Optimization of Turbine Blade Cooling by Using Film Cooling Method

Mr. M. Moovendhan[1], Mr. S.Prashanth[2], Mr. G.Velmurugan[3]
[1]*B.E., Department of Aeronautical Engineering, Excel Engineering College*
[2]*M.E, MBA., Assistant Professor, Department of Aeronautical Engineering, Excel Engineering College*
[3]*M.E.,Assistant Professor, Department of Aeronautical Engineering, Excel Engineering College*

***Abstract*- Film cooling is one of the cooling methodology for gas turbine blades from preventing the blade melting from the hot gases out of the combustion chamber. Gas turbines use film cooling in addition to turbulated internal cooling to protect the blades outer surface from hot gases. The present study concentrates on the experimental and numerical investigation of film cooling performance for a row of cylindrical holes in a modern turbine blade. The adiabatic film effectiveness and the heat transfer coefficient are determined experimentally on a flat plate downstream of a row of inclined different geometrical hole exit by using a single test transient IR thermography technique were used nowadays for analysis. But in this project a single turbine stator blade (IGV) was generated with different leading edge thickness with cylindrical holes have to be generated and tested by passing cold and hot air to through the corresponding places in the design and analyzed by using CFX solver in the ANSYS. The focus of this investigation is to investigate the cooling efficiency by changing the leading edge thickness of the turbine stator blade. Here the most heat affecting place in the blades are cooled by effectively changing the size and shape of the hole and other places the holes remains with same size and shape. In this project first step is planned by taking two different shapes of holes in the leading edge in the turbine blade and then the design and analysis was carried out through the CFX solver in ANSYS. Then in the second step, the trailing edge was analyzed by using different slot shape and finally the project will ends with total flow simulation in a turbine blade.**

***Index terms*- turbine blade, film cooling, cooling effectiveness, turbulence modeling**

INTRODUCTION

Increase of gas turbine thermal efficiency can be achieved by increasing the turbine inlet temperature. Despite considerable progress in blade metallurgy, such increase of gas temperature can only be afforded when the blade can be cooled effectively. Film cooling, convection cooling, impingement cooling and combined cooling are some of the most commonly used cooling techniques. Major effort has been devoted in recent years to developing these techniques.

Haslinger and Hennecke (1997) investigated experimentally leading-edge film cooling of a symmetrical model of AGTB turbine blade. They showed that by increasing the blowing rate, the jets penetrate deeper into the main stream resulting in a decrease of the effectiveness. Film cooling of the same blade model, was investigated numerically by Lakehal et al. (2001). They predicted the effect of blowing rates on the flow and temperature fields and successfully compared their results with the available experimental data (Nemdili et al., 2008) and examined the performance of the SSG (Speziale, Sarkar, and Gatski) Reynolds Stress Model for the prediction of film cooling of a symmetrical blade. They found that the RSM model yields reasonably good agreement with measured data at low blowing ratios. Baheri et al. (2008) investigated the effect of mainstream turbulence intensity on the film cooling effectiveness. They reported that sensitivity of film cooling effectiveness to turbulence intensity decreases for trench shaped holes

FILM COOLING

Over the past 5 decades, aircraft gas turbine engine designers have tried to increase the combustion chamber exit and increase the high-pressure turbines inlet temperature. By achieving higher temperatures at combustion chambers exit will result in improved efficiency and reduced fuel consumption. Similarly, the higher temperatures lead to increased thrust in aircraft performance. Unfortunately, due to these higher temperatures the integrity of the high-pressure turbine components will be get damaged and specifically the turbine blades. In modern gas turbine engines inlet temperatures exceed the melting point of the turbine blade materials. To prevent the failure of turbine blades in gas turbine engines resulting from these higher operating temperatures, film cooling has been used in the turbine blade designs. In film cooling, bled air from the compressor stage is ducted to the internal chambers of the turbine blades, and discharged through the holes in the blade walls. This air passed out through the hole provides a thin, cool, insulating layer along the external surface of the turbine blade.

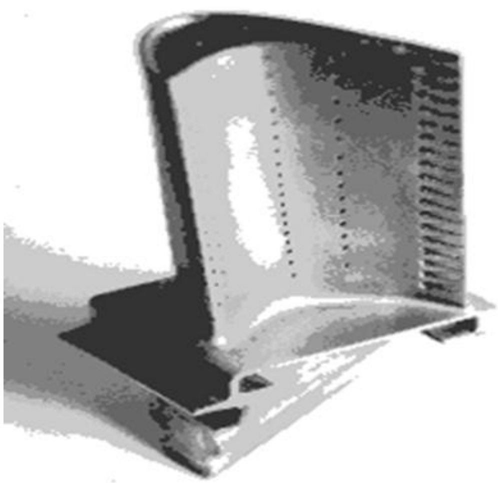

Fig 1. Turbine blade with coolant holes along each side of wall

The first published study of turbine blade tip heat transfer was the investigated by Mayle and Metzger. For normal flat tip models, the tip average heat transfer coefficients were measured with various rotational speeds and Reynolds number. They have found that the tip heat transfer was only a weak function of the rotational speed, and the leakage flow is mainly due to pressure driven, then studied local details of tip heat transfer coefficients for rectangular cavity using moving shroud.

OBJECTIVE OF THE STUDY

The objective of the study is to find the surface adiabatic film cooling effectiveness by,

- Validating the study on film cooling effectiveness.
- Validating the computational data on film cooling by using air and CO2 as coolant.
- Comparing the film cooling effectiveness for both coolants.
- Comparing the counter drilled holes effectiveness of computational and experimental data for different blowing ratios by using air as coolant.
- Comparing the surface wall temperature of the blade from the experimental data with the computational data
- Finding the film cooling effectiveness of the blade and validate the results.

LITERATURE REVIEW

Han, J. C.; Teng, S (2000-01-01) "Effect of Film-Hole Shape on Turbine Blade Film Cooling Performance"Science.gov (United States)

The detailed heat transfer coefficient and film cooling effectiveness distributions as well as tile detailed coolant jet temperature profiles on the suction side of a gas turbine blade are measured using a transient liquid crystal image method and a traversing cold wire and a traversing thermocouple probe, respectively. The blade has only one row of film holes near the gill holes portion on the suction side of the blade. The holes geometries studied include standard cylindrical holes and holes with diffuser shaped exit portion (i.e. fan shaped holes and laidback fan shaped holes). Tests were performed on a five-blade linear cascade in a low-speed wind tunnel. The mainstream Reynolds number based on cascade exit velocity was 5.3 x 10(exp 5). Upstream unsteady wakes were simulated using a spoke-wheel type wake generator. The wake Strouhal number was kept at 0 or 0.1. Coolant blowing ratio was varied from 0.4 to 1.2. Results show that both expanded holes have significantly improved thermal protection over the surface downstream of the ejection location, particularly at high blowing ratios. However, the expanded holes injections induce earlier boundary layer transition to turbulence and enhance heat transfer coefficients at the latter part of the blade suction surface. In general, the unsteady wake tends to reduce film cooling effectiveness.

Barthet, S; Bario, F (2001-05-01)"Experimental investigation of film cooling flow induced by shaped holes on a turbine blade".Science.gov (United States)
The present study is the second half of a piece of work carried out in collaboration with SNECMA. It investigates shaped hole film cooling, numerically and experimentally. The aim of this paper is the experimental analysis of shaped hole film cooling on a large scale turbine blade (1.4 m chord). The test section is a large scale turbine inlet guide vane cascade. The test airfoil is equipped with a row of nine 50 degrees sloped shaped holes. They are located on the suction side at 20% of the curvilinear length of the blade from the stagnation point. The inlet film cooling hole diameter is 12 mm. The jet flow is heated to 55 degrees C above the cross flow temperature. Velocity and temperature field measurements have been done to obtain mean and fluctuating values. The results are compared to those obtained by Béral on the same experimental apparatus and in the same test conditions, for a row of cylindrical holes.
Kim, Y.J.; Kim, S.M.(2004-01-01) [SungKyunKwan Univ., Suwon (Korea). School of Mechanical Engineering "Influence of shaped injection holes on turbine blade leading edge film cooling"
To improve the film cooling performance by shaped injection holes for the turbine blade leading edge region, we have investigated the flow characteristics of the turbine blade leading edge film cooling using five different cylindrical body models with various injection holes, which are a baseline cylindrical hole, two laidback (span wise-diffused) holes, and two tear-drop shaped (span wise- and stream wise-diffused) holes, respectively. Mainstream Reynolds number based on the cylinder diameter was 7.1 x 10{sup 4} and the mainstream turbulence intensities were about 0.2%. The effect of injectant flow rates was studied for various blowing ratios of 0.7, 1.0, 1.3 and 1.7, respectively. The density ratio in the present study is nominally equal to one. Detailed temperature distributions of the cylindrical body surfaces are visualized by means of an infrared thermography (IRT). Results show that the conventional cylindrical holes have poor film cooling performance compared to the shaped holes. Particularly, it can be concluded that the laidback hole (Shape D) provides better film cooling performance than the other holes and the broader region of high effectiveness is formed with fairly uniform distribution.
Kwon, D.G (2001-07-01) [SungKyunKwan University Graduate School, Seoul (Korea); Kim, Y.J. [SungKyunKwan University, Seoul (Korea) ".Effects of various injection hole shapes and injection angles on the characteristics of turbine blade leading edge film cooling"
Using a semi-circled blunt body model, the geometrical effects of injection hole on the turbine blade leading edge film cooling are investigated. The film cooling characteristics of two shaped holes (laterally- and stream wise-diffused holes) and three cylindrical holes with different lateral injection angles, 30 deg., 45 deg., 60 deg., respectively, are compared with those of cylindrical hole with no lateral injection angle experimentally and numerically. Kidney vortices, which decrease the adiabatic film cooling effectiveness, appear on downstream of the cylindrical hole with no lateral injection angle. At downstream of the two shaped holes, the strength of kidney vortices is weaker than that of the cylindrical one. Therefore, two shaped holes have better film cooling characteristics than the cylindrical one. Instead of kidney vortices, single vortex appears on downstream of injection holes with lateral injection angle. But, at downstream of the cylindrical holes with lateral injection angle, the distribution of adiabatic film cooling effectiveness in the lateral direction shows asymmetric nature and high adiabatic film cooling effectiveness regions are more widely distributed than those of the cylindrical hole with no lateral injection angle. As the blowing ratio increases, also, the effects of hole shapes and injection angles increase.
Lu Yiping; Allison, David (2007-10-15) [Mechanical Engineering, Department Louisiana State University, Baton Rouge, LA 70803 (United States); Ekkad, Srinath V. [Mechanical Engineering, Department Louisiana State University, Baton Rouge, LA 70803 (United States)], "Turbine blade showerhead film cooling: Influence of hole angle and shaping"
Detailed film cooling measurements are presented on a turbine blade leading edge model with three rows of showerhead holes. Experiments are run at a mainstream Reynolds number of 19,500 based on cylindrical leading edge diameter. One row of holes is located on the stagnation line and the other two rows are located at 615 deg. on either side of the

stagnation line. The three rows have compound angle holes angled 90 deg. in the flow direction, 30{sup o} along the span wise direction, and the two holes on either side of the stagnation row have and additional angle of 0 deg., 30 deg., and 45 deg. in the transverse direction. The effect of hole shaping of the 30 deg. and 45 deg. holes is also considered. Detailed heat transfer coefficient and film effectiveness measurements are obtained. Results show that, the additional compound angle in the transverse direction for the two rows adjacent to the stagnation row provide significantly higher film effectiveness than the typical leading edge holes with only two angles. Results also show that, the shaping of showerhead holes provides higher film effectiveness than just adding an additional compound angle in the transverse direction and significantly higher effectiveness than the baseline typical leading edge geometry. Heat transfer coefficients are higher as the span wise angle for this study is larger than typical leading edge geometries with an angle of 30 deg. compared to 20 deg. for other studies.

Ahn, Jaeyong; Schobeiri, M.T.; Han, Je-Chin (2007-01-15) [Texas A and M University, College Station, TX (United States). Department of Mechanical Engineering; Moon, Hee-Koo [Solar Turbines Incorporated, San Diego, CA (United States) "Effect of rotation on leading edge region film cooling of a gas turbine blade with three rows of film cooling holes"

Effect of rotation on detailed film cooling effectiveness distributions in the leading edge region of a gas turbine blade with three showerhead rows of radial-angle holes were measured using the Pressure Sensitive Paint (PSP) technique. Tests were conducted on the first-stage rotor blade of a three-stage axial turbine at three rotational speeds. The Reynolds number based on the axial chord length and the exit velocity was 200,000 and the total to exit pressure ratio was 1.12 for the first-stage rotor blade. The corresponding rotor blade inlet and exit Mach number was 0.1 and 0.3, respectively. The film cooling effectiveness distributions were presented along with the discussions on the influences of rotational speed, blowing ratio, and vortices around the leading edge region. Results showed that different rotation speeds significantly change the film cooling traces with the average film cooling effectiveness in the leading edge region increasing with blowing ratio.

DESIGN AND ANALYSIS

The design starts with choosing an appropriate aerofoil for the IGV selection. In this study a patented first section IGV aerofoil has been taken.

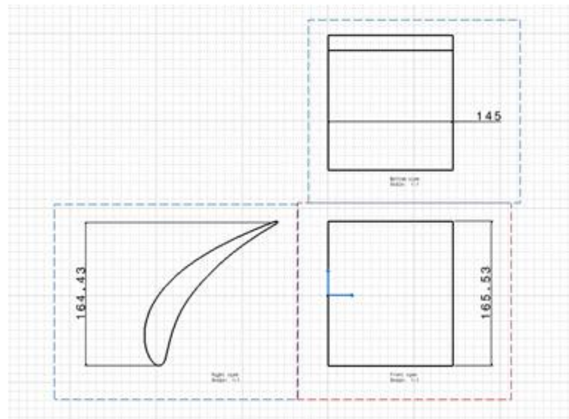

Fig.3.1. Front, top and bottom view of patented aerofoil

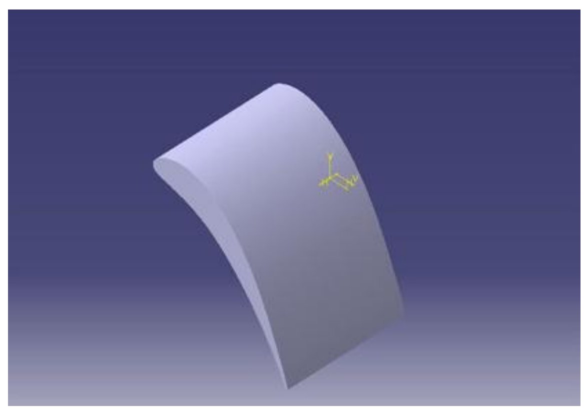

Fig.3.2 isometric view of the patented aero foil

After that hole of circular shapes has been imparted throughout the blades leading edge from the plenum hole shown in fig.3.3

MESHING

Meshing is a discrete representation of the geometry that is involved in the problem. Essentially, it partitions space into elements (or cells or zones) over which the equations can be approximated. Zone boundaries can be free to create computationally best After that the coordinates where imported in CATIA and the IGV design was done and shown in fig.3.1 & 3.2.

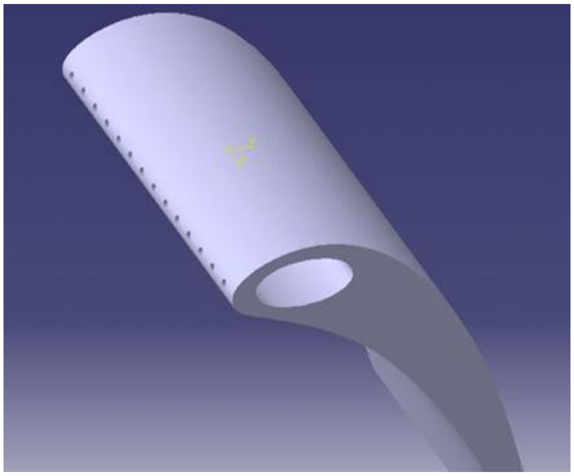

Fig.3.3. patented aero foil with leading holes in wireframe view

FLOW DOMAIN

Once the inlet guide vanes design was done then the domain has to be created for make a flow analysis for the model. The domain was constructed with 7 times of the chord length aft the IGV test section and before it is 5 times of the chord length.

After creating the domain, the IGV one which is slotted with holes are placed at the center or in between the path of two blades of the profile.

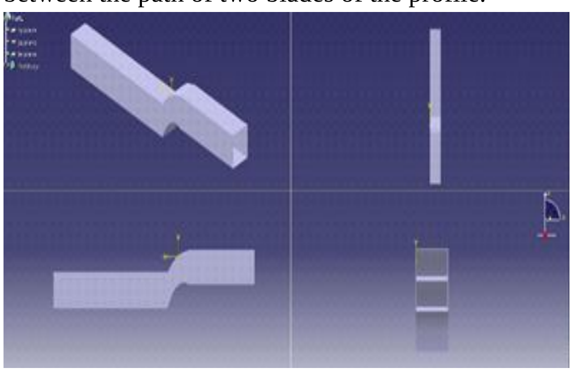

Fig.3.1.1. view of the flow domain shaped zones, or they can be fixed to represent internal or external boundaries within a model.

TYPES OF MESH

Two - dimensional mesh

There are two types of two-dimensional cell shapes that are commonly used. These are the triangle and the quadrilateral.

Triangle:

This cell shape consists of 3 sides and is one of the simplest types of mesh. A triangular surface mesh is always quick and easy to create. It is most common in unstructured grids

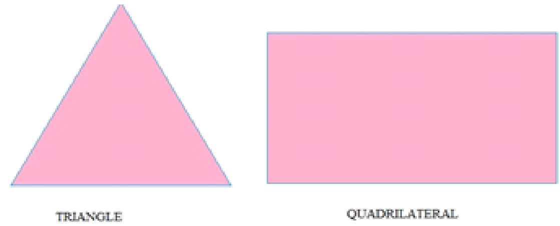

Fig.3.2.1.1. Basic two-dimensional cell shapes

Three – dimensional mesh:

The basic 3-dimensional elements are the tetrahedron, quadrilateral pyramid, triangular prism, and hexahedron. They all have triangular and quadrilateral faces. Extruded 2-dimensional models may be represented entirely by prisms and hexahedra as extruded triangles and quadrilaterals.

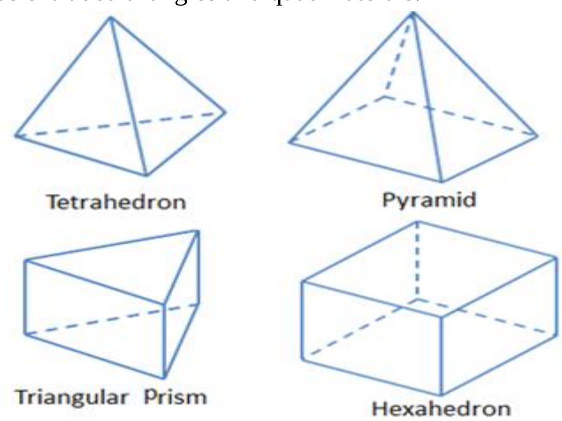

Fig.3.2.1.2.1. Basic Three – Dimensional Cell Shapes

Triangular prism:

A triangular prism has 6 vertices, 9 edges, bounded by 2 triangular and 3 quadrilateral faces. The advantage with this type of layer is that it resolves boundary layer efficiently.

Hexahedron:

A hexahedron, a topological cube, has 8 vertices, 12 edges, bounded by 6 quadrilateral faces. It is also called a hex or brick. For the same cell amount, the accuracy of solutions in hexahedral meshes is the highest.

GRID:

A grid is a small-sized geometrical shape that covers the physical domain, whose objective is to identify the discrete volumes or elements where conservation laws can be applied. Grid generation is the first process involved in computing numerical solutions to the equations that describe a physical process. The result of the solution depends upon the quality of grid. A well-constructed grid can improve the quality of solution whereas, deviations from the numerical solution can be observed with poorly constructed grid. Techniques for creating the cell forms the basis of grid generation. Various methods to generate grids are discussed below.

STRUCTURED GRIDS:

Structured grids are identified by regular connectivity. The possible element choices are quadrilateral in 2D and hexahedra in 3D. This model is highly space efficient, i.e. since the neighborhood relationships are defined by storage arrangement. Some other advantages of structured grid over unstructured are better convergence and higher resolution.

UNSTRUCTURED GRID:

The main importance of this scheme is that it provides a method that will generate the grid automatically. Using this method, grids are segmented into blocks according to the surface of the element and a structure is provided to ensure appropriate connectivity. To interpret the data flow solver is used. When an unstructured scheme is employed, the main interest is to fulfill the demand of the user and a grid generator is used to accomplish this task. The information storage in structured scheme is cell to cell instead of grid to grid and hence the more memory space is needed. Due to random cell location, the solver efficiency in unstructured is less as compared to the structured scheme.

DELAUNAY TRIANGULATION

In mathematics and computational geometry, a Delaunay triangulation for a set P of points in a plane is a triangulation DT(P) such that no point in P is inside the circumcircle of any triangle in DT(P). Delaunay triangulations maximize the minimum angle of all the angles of the triangles in the triangulation; they tend to avoid skinny triangles. The triangulation is named after Boris Delaunay for his work on this topic from 1934.

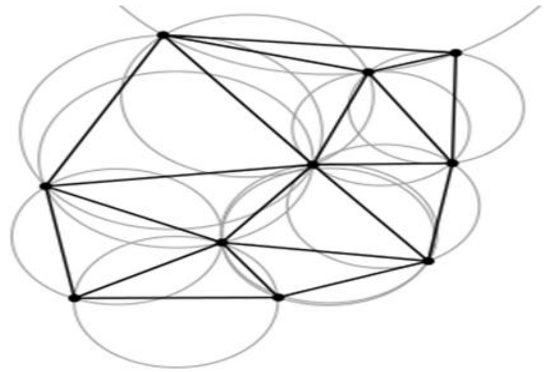

Fig.3.3.3.1. Delaunay triangulation in the plane with circumcircle shown

For a set of points on the same line there is no Delaunay triangulation (the notion of triangulation is degenerate for this case). For four or more points on the same circle (e.g., the vertices of a rectangle) the Delaunay triangulation is not unique: each of the two possible triangulations that split the quadrangle into two triangles satisfies the "Delaunay condition", i.e., the requirement that the circumcircle of all triangles have empty interiors.

By considering circumscribed spheres, the notion of Delaunay triangulation extends to three and higher dimensions. Generalizations are possible to metrics other than Euclidean. However in these cases a Delaunay triangulation is not guaranteed to exist or be unique.

MESH REPORT

The mesh inserted for this domain is unstructured mesh. In unstructured mesh “DELAUNAY TRIANGULATION” method is used to create grids in this domain. The mesh details of this domain are shown in the table 3.4.1.

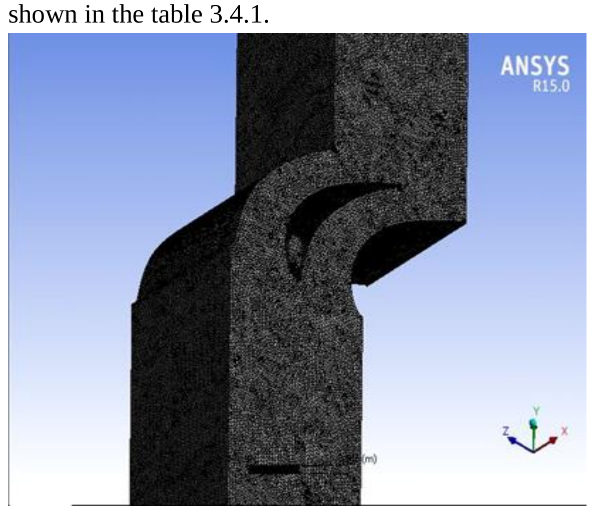

Fig.3.4.1. unstructured mesh for domain

A good mesh leads to a perfect result due correct grid points. If the mesh elemental size is very small and the accuracy of the result is too high. So high density mesh will leads to a proper accuracy or else the accuracy will be more linear.

able 3.4.1. Details of mesh:

Details of "Mesh"	
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Fine
Initial Size Seed	Active Assembly
Smoothing	High
Transition	Fast
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default (1.6551e-004 m)
Max Face Size	Default (1.6551e-002 m)
Max Size	Default (3.3101e-002 m)
Growth Rate	Default (1.850)
Minimum Edge Length	6.9396e-004 m
Inflation	
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes
Advanced	
Defeaturing	
Statistics	
Nodes	4308006
Elements	25124680
Mesh Metric	None

DOMAIN SETUUP & INITIAL CONDITIONS

The domain setup and initial conditions are needed to solve an equation. The computer will takes the input as initial values and convert them into PDE and then it convert into numerical data. The numerical data's iterated form a particular grid element and it is integrated to obtain for the whole grid.

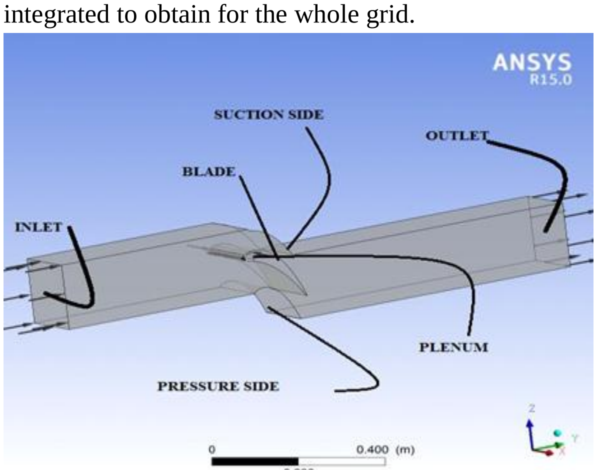

Fig.3.5.1. the domain setup from ANSYS

Table 3.5.1. Boundary condition for inlet

Boundary - inlet	
Type	INLET
Settings	
Flow Regime	Subsonic
Heat Transfer	Total Temperature
Total Temperature	1.7730e+03 [K]
Mass & Momentum	Normal Speed
Normal Speed	1.0000e+02 [m s^-1]
Turbulence	Medium Intensity and Eddy Viscosity Ratio

Table 3.5.2. Boundary condition for plenum

Boundary – plenum	
Type	INLET
Settings	
Flow Regime	Subsonic
Heat Transfer	Total Temperature
Total Temperature	6.0000e+02 [K]
Mass And Momentum	Normal Speed
Normal Speed	1.6000e+02 [m s^-1]
Turbulence	Medium Intensity and Eddy Viscosity Ratio

Table 3.5.3. Boundary condition for outlet

Boundary – outlet	
Type	OUTLET
Settings	
Flow Regime	Subsonic
Mass And Momentum	Average Static Pressure
Pressure Profile Blend	5.0000e-02
Relative Pressure	2.0000e+00 [bar]
Pressure Averaging	Average Over Whole Outlet

Table 3.5.4. Boundary condition for wall

Boundary - WALL	
Type	WALL
Settings	
Heat Transfer	Adiabatic
Mass And Momentum	No Slip Wall
Wall Roughness	Smooth Wall

Table 3.5.5. Boundary condition for Blade

Boundary - blade	
Type	WALL
Settings	
Heat Transfer	Heat Transfer Coefficient
Heat Transfer Coefficient	5.0000e-01 [W m^-2 K^-1]

Outside Temperature	1.7730e+03 [K]
Mass And Momentum	Free Slip Wall
Boundary - wall	
Type	WALL
Settings	
Heat Transfer	Adiabatic
Mass And Momentum	No Slip Wall
Wall Roughness	Smooth Wall

RESULT AND DISCUSSION

This phase will conclude the overall surface adiabatic effectiveness for various blowing ratios. Here were are going to compare the efficiency of two optimized positioned holes efficiency with quad hole configuration for different blowing ratios by using ANSYS CFX solver.

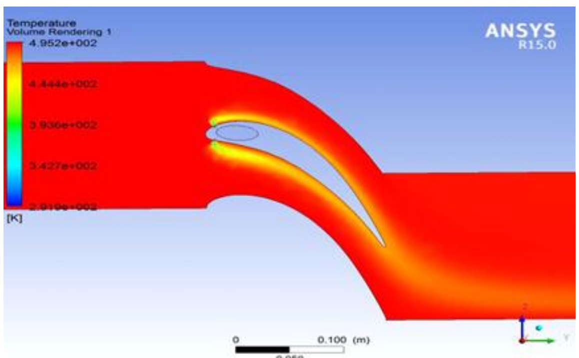

Fig 4.1. temperature volume rendering for 0.5M

At constant blowing ratio with an optimized x/d from the L.E the results shows that the film formation over the blade surface was good with circular holes. But the most efficient of film cooling effectiveness is approximately 6 % more adiabatic effectiveness than other blowing ratios, film around the blade surface that can be seen from the figure 4.1.to 4.2.

After that various blowing ratios are tested from 0.1 to 1.5, the data's are seen from the table 4.1.1. From the table graphs are plotted which was shown in figure 4.1.4. To find blowing ratio the below expression is needed.

$$M = \frac{\rho_c U_c}{\rho_\infty U_\infty}$$

To find effectiveness:

$$\eta = \frac{T_\infty - T_{aw}}{T_\infty - T_{c,exit}}$$

By using this the equations the table 4.1.1. is obtained.

Table 4.1. Computed results for varying blowing ratio's for various hole geometries.

BLOWING RATIO	Effectiveness (η)
0.1	0.079892
0.2	0.087887
0.3	0.088415
0.4	0.089056
0.5	0.089949
0.6	0.092103
0.7	0.096379
0.8	0.100513
0.9	0.10479
1	0.109113
1.1	0.113436
1.2	0.117759
1.3	0.122082
1.4	0.126405
1.5	0.130728

Fig 4.2. Comparison of η Vs. M

Then after that finally a quadrant setup is formed around the leading edge with analyzed x/d for the blade are done and the final analysis for counter dilled shape are completed and shown in fig4.3. to 4.4.

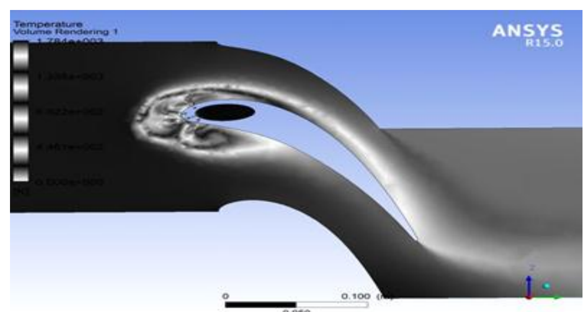

Fig 4.3. Temperature rendering with 4 row in L.E Zebra

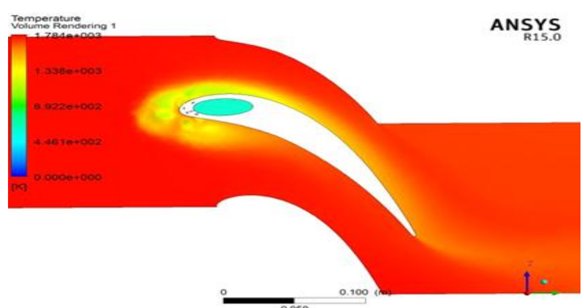

Fig 4.4. Temperature rendering with 4 row in L.E Rainbow map

Table 4.2 Computed results for varying blowing ratio's for quadrant setup2

BLOWING RATIO	EFFECTIVENESS FOR EXPERIMENTAL RESULT
0.1	0.056979
0.2	0.065467
0.3	0.069205
0.4	0.072318
0.5	0.068421
0.6	0.076472
0.7	0.073041
0.8	0.069477
0.9	0.067154
1	0.067492
1.1	0.067831
1.2	0.068169
1.3	0.068508
1.4	0.068846
1.5	0.069185

CONCLUSION

From the optimized chord location of leading edges two more holes are created at 2mm and 3mm of the leading edge to cover the entire blade. Then the simulation is carried for various blowing ratios by keeping hot stream as constant velocity and changing the coolant velocity. So from the study we find that for 2mm holes placed at leading edge locationsat4 point gives maximum film cooling effectiveness at the blowing ratio of 0.6and drop of 10.1 % of adiabatic film cooling temperature is achieved while compared with dual hole design.

REFERENCES

[1] Thole, K.A., Gritsch, M., Schulz, A. and Wittig, S., Flow field Measurements for Film Cooling Holes with Expanded Exits, Journal of Turbo machinery, 1998.

[2] K. M. Womack, R. J. Volino, and M. P. Schultz. Combined effects of wakes and jet pulsing on lm cooling. Journal of Turbo machinery, 130(4):041010-12, 2008.

[3] B.D. Mouzon, E.J. Terrell, J.E. Albert, and D.G. Bogard, "Net Heat Flux Reduction and Overall Effectiveness for a Turbine Blade Leading Edge," ASME paper GT2005-69002, 2005.

[4] D. G. Bogard and K.A. Thole, "Gas Turbine Film Cooling," accepted AIAA Journal of Propulsion and Power, 2006.

[5] S. Baldauf, M. Scheurlen, A. Schulz, and S. Wittig, "Correlation of Film-Cooling Effectiveness from Thermo graphic Measurements at Engine like Conditions," Journal of Turbo machinery 124 (2002): 686-698.

[6] K.A. Thole, A. Sinha, D. G. Bogard, and M. E. Crawford, "Mean Temperature Measurements of Jets with a Crossflow for Gas Turbine Film Cooling Application," Rotating Machinery Transport Phenomena, J. H. Kim and W. J. Yang, ed. Hemisphere Publishing Corporation, New York, New York, 1992.

[7] R. J. Goldstein, "Film Cooling," Advances in Heat Transfer 7 (1971): 321-380.

[8] B. Sen, D.L. Schmidt, and D.G. Bogard, "Film Cooling with Compound angle Holes: Heat Transfer," ASME Journal of Turbo machinery 118, no. 4 (1996): 800-806; also see note 14 (Schmidt).