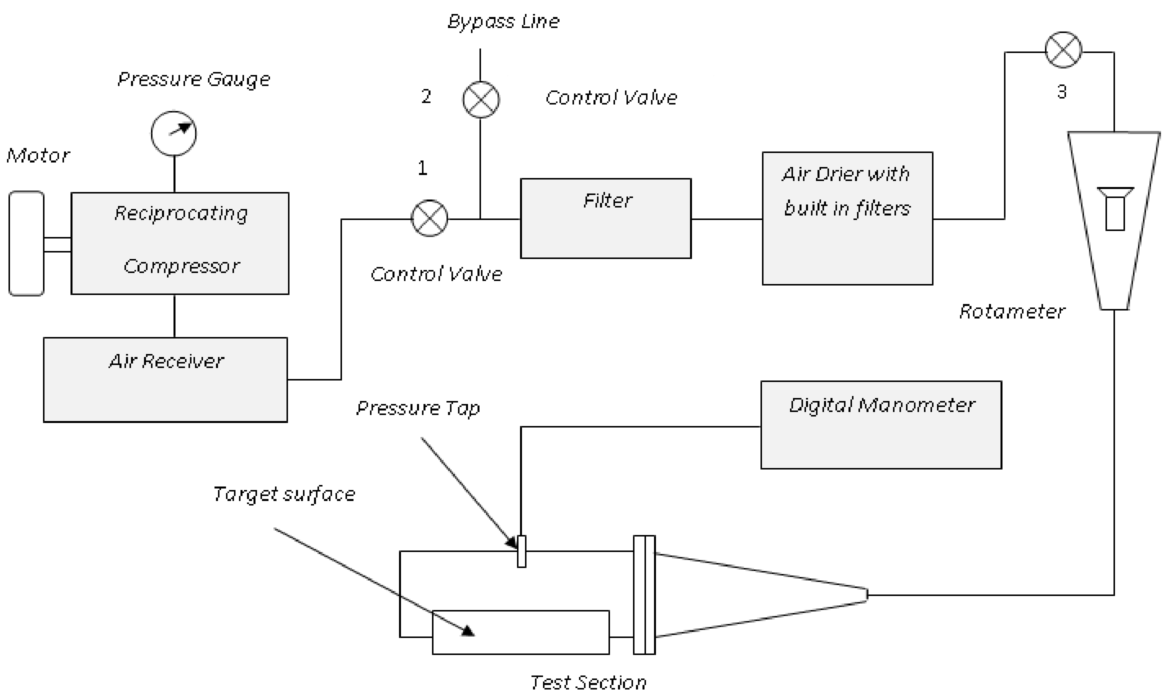

Figure 4: Schematic Layout of the Experimental Setup for Discharge Coefficient Measurement

6. RESULTS AND DISCUSSION

The discharge coefficient of staggered impingement holes for the discharge configuration studied is found to decrease only for h/d 0.5 & 1.0 as evident from the Figure 5. It is also confirmed that Reynolds number has no effect on the discharge coefficient. The percentage deviation of discharge coefficient between side wall open and close is also found to be maximum for h/d 0.5 & 1.0. The selection of turbulence model, a main part of this study is based on the comparison of the numerical results with the experiment as shown in Figure 3. Finally realizable k-e model with the non-equilibrium wall function as the near wall treatment is selected based on the close prediction with the experimental value, the deviation being up to 5%.

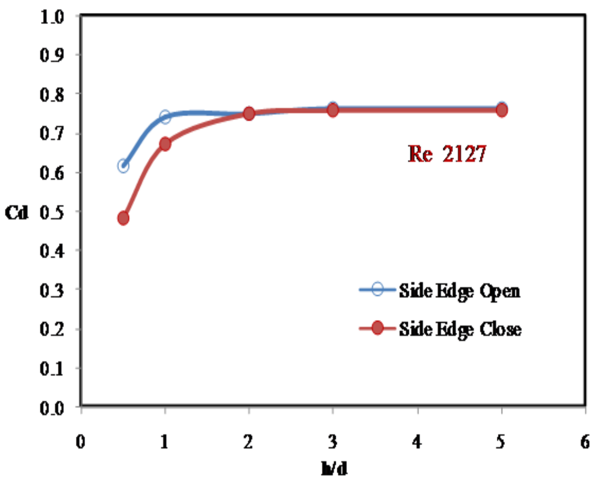

Figure 5: Effect of h/d on C_d for different Re

REFERENCES:

[1] Cho, H. H., Rhee, D. H. (2001). Local Heat/Mass Transfer Measurement on the Effusion Plate in Impingement/Effusion Cooling Systems. Journal of Turbomachinery 123:601–608. Cho, H. H., & Rhee, D. H. (2001).

[2] Dano, B. P. E., Liburdy, J. A., & Kanokjaruvijit, K. (2005). Flow characteristics and heat transfer performances of a semi-confined impinging array of jets: effect of nozzle geometry. International Journal of Heat and Mass Transfer, 48(3-4), 691–701.doi:10.1016/j.ijheatmasstransfer.2004.07.046

[3] Florschuetz, L.W., Y. Isoda, (1983) Flow Distributions and Discharge Coefficient Effects for Jet Array Impingement with Initial Crossflow, ASME Journal of Engineering for Power, 105, 296-304

[4] Hay N., and Lampard D., (1998) Discharge Coefficient of Turbine Cooling Holes: A Review, ASME Journal of Turbomachinery, 120, 314-319

[5] Hay N., Henshall S. E., and Manning A., (1994) Discharge Coefficient of Holes Angled to the Flow Direction, ASME Journal of Turbomachinery, 116, 92-96

[6] Hollworth BR, Dangan L (1980). Arrays of Impinging Jets With Spent Fluid Removal Through Vent Holes on the Target Surface. Part I: Average Heat Transfer. Transactions of the ASME Journal of Engineering for Power 102:994–999.

[7] Hollworth BR, Lehmann G (1983). Arrays of Impinging Jets With Spent Fluid Removal Through Vent Holes on the Target Surface. Part II: Local Heat Transfer. Transactions of the ASME Journal of Engineering for Power 105:393–402.

[8] Hwang, C.J. and Liu, J. L. (1989) Numerical study of two dimensional impinging jet flow fields, AIAA Journal, 27, 841-842 179

[9] Lbrahim, M., Kochuparambil, B. J., Ekkad, S. V., Simon, T. W. (2005). CFD for Jet Impingement Heat Transfer With Single Jets and Arrays. ASME Paper No. GT 2005-68341.

[10] Li, Y., P. Xu, H. Quin, Qi-Hong Deng and J. Wu (2006) Flow bifurcation due to opposing buoyancy in two vertically connected open cavities International Journal of Heat Mass Transfer, 49, 3298-3312

[11] Lichtarowicz A., Duggins R.K., Markland E., (1965) Discharge coefficients for Incompressible Non-Cavitating Flow through Long Orifices, Journal Mechanical Engineering Science, 7, No 2, 210-219

[12] Lin YL, Shih TIP (2001). Film Cooling of a Cylindrical Leading Edge With Injection Through Rows of Compound-Angle Holes. Journal of Heat Transfer 123:645–654.

[13] Rhee, D. H., Choi, J. H., & Cho, H. H. (2003b). Heat (Mass) Transfer on Effusion Plate in Impingement/ Effusion Cooling Systems. Journal of Thermophysics and Heat Transfer, 17 (1), 95–102.doi:10.2514/2.6739

[14] Rhee, D.-H., Yoon, P.-H., & Cho, H. H. (2003a). Local heat/mass transfer and flow characteristics of array impinging jets with effusion holes ejecting spent air. International Journal of Heat and Mass Transfer, 46(6), 1049–1061.doi:10.1016/s0017-9310(02)00363-0

[15] Rohde, J. E., Richards, H. T., Metger, G. W. (1969) "Discharge Coefficients for Thick Plate Orifices with Approach Flow Perpendicular and Inclined to the Orifice Axis," NASA TN D-5467.

[16] Royne, A., & Dey, C. J. (2006). Effect of nozzle geometry on pressure drop and heat transfer in submerged jet arrays. International Journal of Heat and Mass Transfer, 49(3-4), 800–804.doi:10.1016/ j.ijheatmasstransfer.2005.11.014

[17] Tang, C., & Chang, Y. (2007) Computational and experimental study of the orifice discharge coefficient influenced by the presence of a rigid plate, Progress in Computational Fluid Dynamics, 7, No 8, 439-446

[18] Xu, L., Chang, H., & Chang, G. (2006).Local Internal Impingement Heat Transfer Near a 90 Deg Row of Film Cooling Holes: Part 2 — Effect of Jet Row Position. Volume 3: Heat Transfer, Parts A and B.doi:10.1115/ gt2006-90196

[19] Xu, L., Chang, H., & Zhang, J. (2006). Local Internal Impingement Heat Transfer Near a 30 and 90 Deg Row of Film Cooling Holes: Part 1 Effect of Flow Parameters. Volume 3: Heat Transfer, Parts A and B.doi:10.1115/gt2006-90197