Numerical Investigation of Jet Impingement on a Gas Turbine Blade Leading Edge using CFD

Akshay Raj K.S.
PG Student
Department of Thermal Engineering
JSSATE, Bangalore., VTU Belagavi

Abstract— In this paper, Computational Fluid Dynamics (CFD) of air jet impingement flow characteristics and heat transfer parameters are investigated over a leading edge curvature of a turbine blade. This simulation was performed using RANS method with k-ε model. Analysis is done to study the flow characteristics in impinging process for different impinging height to diameter (h/B), various Reynolds number (Re) for their respective exit velocity of jet (Ue) and for different hydraulic diameter of 2B. The distribution of temperature and temperature fluctuations on the concave surface has been measured. Local Nusselt number has been measured over a curved surface for different hydraulic diameters. Change of jet Reynolds number, distance between the nozzle and target surface and the distance from the stagnation point in the peripheral direction of the curved surface have been considered. Prediction are compared and validation with experiment data. The simulated heat transfer data can be used in many practical experiments.

Key words: Gas Turbine Blade, CFD

I. INTRODUCTION

Gas turbines play a very important role in today’s industry. Also there a necessity of higher power output and increased thermal efficiency. This can be achieved by increasing the temperature of gas entering the turbine. Today’s gas turbines can support up to 1500°C of inlet temperature. As this temperature is higher than metal aero foils, cooling air around 650°C is retrieved from compressor and passed through air foils so that they can withstand these temperature. With this mechanism temperature can be lowered to 1000°C which permits the normal operation.

Turbine blades are exposed to very high temperature and pressure gas produced by combustor. To withstand this environment, turbine blades are made up of exotic materials like super alloys. For cooling purpose different methods like internal air channels, impingement cooling, film cooling, transpiration cooling, thermal barrier coatings are used. Jet impingement is one of the major cooling techniques used in gas turbine leading edge in order to reduce the temperature. If the turbine blades are operate in the same condition for a longer time more thermal stresses will developed over the blade surface this will results in distortion and deformation. In order to avoid this cooling techniques are used. The turbine blades extracts the energy from high temperature, high pressure gas produced by combustor. To withstand this environment, turbine blades will be made up of exotic materials like super alloys. And for cooling purpose different methods like internal air channels, boundary layer cooling and thermal barrier coatings are used.

Single turbine section of gas turbine engine is made up of a disk or hub which holds many turbine blades. Compressor section which can be axial or centrifugal gets connected to turbine section via a shaft or spool. Through the compressor stages of the engine, air is compressed to increase the pressure and the temperature. Combustor which sits between compressor stages and turbine stages increased the temperature greatly by combustion of the fuel. High pressure and temperature exhaust gases when passed through the turbine stages, its energy gets absorbed in the turbine section. Number of turbine stages increases with the bypass ratio. Design of the turbine blades varies with the number of turbine stages. One kind of design is, twin spool design which has a high as well as low pressure spool. One more design has three spools, which has three spools in a row. Low pressure turbine gets exposed low temperature and high pressure turbine to hotter and highest pressure. But the principles of aerodynamic and thermodynamic will not change because of the different conditions, high and low pressure blades are made up of different material.

If temperature prediction of blade differs by only 30°C it can reduce the life of the turbine blade by half. To avoid these early failures there is a necessity of accurate prediction of local heat transfer coefficient and local airfoil metal temperatures. But this is a tedious job for the designers.

The investigation is made for the heat transfer and flow characteristics in jet impingement over a curved surface for various h/B and Reynolds number using one-equation model (OEM) LES and SST-SAS method [1]. Nusselt number comparison clearly shows that one-equation and SST-SAS models are more accurate while defining the heat transfer rate other than RNG k-ε method. In velocity distribution one-equation (OEM) and SST-SAS model outcomes are more reliable and accurate.

Following results are observed from the applied model:
1) RNG k-ε is failed to predict the peak velocity at certain points but one-equation (OEM) LES and SST model successfully defines these locations.
2) RNG k-ε method well defines the maximum velocity but fails to predict the velocities at other locations.
3) one-equation(OEM) performance is better at smaller h/B ratios where the flow field is under very high strain and rapid deformation.

Experimental analysis is carried out for jet impingement technology. The variation in the mean velocity in the different jet regions are measured [2]. Measurements have made for impinging jet flow and heat rate over curved surface.

NOMENCLATURE
h= Distance from target to impinging surface (mm)
B= inlet width (mm)
Ue= jet exit velocity (m/s)
The surface temperature in turbulent flow is carried out for a turbulent jet inlet angle are suitable for swirl cooling at the leading edge of the turbine blade [8]. Numerical simulation is carried out to investigate the thermodynamic parameters by swirl cooling method. The effect of inlet jet angle and tangential jet aspect ratio are considered. The result from the simulation is compared with experimental data. Outcomes predicted that SST k-omega turbulence method is good among all the methods. With increase in the nozzle aspect ratio, peripheral velocity reduces so overall Nusselt number decreases. When the inlet jet angle increases, the peripheral velocity component increases first then it decreases. Maximum swirl takes place at jet angle 90°. The Nusselt number will be less for a jet angle having 60° is highest in all the cases which indicate that there will be lower thermal stresses for 60° jet angle swirl chamber, when Reynolds number or nozzle aspect ratio increases and friction co-efficient decreases. But friction co-efficient increases with increase in jet angle. Increase in the Reynolds number increases the local Nusselt number but reduces with increase in jet aspect ratio. In order to achieve better results jet aspect ratio must be maintained higher than 4.5 and 30° jet inlet angle are suitable for swirl cooling at the leading edge of turbine blade.

The study indicates the comparison between two types of impinging jets at different temperatures is used to heat or cool the surfaces [5]. The application of premixed jet of flame is to increase the temperature of target plate. Here the comparison is made for the heat characteristics between isothermal jet of air and premixed flame of jet. Isothermal air jet is in atmospheric temperature (about 30°C) and premixed flame jet is at 100°C. According to the study if the Reynolds number \( Re \) is kept constant and the associated Nusselt number \( Nu \) for the air jet is not dependent on difference in temperature between surrounding gas and jet itself. The results of the study states that the two jets have almost same heat transfer characteristics.

Findings from the study are:

- For a constant temperature and constant Reynolds number air jet, the Nusselt number is not dependent on temperature.
- If the Reynolds number is constant, the premixed flame jet more with surrounding fluid compared with isothermal air jet, so the isothermal air jet have higher effectiveness.
- If the jet to surface spacing is kept constant, the effectiveness distribution will be same for different Reynolds numbers. Here jet to surface spacing and temperature of the jet are inversely proportional.

Numerical analysis is done over a semi-circular concave surface using the concept of jet impingement [6]. Here the heat flux is kept constant over the surface. The analysis is based on the Navier-Stokes equations conjugated with energy equations which are non-isothermal of steady turbulent flow. Side jet and central jet arrangements are compared with different Reynolds number wall jet distances. Comparison is based on the average Nusselt number and surface temperature for different configurations.

Results explained that the surface temperature in central jet configuration is lower than that of side jet. There is larger Nusselt number at the point of impingement and decreases as move away from the stagnation point. Increases in the jet to wall distance decrease the Nusselt number over the target surface.

A numerical simulation is done to study the local heat transfer rate and flow characteristics in jet impingement method [7]. The position of the jet nozzle and Mach number has been analyzed. The Nusselt number over the surface increases with increase in the impingement Mach number. Nusselt number also increases with decrease in jet to wall distance. Side jet entry considerably increases the local heat transfer rate on the leading edge of the turbine blade. But optimization is required for the nozzle jet position and path shape of the fluid flow passage in order to enhance the cooling rate and reduce the thermal stresses.

The study tells about the effect of coolant swirl at the leading edge of the turbine blade [8]. Numerical simulation is carried out to investigate the thermodynamic parameters by swirl cooling method. The effect of inlet jet angle and tangential jet aspect ratio are considered. The result from the simulation is compared with experimental data. Outcomes predicted that SST k-omega turbulence method is good among all the methods. With increase in the nozzle aspect ratio, peripheral velocity reduces so overall Nusselt number decreases. When the inlet jet angle increases, the peripheral velocity component increases first then it decreases. Maximum swirl takes place at jet angle 90°. The Nusselt number will be less for a jet angle having 60° is highest in all the cases which indicate that there will be lower thermal stresses for 60° jet angle swirl chamber, when Reynolds number or nozzle aspect ratio increases and friction co-efficient decreases. But friction co-efficient increases with increase in jet angle. Increase in the Reynolds number increases the local Nusselt number but reduces with increase in jet aspect ratio. In order to achieve better results jet aspect ratio must be maintained higher than 4.5 and 30° jet inlet angle are suitable for swirl cooling at the leading edge of turbine blade.
II. PHYSICAL AND MATHEMATICAL MODEL

A two dimensional, steady system of turbulent flow air jet impingement cooling analysis is performed for a leading edge of a gas turbine blade. Models are prepared for different values of inlet diameter B and jet to surface distance h. The temperature of the target surface is dependent on the h/B ratio. Reynolds number and jet to surface distances are the major factors which are considered in the heat transfer of a turbine blade leading edge.

A. Geometrical Details

Dimensions for the analysis of turbine blade leading edge are taken from Javad Taghinia et al. [1]. Fig. 1 shows the computational domain. The different parameters are indicated in Table 1.

<table>
<thead>
<tr>
<th>Dimension</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>B (mm)</td>
<td>5, 4, 3</td>
</tr>
<tr>
<td>D (mm)</td>
<td>150</td>
</tr>
<tr>
<td>h (mm)</td>
<td>10, 20, 30, 50</td>
</tr>
</tbody>
</table>

Table 1: Geometrical details

B. Turbulence Model

Simulations are performed using CFD code ANSYS-FLUENT and the results are viewed in CFX 11.0. RANS method with k-ε model is used for simulation. This model is used as it is robust and provides better convergence with moderate accuracy. Finite Volume Method (FVM) is used for discretization. Second order pressure and second order upwind momentum scheme was chosen as it provides more accurate results. High velocity air is used in jet impingement so compressibility factor has to be considered.

C. Mesh Procedure

ANSYS-ICEM software is used to create the unstructured grid for the computational domain. The mesh mainly consists of quadrilateral elements in it. For the investigation of RANS method with k-ε model the y⁺ value must be less than one. In the investigation mathematically calculated y⁺ value is used i.e. 5x10⁻³ and increasing factor of 1.1 is adopted.

C. Boundary Conditions

<table>
<thead>
<tr>
<th>Boundaries</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Type = Velocity inlet</td>
</tr>
<tr>
<td></td>
<td>Velocity magnitude (m/s) = appropriate jet exit velocities</td>
</tr>
<tr>
<td></td>
<td>Specification method = Intensity and length scale</td>
</tr>
<tr>
<td></td>
<td>Turbulence intensity (%) = 4</td>
</tr>
<tr>
<td></td>
<td>Turbulent length scale (m) = 0.000428</td>
</tr>
<tr>
<td>Inlet wall</td>
<td>Type = wall (wall with friction)</td>
</tr>
<tr>
<td>Outlets</td>
<td>Type = Pressure outlet</td>
</tr>
<tr>
<td></td>
<td>Specification method = Intensity and length scale</td>
</tr>
<tr>
<td></td>
<td>Backflow turbulence intensity (%) = 10</td>
</tr>
<tr>
<td></td>
<td>Backflow turbulent length scale (m) = 0.0016</td>
</tr>
<tr>
<td>Target surface</td>
<td>Type = wall (wall with friction)</td>
</tr>
<tr>
<td></td>
<td>Heat flux (w/m²) = 5000</td>
</tr>
<tr>
<td></td>
<td>Wall thickness (m) = 0.00432</td>
</tr>
</tbody>
</table>

Table 2: Boundary Conditions

D. Comparison with Available Experimental Data

The results obtained from the RANS method with k-ε model is validated with Javad Taghinia et al. [1]. Validation proves that the results in the project are correct and reliable.

E. Grid Independence Analyses

Target wall temperature is calculated for different elements and nodes numbers. Increase in the number of nodes considerably increase the computational time. So optimum value of number of nodes are must be used for minimize the computational time. Nodes of 51,120 and elements of 51,675 proves to be the best one as it provides accurate results compared to other cases and any further increase in nodes or elements results in less improvement in accuracy.

III. RESULTS AND DISCUSSION

Nusselt number is inversely proportional to the temperature difference between the inlet air and target surface.

The flow field can be divided in to three regions. They are:
- Free jet region
- Impinging region
- Wall jet region

As the jet to surface distance increases free jet length also increases. Impinging jet region have high pressure field and stagnation point will be formed exactly opposite to the jet exit because of flow deformation. As the air flows away from the stagnation point it loses its kinetic energy and cooling at these regions will be less compared to the region near stagnation point.

As mentioned above the simulations were performed for different cases. And the results obtained are tabulated for different values of Inlet width (B), ratio of distance from jet exit to inlet width (h/B) and Reynolds number and their appropriate velocities.

All rights reserved by www.ijsrd.com

Fig. 3: Flow field regions in the computational domain
Simulation outcomes from RANS k-ε model are listed below:

A. Case (I): For $B = 5\text{mm}$

<table>
<thead>
<tr>
<th>Values</th>
<th>$h/B$</th>
<th>$T_{\text{min}}$ (K)</th>
<th>$Nu_{\text{avg}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re=2960 \ U_c=4.4m/s$</td>
<td>2</td>
<td>341.7</td>
<td>42.98</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>344.1</td>
<td>41.99</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>346.8</td>
<td>39.56</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>360.6</td>
<td>29.46</td>
</tr>
<tr>
<td>$Re=4740 \ U_c=7.1m/s$</td>
<td>2</td>
<td>333.82</td>
<td>54.01</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>334.06</td>
<td>53.62</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>341.99</td>
<td>43.05</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>327.48</td>
<td>67.06</td>
</tr>
<tr>
<td>$Re=6500 \ U_c=9.75m/s$</td>
<td>2</td>
<td>328.17</td>
<td>65.73</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>328.17</td>
<td>65.73</td>
</tr>
<tr>
<td></td>
<td>6</td>
<td>327.18</td>
<td>68.13</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>323.2</td>
<td>57.33</td>
</tr>
<tr>
<td>$Re=8000 \ U_c=12m/s$</td>
<td>2</td>
<td>325.2</td>
<td>73.48</td>
</tr>
<tr>
<td></td>
<td>4</td>
<td>324.04</td>
<td>77.03</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>327.2</td>
<td>68.08</td>
</tr>
</tbody>
</table>

Table 3: Minimum wall temperature for $B = 5\text{mm}$

B. Case (II): For $B = 4\text{mm}$

<table>
<thead>
<tr>
<th>Values</th>
<th>$h/B$</th>
<th>$T_{\text{min}}$ (K)</th>
<th>$Nu_{\text{avg}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re=2960 \ U_c=5.55m/s$</td>
<td>2.5</td>
<td>333.78</td>
<td>43.28</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>336.3</td>
<td>40.81</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>341.6</td>
<td>35.61</td>
</tr>
<tr>
<td></td>
<td>12.5</td>
<td>354.8</td>
<td>27.03</td>
</tr>
<tr>
<td>$Re=4740 \ U_c=8.88m/s$</td>
<td>2.5</td>
<td>325.03</td>
<td>59.18</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>325.98</td>
<td>57.02</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>328.9</td>
<td>51.26</td>
</tr>
<tr>
<td></td>
<td>12.5</td>
<td>337.32</td>
<td>39.69</td>
</tr>
<tr>
<td>$Re=6500 \ U_c=12.18m/s$</td>
<td>2.5</td>
<td>321.1</td>
<td>70.48</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>321.17</td>
<td>69.98</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>322.44</td>
<td>66.01</td>
</tr>
<tr>
<td></td>
<td>12.5</td>
<td>328.76</td>
<td>51.67</td>
</tr>
<tr>
<td>$Re=8000 \ U_c=15m/s$</td>
<td>2.5</td>
<td>318.76</td>
<td>79.48</td>
</tr>
<tr>
<td></td>
<td>5</td>
<td>318.68</td>
<td>79.3</td>
</tr>
<tr>
<td></td>
<td>7.5</td>
<td>319.05</td>
<td>77.76</td>
</tr>
<tr>
<td></td>
<td>12.5</td>
<td>324.13</td>
<td>61.39</td>
</tr>
</tbody>
</table>

Table 4: Minimum wall temperature for $B = 4\text{mm}$

C. Case (III): For $B = 3\text{mm}$

<table>
<thead>
<tr>
<th>Values</th>
<th>$h/B$</th>
<th>$T_{\text{min}}$ (K)</th>
<th>$Nu_{\text{avg}}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$Re=2960 \ U_c=7.4m/s$</td>
<td>3.3</td>
<td>325.76</td>
<td>43.01</td>
</tr>
<tr>
<td></td>
<td>6.66</td>
<td>329.84</td>
<td>37.23</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>336.29</td>
<td>30.61</td>
</tr>
<tr>
<td></td>
<td>16.66</td>
<td>348.4</td>
<td>22.95</td>
</tr>
<tr>
<td>$Re=4740 \ U_c=11.85m/s$</td>
<td>3.3</td>
<td>319.02</td>
<td>58.41</td>
</tr>
<tr>
<td></td>
<td>6.66</td>
<td>320.76</td>
<td>53.52</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>324.79</td>
<td>44.82</td>
</tr>
<tr>
<td></td>
<td>16.66</td>
<td>332.57</td>
<td>34.11</td>
</tr>
<tr>
<td>$Re=6500 \ U_c=16.25m/s$</td>
<td>3.3</td>
<td>315.81</td>
<td>71.05</td>
</tr>
<tr>
<td></td>
<td>6.66</td>
<td>316.08</td>
<td>71.75</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>318.79</td>
<td>61.4</td>
</tr>
<tr>
<td></td>
<td>16.66</td>
<td>324.82</td>
<td>46.48</td>
</tr>
<tr>
<td>$Re=8000 \ U_c=20m/s$</td>
<td>3.3</td>
<td>313.84</td>
<td>81.34</td>
</tr>
<tr>
<td></td>
<td>6.66</td>
<td>313.7</td>
<td>84.22</td>
</tr>
<tr>
<td></td>
<td>10</td>
<td>315.65</td>
<td>73.72</td>
</tr>
<tr>
<td></td>
<td>16.66</td>
<td>320.56</td>
<td>56.12</td>
</tr>
</tbody>
</table>

Table 5: Minimum wall temperature for $B = 3\text{mm}$

Below figure shows the Nusselt number over various points on impinging wall. When $s=0$, Nusselt number will be very high as the air flows away from the stagnation point the value of Nusselt number reduces and reaches the minimum value at the end of the flow.

Fig. 4: Nusselt number over various points on impinging curve for $B=5\text{mm}$

Fig. 5: Nusselt number over various points on impinging curve for $B=4\text{mm}$
Numerical Investigation of Jet Impingement on a Gas Turbine Blade Leading Edge using CFD
(IJSRD/Vol. 4/Issue 05/2016/095)

Fig. 6: Nusselt number over various points on impinging curve for \( B = 3\text{mm} \)

Average velocity of coolant at various points on center position of the jet for different values for \( B \) is represented in following figure. The velocity is normalized using jet exit velocity \( U_e \) and \( h/B \) is normalized using jet to target wall distance \( (h/B) \) in the center line direction. When the \( h \) value increases there is also increase in the boundary and thickness of the shear layer at wall jet region. At lower jet to surface distances \( (h/B) \), entrainment of the fluid occurs close to the jet, while at higher jet to wall distances this entrainment diffuses in the entire domain.

Fig. 7: Average velocity of coolant at various points on center position of the jet for \( B = 5\text{mm} \)

Fig. 8: Average velocity of coolant at various points on center position of the jet for \( B = 4\text{mm} \)

Fig. 9: Average velocity of coolant at various points on center position of the jet for \( B = 3\text{mm} \)

IV. CONCLUSIONS

In the present work, Analysis is done for various models having different inlet width \( (B = 5\text{mm, 4mm, 3mm}) \), jet to surface distance \( (h = 10\text{mm, 20mm, 30mm, 50mm}) \) and jet to surface ratio \( (h/B) \) to find the improved jet impingement geometry in order to enhance the heat transfer rate. The models are tested for different Reynolds number \( (2960, 4740, 6500, \text{and} \ 8000) \) in order to find the optimum value of \( h/B \). By reducing the inlet width considerably increases the jet velocity so cooling rate is maximum. But decreasing the hole diameter increases manufacturing complexity. We have to adopt the Laser drilling mechanism for drilling the small hole which is costlier. So the manufacturer has to select the optimum value of \( h/B \) according to his criteria.

V. FUTURE ENHANCEMENT

Based on the outcomes of the present study the increase in the Reynolds number tends to increase the jet velocity resulting in enhanced cooling. The project also been conducted for corrugated walls and dimpled surfaces in order to get better cooling. Different jet velocities also an important phenomenon in jet impingement so analysis can be done for various Mach numbers in order to get better cooling. In present jet impingement cooling the Mach number varies in the range of 0.1 to 0.3. Higher jet Mach number resulting in increased pressure drop but also yield enhanced heat transfer rate.

REFERENCES


