Experimental Investigation of Static Pressure Distribution on the Flat Surface Due to Impingement of Air Jets

Ritesh. B. Dhamoji [1], Harish Kumar [2], Raju. B. S [3], Manjunatha L H [4]
[1] PG Student, School of Mechanical Engineering, REVA University, Bangalore.
[2] Assistant Professor, School of Mechanical Engineering, Reva University
[3] Associate professor, school of Mechanical engineering, Reva University
[4] Professor & Head, Mechanical Engineering, REVA ITM/RE

Abstract: The promising technique of impingement cooling of such components needs primary attention as this technique is associated with non-uniform distribution of heat transfer coefficients. The designer sand researchers may choose suitable impingement cooling system based on the permissible non-uniformities of heat transfer rates depending on scheme of application. The present work is to address the issue of non-uniformity. This may be accomplished by obtaining the local heat transfer and fluid flow distributions due to various schemes of jet impingement cooling and there by quantify its degree of non-uniformity.

Hence, present work aims to investigate the local heat transfer and fluid flow characteristics due to impingement on flat surface which simulates leading edge of a typical gas-turbine blade. Further it is aimed to investigate separately the influence of geometric parameters of jets and target surface on local distribution of heat transfer coefficients and wall static pressure distribution. Thin foil and Infrared radiometry technique used by Lytle and Webb [27] will be considered in the present study of local temperature measurement. The uncertainty analysis will be carried out for all the parameter estimation as detailed by Moffat

Keywords: Air jets, Static Pressure distribution, Venturimeter

1. INTRODUCTION

The efficiency of the gas-turbine engines depend primarily on the turbine in hot gas temperatures. The metallurgical considerations of the gas-turbine blade put a limit on the maximum in hot gas temperature. Hence, an option to improve the engine efficiency could be to device an effective method to cool the turbine blades. A promising method of cooling turbine blades is to impinge cool air on the internal surfaces of the blades so that the gas turbine cycle may be operated at higher engine compression ratios with higher inlet gas temperatures for higher efficiencies and reduced fuel consumption. VanTreuren [1] reports a reduction of blade metal temperature of 40°C can improve blade life tenfold. Hanetal.[2] reports that turbine entry temperatures in some of the advanced gas turbines are far higher than the melting point of the blade material hence, the turbine blades need to be efficiently cooled using the relatively cool air bled from the compressor for improved performance.

The jet impingement heat transfer is one of the well-established high performance techniques for heating, cooling and drying of a surface. Such impinging flow devices allow for short flow paths on the surface with relatively high heat transfer rates. Interest and researching this topic continues unabated and may have even accelerated in recent years because of its high potential of local heat transfer enhancements. Applications of the impinging jets include drying of textile sand film; cooling of gas turbine components and the outer wall of combustors; and cooling of electronic equipment.

Single jet finds its application mostly where highly localized heating or cooling is necessary. However, when large surface areas require cooling or heating, multiple jet impingements are desirable. The proposed research work is to focus study on gas turbine blade cooling application which requires multiple jet-impingements.

Hence, this region needs primary attention of efficient cooling method. The internal passage at the leading edge may be considered to have a semi-circular concave surface and this region may be convectively cooled by a span wise row of impinging jets. In order to design and choose an effective cooling method, the knowledge of the local heat
transfer characteristics and wall static pressure distribution of the blade are essential.

2. LITERATURE REVIEW

The high heat transfer rates associated with impinging air jet is well recognized and documented for many years. Review of the experimental work on impinging jets is reported by Living ood and Hrycak [3], Martin [4], Jambunathanet al.[5] and Viskanta[6]. The available literature reveals that there have been some experimental investigations on flow and heat transfer characteristics of semi-circular concave surface with arrow of impinging jets which typically simulate cooling of leading edge of gas turbine blades. One of the first investigations on the impingement of a row of circular jets on a concave surface is reported by Chuppeta[7]. Their configuration simulated cooling internal passages of leading edge of typical gas-turbine blade. Jusios [8], Metzgetetal.[9,10] and Dyban and Mazur[11] have studied the influence of Reynolds number(Re) and other geometric parameters on the average heat transfer characteristics in a semi-circular concave surface impinged by single row of circular jets. Taslimet al. [12-15], Taslimand Khanicheh[16] and Taslimand Bethka[17] made extensive experimental and numerical investigation on off low and heat transfer due to impingement on a smooth and rib-roughened leading-wall of gas turbine for constant z/d and v/d at different Reynolds number. Their study included average heat transfer characteristics for different flow conditions. They reported that the air mass flow rate through all the holes remains almost same for the cases of flow entering the supply channel from one end or both ends. Iacovidesetal.[18], studied experimentally the flow and thermal development of a row of cooling jets impinging on a rotating concave surface. Cooling fluid is injected from a row of five jet holes along the center line of the flat surface of the passage and strikes the concave surface. Craftetal [19] studied modeling of three-dimensional jet array impingement and heat transfer on a concave surface. Fenotetal.[20] carried out experimental investigation of heat transfer due to arrow of air jets impinging on a concave semi-cylindrical surface. The jets are issued from round tubes and flow to the supply channel is normal to the concave surface. However most of the heat transfer studies are not well supported by the wall static pressure measurements on the cylindrical concave surface along both the longitudinal and circumferential directions. Tabakoff and Clevenger[21] studied the effect of surface curvature on wall static pressure distribution for slot jet impingement. They varied the ratio of diameter of flat surface to slot width between 5.0 and 20.0 at constant jet-to-surface distance of six times jet width. They found that, the wall static pressure decrease along the curvature at higher rate at lower ratios of diameter of flat surface to slot width. Florschuetzet al.[22] studied flow distribution characteristics for arrays of impinging jets on flat surface. They developed a theoretical model to predict the row-by-row flow distribution and compared their results with the experiments. Chioietal[23] carried out LD A measurement of mean and fluctuating components of velocity in an experimental study with converging slot jet impinging on a concave surface. Recently, Ramakumarand Prasad [24, 25] reported experimental and computational results of the flow characteristics from multiple circular air jets impinging on a concave surface. They reported experimental results for the configuration of D/d=30, s/d=5.4 and z/d=1.0. However, their study does not explicitly correlate the influence of jet-to-plate distance on wall static pressure at a given Reynolds number. Bunker[26] reports about many and mostly unattended major thermal issues of turbine cooling as advanced engine design has allowed surpassing normal material temperature limits. One of the key issues includes uniformity of internal cooling of turbine blade passages.

3. CONCLUSIONS FROM LITERATURE SURVEY AND OBJECTIVE OF PRESENT PROPOSED WORK:

The promising technique of impingement cooling of such components needs primary attention as this technique is associated with non-uniform distribution of heat transfer coefficients. The designer and researchers may choose suitable impingement cooling system based on the permissible non-uniformities of heat transfer rates depending on scheme of application. The present work is to address the issue of non-uniformity. This may be accomplished by obtaining the local heat transfer and fluid flow distributions due to various schemes of jet impingement cooling and there by quantify its degree of non-uniformity.

Hence, present work aims to investigate the local heat transfer and fluid flow characteristics due to impingement on flat surface which simulates leading edge of a typical gas-turbine blade. Further it is aimed to investigate separately the influence of geometric parameters of jets and target surface on local distribution of heat transfer coefficients and wall static pressure distribution. Thin foil and Infrared radiometry technique used by Lytle and Webb [27] will be considered in the present study of local temperature measurement. The uncertainty analysis will be carried out for all the parameter estimation as detailed by Moffat [28].

4. OBJECTIVES OF THE PRESENT WORK

Based on the literature review the following objectives are defined for the present work. The present work is to study experimentally the distribution of static pressure on the flat surface due to air jet impingement from long pipe circular nozzle.

- Study the influence of Reynolds number of flow on the static pressure distribution on the flat surface. The experiment is conducted for the
Reynolds number of flow ranging from 12000 to 47000.

- Study the influence of longitudinal distance (X) on the static pressure distribution on the flat surface for various Reynolds number of flow and distance between target plate and nozzle (Z).
- Study the influence of longitudinal distance from point of impingement of jet (X) on the static pressure distribution on the flat surface for various Reynolds number of flow and distance between target plate and nozzle (Z).
- Study the influence of distance between target plate and nozzle (Z) on the static pressure distribution on the flat surface for different Reynolds number and along transverse axis (Z).

5. DESIGN OF VENTURI METER

Discharge through a pipe is usually measured by providing co-axial area contraction within the pipe and by recording the pressure drop across the contraction.

A venturimeter is a device consisting of a short length of gradual convergence and a longer length of gradual divergence. Semi angle of convergence is 8° and the semi angle of convergence is 3° to 5°. A pressure tapping is provided at a location before the convergence and another pressure tapping is provided at the throat of the venturimeter. The pressure difference (p1-p2) between the two tappings is measured by means of a U tube manometer. The manometer may contain water or mercury as manometric fluid depending upon the pressure difference is expected. A flow nozzle is a device in which the contraction of area is brought by nozzle. One of the pressure tappings is provided at a distance of one diameter upstream the nozzle plate and other at the nozzle exit.

Air blower AEG GM 600E (600W, 6 Speed, 0-16000rpm)
- Minimum pressure rise at lower speed of blower = 4mm of Hg
- Maximum pressure rise at maximum speed of blower=33mm of Hg

Density of air at 35°C (ρa)

\[ \rho_a = \frac{P}{RT} = \frac{101.325 \times 10^5}{0.287 \times 308} = 1.146 \text{ Kg/m}^3 \quad \text{.....eq. (1)} \]

Pressure head (h_a)

\[ h_a = \frac{\rho_a \cdot V_o^2 \cdot 2}{\rho_a} = \frac{13.6 \times 10^3 \times 4 \times 10^{-3}}{1.146} = 47.47 \quad \text{.....eq.(2)} \]

Velocity of air through Orifice (V_o)

\[ V_o = C_d \sqrt{2 \times g \times h_a} = 0.62 \sqrt{2 \times 9.81 \times 47.47} = 18.92 \text{ m/s} \quad \text{.....eq. (3)} \]

Flow rate (Q_o)

\[ A_o = \frac{\pi}{4} \times d^2, \quad \rho_a = \frac{\pi}{4} \times (20 \times 10^{-5})^2 = 3.14 \times 10^{-4} \text{ m}^2 \quad \text{.....eq. (4)} \]

\[ Q_o = A_o \times V_o = 3.14 \times 10^{-4} \times 18.92 = 0.536 \text{ m}^3/\text{min} \]

Consider,

Minimum flow rate Q_{min} = 0.36 \text{ m}^3/\text{min} 

Maximum flow rate Q_{max} = 2.8 \text{ m}^3/\text{min} \quad \text{(Rated by Manf.)}

5.1. THROAT DIAMETER

Throat diameter is designed to get minimum 50mm deflection in water manometer for Q_{min}

Flow through Venturimeter is given by

\[ Q = \frac{a_1 \cdot a_2}{\sqrt{a_1^2 - a_2^2}} \sqrt{2 \times g \times h_a} \quad \text{.....eq. (5)} \]

For 50mm deflection of water at 35°C

\[ h_a = 43.63 \text{ m of air} \]

\[ a_2 = 1.90 \times 10^{-4} \text{ m}^2 \]

\[ d_2^2 = \frac{a_1}{8} \times a_2 = \frac{5.1 \times 10^{-4}}{8} \times 1.90 \times 10^{-4} \]

\[ d_2 = 0.01556 \text{ m} = 15.56 \text{ mm} \]

Considering d_2 = 16 mm

Inlet diameter d_1 = 25.4 mm

Throat diameter d_2 = 16 mm

5.2. Length of manometer:

For maximum flow rate length of manometer is

\[ h_{max} = \frac{h_a \times a_2}{\frac{d_2}{h_w}} = \frac{2321.146 \times 1.90 \times 10^{-4}}{1000} = 2.65 \text{ m of water} \]

5.3. Design of venturimeter:

For convergent angle \( \theta_1 = 10° \)

\[ \tan \theta_1 = \frac{d_1 - d_2}{2l_1} = \frac{25.4 - 16}{2 \times 10} = 2.8 \quad \text{.....eq. (7)} \]

\[ l_1 = 26.65 \text{ mm} \approx 27 \text{ mm} \]

For divergent angle \( \theta_2 = 5° \)

\[ \tan \theta_2 = \frac{d_2 - d_1}{2l_2} = \frac{25.4 - 16}{2 \times 63} \]

\[ l_2 = 53.721 \text{ mm} \approx 54 \text{ mm} \]

Fig.5.1: 2-D Venturimeter

Venturimeter Dimensions:

- Inlet Diameter: \( d_1 = 25.4 \text{ mm} \)
- Throat Diameter: \( d_2 = 16 \text{ mm} \)
- Convergent Angle: \( \theta_1 = 10° \)
- Divergent Angle: \( \theta_2 = 5° \)
5.4. Calibration of venturi meter

Venturi meter is a device used to measure flow rates. The basic principle on which venturi meter works is that by reducing the cross-sectional area of the passage, a pressure difference is created and measurement of pressure difference ensures the discharge through the pipe. Since the cross-sectional area of throat is smaller than the cross-sectional area of the inlet section, because of which the velocity of flow at throat becomes greater than at the inlet section. The increase in velocity of flow at throat resists decrease in pressure at this section. The pressure difference between these two sections is determined by connecting a differential manometer between the taps provided at these sections; measurement of pressure difference enables the rate of flow to be calculated.

The actual discharge (Q_{act}) is calculated by

\[ Q_{act} = \frac{A_{h}}{x} \]

If "Ao" and "A2" be the cross-sectional areas of the inlet and throat sections respectively and "H" is the difference between pressure head sand "g" acceleration due to gravity then the theoretical discharge (Q_{the}) is given by

\[ Q_{the} = \frac{2g * (C_{v} - 1) - h}{x} \]  

The co-efficient of discharge is given by

\[ C_{d} = \frac{Q_{act}}{Q_{the}} \]  

The diameter of the inlet and outlet are measured and their cross-sectional area is calculated. A differential manometer is connected at the inlet and throat sections and water is allowed to pass through the venturi meter. The dimensions of the collecting tank are noted, the difference between the two limbs of the manometer are noted. The flow is then varied and above procedure is repeated for different Reynolds number. Reynolds numbers are varied from 10000-100000 trials for particular Reynolds number is done and the results are tabulated. Actual and theoretical values of discharge are calculated, and co-efficient of discharge determined by the ratio of actual/theoretical discharge values.

\[ \frac{m_{max}}{m_{min}} = \frac{2.8 + 1.1425}{60} = 0.053316 \text{ kg/s} \]

Reynolds number for minimum flow rate of air

\[ R_{e_{min}} = \frac{\rho \sqrt{Vd_{1}}}{\mu} = \frac{4 \times m_{min}}{\pi d_{1} \mu} = \frac{4 \times 6.8 \times 10^{-3}}{1.983 \times 10^{-5}} = 16846.94 \]  

Reynolds number for maximum flow rate of air

\[ R_{e_{max}} = \frac{4 \times m_{max}}{\pi d_{1} \mu} = \frac{4 \times 0.053316}{1.983 \times 10^{-5}} = 134775.59 \]  

Venturi meter calibration with water at average temperature 30°C

\[ \mu_{w} = 8.315 \times 10^{-4} \text{ kg/m-s} \]

\[ R_{e} = \frac{\rho \sqrt{Vd_{1}}}{\mu_{w}} = \frac{4 \times m_{w}}{\pi d_{1} \mu_{w}} \]

\[ R_{e} = 4 \times \left( \frac{a_{1} + a_{2}}{a_{1} - a_{2}} \right) \left( \frac{\sqrt{2} g h_{w}}{\rho \mu_{w} d_{1}} \right) \]  

\[ \text{ Venturi meter calibration with water at average temperature 30°C} \]

\[ d_{1} = 0.0254 \text{ m} \]

\[ d_{2} = 0.016 \text{ m} \]

\[ a_{1} = \text{Area of pipe} = \frac{\pi d_{1}^{2}}{4} = \frac{\pi \times 0.0254^{2}}{4} = 5.06 \times 10^{-4} \text{ m}^{2} \]

\[ a_{2} = \text{Area of throat} = \frac{\pi d_{2}^{2}}{4} = \frac{\pi \times 0.016^{2}}{4} = 2.016 \times 10^{-4} \text{ m}^{2} \]

\[ R_{e} = 4 \left( \frac{5.607 \times 10^{-4} + 2.016 \times 10^{-4}}{0.0167} \right) \left( \frac{2.91}{1000} \right) \]  

\[ R_{e} = \frac{0.0254 + 0.315 \times 10^{-4}}{1633.33 \text{ mm} + 1000} \]

\[ R_{e} = 21.55 \times 10^{4} \text{ m} \]

For minimum Reynolds number \( R_{e_{min}} = 16846 \)

\[ h_{m} = \left( \frac{16846}{21.55 \times 10^{4}} \right)^{2} = 6.1108 \times 10^{-3} \text{ m} \]

For maximum Reynolds number \( R_{e_{max}} = 134775 \)

\[ h_{m} = \left( \frac{134775}{21.55 \times 10^{4}} \right)^{2} = 0.3911 \text{ m} \]

Venturi meter is calibrated between \( R_{e} \) number 10000-10000

Deflections of mercury in differential manometer are:

- If \( R_{e} = 10000 \)

\[ \sqrt{hm} = \frac{R_{e} \times 21.55 \times 10^{4}}{21.55 \times 10^{4}} \]  

\[ h_{m} = \left( \frac{R_{e}}{21.55 \times 10^{4}} \right)^{2} = \left( \frac{10000}{21.55 \times 10^{4}} \right)^{2} = 2.1533 \times 10^{-3} \text{ m} = 2.1533 \text{ mm} \]

- If \( R_{e} = 12500 \)

\[ h_{m} = \left( \frac{R_{e}}{21.55 \times 10^{4}} \right)^{2} = \left( \frac{12500}{21.55 \times 10^{4}} \right)^{2} = 3.364 \times 10^{-3} \text{ m} = 3.364 \text{ mm} \]

- If \( R_{e} = 15000 \)

\[ h_{m} = \left( \frac{R_{e}}{21.55 \times 10^{4}} \right)^{2} = \left( \frac{15000}{21.55 \times 10^{4}} \right)^{2} = 4.845 \times 10^{-3} \text{ m} = 4.845 \text{ mm} \]

- If \( R_{e} = 17500 \)

\[ h_{m} = \left( \frac{R_{e}}{21.55 \times 10^{4}} \right)^{2} = \left( \frac{17500}{21.55 \times 10^{4}} \right)^{2} = 6.594 \times 10^{-3} \text{ m} = 6.594 \text{ mm} \]
power to the water bath is switched off and temperature of water is allowed to drop. During cooling, mill volt meter readings, temperature readings from the calibrated thermometer, RTD and the K type thermocouple are noted at every 5°C decrease in thermometer reading.

**Fig.5.4: Thermocouple arrangement**

1) Thermos flask 2) Water bath 3) Mill volt meter 4) Tutor
5) Calibrated thermometer 6) K type thermocouple wire 7) Test tube
Experimental set up for calibration of thermocouple. A linear fit is obtained (Fig. 5.4) for the variation of emf with average temperature values from 95°C to 25°C. And the calibration equation is $T(0°C) = 23.188*v + 3.8439$

**Table No 3: Calibration of Thermocouple**

<table>
<thead>
<tr>
<th>Temperature</th>
<th>Voltmeter reading</th>
</tr>
</thead>
<tbody>
<tr>
<td>25.6</td>
<td>0.94</td>
</tr>
<tr>
<td>30.1</td>
<td>1.13</td>
</tr>
<tr>
<td>34.6</td>
<td>1.33</td>
</tr>
<tr>
<td>39.8</td>
<td>1.55</td>
</tr>
<tr>
<td>44.9</td>
<td>1.77</td>
</tr>
<tr>
<td>49.8</td>
<td>1.98</td>
</tr>
<tr>
<td>54.7</td>
<td>2.19</td>
</tr>
<tr>
<td>59.7</td>
<td>2.41</td>
</tr>
<tr>
<td>64.6</td>
<td>2.62</td>
</tr>
<tr>
<td>69.7</td>
<td>2.84</td>
</tr>
<tr>
<td>75</td>
<td>3.07</td>
</tr>
<tr>
<td>79.8</td>
<td>3.28</td>
</tr>
<tr>
<td>84.6</td>
<td>3.48</td>
</tr>
<tr>
<td>89.7</td>
<td>3.7</td>
</tr>
<tr>
<td>94.7</td>
<td>3.92</td>
</tr>
</tbody>
</table>
Pressure above X-X in the left limb = \(1045 \times 9.81 \times H_wN/m^2\)

\[\ldots\text{eq. (16)}\]

Pressure above X-X in the right limb = \(1000 \times 9.81 \times H_wN/m^2\)

\[\ldots\text{eq. (17)}\]

Equating the two pressures, we get

\[1045 \times 9.81 \times H_w = 1000 \times 9.81 \times H_w\]

\[H_w = 1.045 \times H_w(1)\ldots\text{eq. (18)}\]

When pressure head over the surface in C increased by 1mm of water, let the separation level falls by an amount \(Z\). Then \(Y-Y\) becomes the new separation level. Now fall in surface level of C multiplied by Cross sectional area of bulb C must be equal to the fall in separation level multiplied by cross sectional area of the limb.

Therefore,

\[\text{Fall in separation level} \times a = \frac{Z^2}{\frac{7.143 \times 10^{-3}}{2}}\]

Also,

\[\text{Rise in surface level} = \frac{Z}{90.95}\]

The pressure of 1mm of water = \(\rho g h = 1000 \times 9.81 \times 0.001 = 9.81 \frac{N}{m^2}\)

Pressure above \(Y-Y\) in the left limb = \(1000 \times 9.81 \left(\frac{Z}{90.95}\right)\ldots\text{eq. (19)}\)

Pressure above \(Y-Y\) in the right limb = \(1000 \times 9.81 \left[\frac{Z^2}{2} + 9.81\right]\ldots\text{eq. (20)}\)

Equating the two pressures, we get

\[1000 \times 9.81 \left[\frac{Z^2}{2} + 9.81\right] + 9.81 = 1000 \times 9.81 \left[\frac{Z}{90.95}\right] + \frac{Z}{90.95}\ldots\text{eq. (21)}\]

Solving and substituting result 1, we get \(Z = 14.59\)mm

6. EXPERIMENTAL SETUP AND METHODOLOGY

The schematic lay-out of the experimental set-up is shown in Fig. ----. The experimental set up consists of Air blower of 600 watts capacity (Make- AEG GM 600E), 6 speed having minimum and maximum flow rate 0.36 m³/min and 2.8 m³/min. The venturimeter is designed for this flow range and throat and inlet diameters are 16 mm and 25.4 mm. The venturimeter is calibrated with water for the Reynolds No. 10000-100000. \(C_q\) of venturimeter found to be 0.9235±2%

The flow rate is controlled by flow control valves, the Reynolds number is set by adjusting the flow rate with the calibrated venturimeter. The temperature of the air is measured by using k-type thermocouple placed near the nozzle exit. It is calibrated with RTD and the relation between temperature and \(mv\) obtained is \(t=23.188v+3.8439\) with \(R^2 = 1\).

In addition, all experiments were performed under a steady state condition so that accurate temperature data could be obtained.
Acrylic cylinder of inner diameter 50mm and thickness 5mm is used as flat surface and static pressure difference is measured by double bulb, two fluid micro-manometer using Benzyl alcohol and water as manometer fluids, with magnification factor 14.59.

7. EXPERIMENTATION

The experiment carried out for different (Z/D) ratios. The (Z/D) ratio ranging from 0.5 to 4.5

Observation table for (Z/D)=0.5

<table>
<thead>
<tr>
<th>C_p</th>
<th>(X/D)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.5</td>
<td>1</td>
</tr>
<tr>
<td>0.6</td>
<td>2</td>
</tr>
<tr>
<td>0.7</td>
<td>3</td>
</tr>
<tr>
<td>0.8</td>
<td>4</td>
</tr>
</tbody>
</table>

8. RESULTS AND DISCUSSION

An experiment is conducted on the concave flat surface to determine coefficient of static pressure (\(C_p=\Delta p/(0.5\rho AV^2)\)) by impinging air jet by the circular straight nozzle at steady state. Experiments are conducted for different Reynolds number ranging from 12000 to 47000 for various circumferential angles (\(\Theta=0-35^\circ\)), longitudinal distance from point of impingement of jet (X=0 -14 mm), and distance between target plate and nozzle (Z=7.75-124 mm). However the dimensionless distance \(x/d\) and \(Z/d\) are considered for the analysis, where \(d=15.5\)mm, is the diameter of circular straight nozzle.

From the graph of \(C_p\) vs \(X/D\), \(C_p\) decreases as \(Z/D\) ratio increases up to 1. The value of \(C_p\) remains more or less uniform in the range of \(Z/D=1\) to 3. This may be because of target plate located within the potential core of free jet. Then the appreciable decrease of \(C_p\) is observed for further increase of \(Z/D\) ratio.

The effect of Circumferential angle of concave flat surface \(\Theta\) from point of impingement of air jet, the longitudinal distance \(X/D\) and distance of target surface form nozzle \(Z/D\) on Coefficient of static pressure \(C_p\) is experimentally investigated for different Reynolds number of flow at

![Fig.8.1: Influence of longitudinal distance from point of impingement(X/D) on C_p for various Z/D ratio for Re=30000](image)

![Fig.8.2: Influence of longitudinal distance from point of impingement(X/D) on C_p for various Z/D ratio](image)
steady state. The followings are the main conclusions that may be drawn from this study.

- The static pressure distribution on the target surface due to impingement of jet is independent of Reynolds Number of flow.
- The values of static pressure Coefficient $C_p$ at stagnation points are higher due to higher centerline velocities at stagnation.
- The values of static pressure Coefficient $C_p$ on the concave flat surface are almost uniform up to curvature angle of 5°, and decrease appreciably for higher values of $\alpha$.
- The values of static pressure Coefficient $C_p$ on the concave flat surface are higher for lower $X/D$ ratio, they decrease gradually up to $X/D=0.13$, and appreciable decrease of $C_p$ is observed for further increase of $X/D$ ratio.

The potential core of free jet is observed for the $Z/D$ ratio between 1and 3. The velocity decay is minimum for this range of $Z$.

10. REFERENCES