International Journal of Advances in Engineering and Management (IJAEM)
Volume 3, Issue 7 July 2021, pp: 3022-3028 www.ijaem.net ISSN: 2395-5252

Static Structural and CFD Analysis of a Gas Turbine Blades Cooling with Nano Fluid as a Coolant

Nikhil Kumar H N, Shankar Nag G L,
Student, Department of Mechanical Engineering, SSIT, Tumkur, Karnataka, India.
Assistant Professor, Department of Mechanical Engineering, SSIT, Tumkur, Karnataka,

Submitted: 10-07-2021 Revised: 23-07-2021 Accepted: 26-07-2021

ABSTRACT— The turbine inlet temperature role is very significant in overall performance of the turbine. The increase in the turbine inlet temperature results in increase in overall output of the Turbine. So it is necessary to arrange elaborate methods to increase the turbine inlet temperature. But due to the current day material property limit, it is not possible to increase the turbine blade temperature to higher level without cooling. So keeping this in mind, it is necessary to arrange cooling methods to increase the turbine inlet temperature. Hence, one of the method is the blade cooling technique with different cooling hole arrangements.

The present work deals with CFD analysis of Gas turbine blade cooling with staggered holes. In this paper, two blade geometries, one with inline holes and other with staggered holes are arranged on blade surface and the analysis was done duly applied the same boundary conditions over the two models and the results are compared. From the investigation, it is observed that the uniformity of temperature distribution over the blade surface is better in staged holes arrangement compared to inline holes arrangement, further, the maximum temperature over the blade surface is less compared to the inline holes arrangement.

Index Terms ;CFD Analysis ,gas turbine blades, nano fluid

I. INTRODUCTION

The purpose of turbine technology are to extract the maximum quantity of energy from the working fluid toconvert it into useful work with maximum efficiency by means of a plant having maximum reliability, minimum cost, minimum supervision and minimum starting time. The gas turbine obtains its power by utilizing the energy of burnt gases and the air which is at high temperature and pressure by expanding through the several rings of fixed and moving blades. To get a high pressure of order 4 to 10 bar of working fluid where fuel is continuously burnt with compressed air to produce a steam of hot,

1.1 Blade design & gas turbine

The gas turbine business has changed over the last five to ten years. In the seventies and eighties a large number of physics-based technology programs were carried out, especially in the aero/thermo and acoustics area. These programs all together have led to an essential reduction of SFC and hence improved the cost situation for the airlines. In the nineties engine maker have put more emphasis on cost and quality issues. The common core family concept of the BR700 engine family is one successful example of an innovative technical approach to tackle both engine development costs, time and quality for a number of similar derivative products with a significant amount of common components.

On the other hand the cost competition has gradually changed into a time- and innovation competition (time based competition). Every player in the global aero engines market is trying to find and strengthen its unique selling proposition to stay in the market and make profit with reduced time-to-market and innovative engine concepts.

The main condition for that is for a single company to be able to design and certify the right engine at the right time for the right application. Technology is still very important and will be developed. But with limited resources, companies must re-think the cost benefit of certain purely physics-based research programs of the past because further return of investment will be quite low. High tech as the one and only unique selling proposition (USP) is not enough for today’s customers, they assume it and demand reduced costs for the product as well.

That’s the reason companies have to focus efforts and technology programs on those things that impact units cost most: reduction of hardware costs, design & re-design costs and reduction of expensive test series by numerical simulation and concurrent simultaneous engineering.

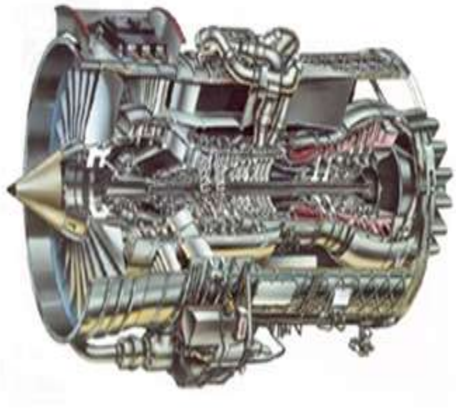


Abbildung 3 The dressed BR715 engine

One way to proceed is improvements of methods in the engineering development areas. The objective is to improve the engineering design process, which to a large extend now is computer based. Concurrent simultaneous engineering is the key word for these activities in the past. The complex blade design process is one major part of that.

II. LITERATURE REVIEW

2.1 Literature Survey

2.1.1 CFD ANALYSIS OF GAS TURBINE BLADE COOLING WITH STAGGERED HOLES

G.Narendar1, J. Narsingrao2, S.Charvani work on The present work deals with CFD analysis of Gas turbine blade cooling with staggered holes. In this paper, two blade geometries, one with inline holes and other with staggered holes are arranged on blade surface and the analysis was done duly applied the same boundary conditions over the two models and the results are compared. From the investigation, it is observed that the uniformity of temperature distribution over the blade surface is better in staged holes arrangement compared to inline holes arrangement, further, the maximum temperature over the blade surface is less compared to the inline holes arrangement. In this present work, analysis was done on Gas Turbine Blade film cooling by arranging staggered holes on suction and pressure side of blade surface with using Inconel 718 as blade material and compared the results with the inline hole arrangement results.

The Maximum temperature region (i.e., area) over the blade surface is very less in case of staggered holes arrangemnt when compared with the inline holes arrangement. The uniformity of the temperature distribution over the blade surface is better in case of holes arranged in staggered manner as compared to inline holes arrangement. Cooling effect is better when provided holes staggeredly on blade surface.

2.1.2 FLOW ANALYSIS OF GAS TURBINE BLADE FOR OPTIMUM COOLING

Ch. Indira Priyadarsini1, M. Yashwanth 2, T. Aasish Kashyap3, The aim of the work is to evaluate the need in the gas turbine industry for a systematic validated computational methodology for heat transfer problem to the optimum cooling. In order to raise thermal efficiency of a gas turbine, higher turbine inlet temperature (TIT) is needed. Various techniques have been proposed for the cooling of blades and one such technique is to have axial holes along the blade span. Cooling of blades has been a major concern since they are in a high temperature environment. In a high temperature gas turbine, turbine blade cooling designs require key technologies. Computational fluid dynamic (CFD) is used to analyze flow performance due to changing the material properties of like Nickel- Chromium alloy and Titanium-Aluminum Alloy. Three different models with different number of holes (6,8,12) were analyzed in this paper to find out the optimum number of holes for good cooling rate. In computational fluid dynamic (CFD), flow analysis is carried out. Graphs are plotted for velocity, pressure and temperature distribution for existing design. 3D model of the blade is shown. When the numbers of holes are increased in the blade, the temperature distribution falls down. Our calculations were compared adequately well with published experimental data as well as our own fundamental test data. Attention is focused on the using commercially available numerical methods, computational fluid dynamic (CFD) for fluid flow, temperature distribution for number of holes are analyzed.

2.1.3 CFD ANALYSIS ON RADIALCOOLING OF GAS TURBINE BLADE

Harsha D A, DR.Yogananda A Gas turbines are extensively used for air craft propulsion, land based power generation and industrial applications. Thermal efficiency of gas turbine improved by increasing turbine rotor inlet temperature. The current rotor inlet temperature in advanced gas turbine is for above the melting point of blade material. A sophisticated cooling scheme must be developed for continuous safe operation of gas turbines with high

performance. Gas turbines are cooled externally and internally. Several methods have been suggested for the cooling of blades and vanes. The techniques that involve to cool the blades and vanes by using cooling methods is to have radial holes to pass high velocity cooling air along the blade span. In this thesis, a turbine blade is designed and modelled in CATIA v5 and ICEM CFD software. The turbine blades are designed using cooling holes. The turbine blade is designed with 12 holes. CFD analysis is done to determine the pressure distribution, velocity, temperature distribution and heat transfer rate by applying the inlet velocities.

Thermal and Structural analysis is done to determine the heat transfer rates and strength of the blade. The present used material for blade is chromium steel. In this thesis, it is replaced with Titanium aluminium alloy. The better material for blade is analysed In this work, a turbine blade is designed and modelled in CATIA v5 software. The turbine blades are designed using cooling holes. The turbine blade is designed with 12 holes. The present used material for blade is Titanium Alloy. Thermal and Structural analysis is done By observing the CFD analysis results, the velocity of main stream(hot air) is decreased from 12 holes .But the value of pressure is slightly increased. The total heat transfer rate is maximum and the temperature of the leading edge is minimum for the blade consisting of 12 holes for titanium aluminum alloy The temperature of the surface of blade with 12 holes for chromium is minimum. It is found that the emperature leaving the trailing edge is low leading to decrease in thermal efficiency of gas turbine. Result showed that heat transfer coefficient and Nusselt number on the surface of holes are nearly constant for all blade materials with different numbers of holes also the heat transfer coefficient is high at entrance region.

2.1.4 CFD Analysis of a Gas Turbine Blade Cooling in the Presence of Holes

Gas turbines are extensively used for air craft propulsion, land based power generation and industrial applications. Thermal efficiency of gas turbine improved by increasing turbine rotor inlet temperature. The current rotor inlet temperature in advanced gas turbine is for above the melting point of blade material. A sophisticated cooling scheme must be developed for continuous safe operation of gas turbines with high performance. Gas turbines are cooled externally and internally. Several methods have been suggested for the cooling of blades and vanes. The techniques that involve to cool the blades and vanes by using cooling methods is to have radial holes to pass high velocity cooling air along the blade span. In this thesis, a turbine blade is designed and modeled in CATIA v5 software. The turbine blades are designed using cooling holes. The turbine blade is designed with no holes, 4 holes, 8 holes and 12 holes. CFD analysis is done to determine the pressure distribution, velocity, temperature distribution and heat transfer rate by applying the inlet velocities.Thermal and Structural analysis is done to determine the heat transfer rates and strength of the blade. The present used material for blade is chromium steel. In this thesis, it is replaced with Inconel 718 and N-155. The better material for blade is analyzed.

2.1.5 CFD Simulation on Gas turbine blade and Effect of Hole Shape on leading edge Film Cooling Effectiveness

Shridhar Paregouda1, Prof. Dr. T. Nageswara Rao, In order to raise thermal efficiency of a gas turbine, higher turbine inlet temperature (TIT) is needed. However, higher TIT increases thermal load to its hot-section components and reducing their life span. Therefore, very complicated cooling technology such as film cooling and internal cooling is required especially for HP turbine blades. In film cooling, relatively cool air is injected onto the blade surface to form a protective layer between the surface and hot mainstream gas. The highest thermal load usually occurs at the leading edge of the airfoil, and failure is likely to happen in this region. Film cooling is typically applied to the leading edge through an array of hole rows called showerhead. In this project initially benchmarks study the current state of heat transfer prediction for commonly used CFD software ANSYS Fluent. The predictions Reynolds-Averaged Navier-Stokes solutions for a baseline Flat film cooling geometry will be analyzed and compared with experimental data. The Fluent finite volume code will be used to perform the computations with the realizable k-ε turbulence model. The film hole is angled at 30° to the crossflow with a Reynolds number of 17,400. The focus of this investigation is to investigate advanced cooling hole geometries on film cooling effectiveness over flat surface. Three film-cooling holes with different hole geometries including a standard cylindrical hole and two holes with a diffuser shaped exit portion (i.e. a fanshaped and a laidback fanshaped hole) will be studied. Finally optimized shape of the hole configuration is included in NASA Mark II vane turbine geometry to study heat transfer characteristics of blade.

2.2 SPECIFIC AIRFOILS FILM COOLING CONFIGURATION

In most of the studies and researches have investigated film effectiveness of the generic film cooling configurations using flat surface facilities. This is while in order to investigate film effectiveness on real turbine engines, it is essential to have special considerations for film cooling configurations which are utilized for turbine airfoils. This is due to the fact that for airfoil like geometries the flow conditions would be rather different than flat plates.

In turbine airfoils, cooling of the leading edge has been reported to be of a greater importance. This is firstly due to maximum heat load, which in general occurs at leading edge. In addition, flow around the stagnation point at the leading edge of the turbine airfoils has been proven to be complex and it is necessary to consider such complexity when studying film cooling performance.

In this context, film effectiveness can be investigated with respect to different aspects such as surface curvature, surface roughness, hole blockage etc. In general, arrays of closely spaced coolant holes are involved for film cooling of turbine airfoils leading edge, which can provide a dense coverage of coolant and consequently reduce the heat loads in this region.

Significant differences in film effectiveness performance between the turbine airfoil leading edge and flat plate facilities or over the main body of airfoil has been reported. This difference is reported to be due to the big difference between interaction of the mainstream and coolant holes in these cases.

Typically the suction side of turbine airfoils is consisting of strong convex curvature, which can conclude to increase in film effectiveness. On the other hand, the pressure side have region of mild to strong concave curvature. It is known that concave curvature can decrease the film effectiveness.

In conclusion, special considerations should be given to film cooling design of turbine engine airfoils particularly.

III. PROBLEM FORMATION AND OBJECTIVE OF WORK

The overall objective of this research is to investigate the potential for reducing computational cost of CFD calculations for studying different aspects of film cooling in the early stage of gas turbine film cooling design. This has to be established by validating the CFD results using experimental measurements. In order to accomplish this, steps have been followed.

In the first step a computational domain without any cooling holes which follows experimental apparatus has been facilitated for validation of the model, using aerodynamic results (this is called the full model). This model was rather large and had disadvantage of high computational cost, thus not appropriate for investigation of different film configurations. Therefore, the computational domain is reduced.

Finally, this work aims to show the applicability of the introduced strategy for industrial applications where from industry perspective there might be computer power limitations for performing CFD analysis. Thus they can investigate different aspects of film cooling at low computational cost and turn-around time and validate obtained results with their experimental results.

3.1 SOLUTION METHODOLOGY

3.2.1 Blade Design

Gas turbine is a power producing mechanical device. Combustion gases from the combustor directly goes to turbine blade. Blade is the main component of any gas turbine that extract the power from high temperature and high-pressure gas. Blade is made of material that can withstand high temperature. Inconel -718 (Ni-Cr alloy) alloy is used for the blade.

3.2 MATERIAL SELECTIONS

Engineered suspensions of Nano sized particles (nanofluids) may be characterized by enhanced thermal properties. Due to the increasing need for ultrahigh performance cooling systems, Nano fluids have been recently investigated as next-generation coolants for car gas turbine . However, the multistate nature of Nano fluids implies nontrivial relations between their design characteristics and the resulting thermo physical properties, which are far from being fully understood. In this work, the role of fundamental heat and mass transfer mechanisms governing thermo-physical properties of Nano fluids is reviewed, both from experimental and theoretical point of view. Particular focus is devoted to highlight the advantages of using Nano fluids as coolants for automotive heat exchangers, and a number of design guidelines are reported for balancing thermal conductivity and viscosity enhancement in Nano fluids. We hope that this review may help further the translation of nanofluid technology from small-scale research laboratories to industrial application in the automotive sector.

Nano fluids are produced by several techniques:

1. Direct Evaporation
2. Gas condensation/dispersion
3. Chemical vapour condensation
4. Chemical precipitation
5. Bio-based

Several liquids including water, ethylene glycol, and oils have been used as base fluids. Although stabilization can be a challenge, on-going research indicates that it is possible. Nano-materials used so far in nanofluid synthesis include metallic particles, oxide particles, carbon nanotubes, graphene nano-flakes and ceramic particles

A bio-based, environmentally friendly approach for the covalent functionalization of multi-walled carbon nanotubes (MWCNTs) using clove buds was developed There are no any toxic and hazardous acids which are typically used in common carbon nanomaterial functionalization procedures, employed in this synthesis. The MWCNTs are functionalized in one pot using a free radical grafting reaction. The clove-functionalized MWCNTs are then dispersed in distilled water (DI water), producing a highly stable MWCNT aqueous suspension (MWCNTs Nanofluid)

IV. FEM APPROACH

A. Stress Analysis

The finite element method is a numerical technique for resolving a difference or integral equation.

It has been functional to several physical problems, where the governing differential equations are obtainable. The technique essentially consists of pretentious the piecewise incessant function for the explanation and obtaining the parameters of the purposes in a method that decreases the mistake in the solution. Here we have to find maximum stresses in each component of bearing.

B. Geometric Model

Integral Shaft Bearing product particulars are updated as per industrial requirements. Figure 4.1 shows the complete Assembly of 3D FEM geometry of whole Bearing created in CAD environment of CATIAV5 is effectively imported to ANSYS atmosphere

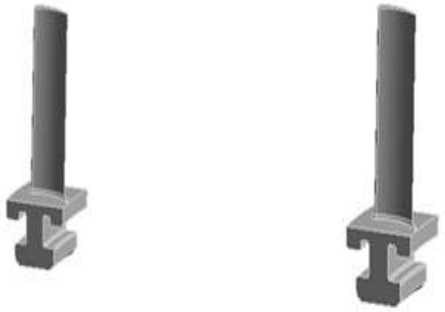


Fig 4.1 3D FEM geometry model

4.1 Mesh Generation

In this analysis, mesh generation is auto mesh generation with element size is 20. This element size is used for all the body. The hex-dominant technique is

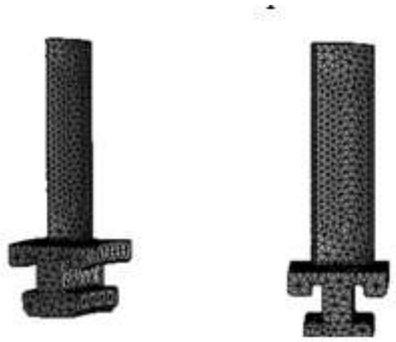


Fig 4.2 Meshed geometry model

4.2 Loading and Boundary Conditions

Filling and boundary conditions essentially contain of two steps first is provision and second is applying loads. Rotational velocity is 1420 rpm and fixed support on both the end.

4.3, Results and Observation

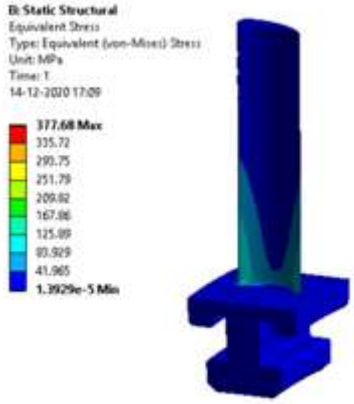


Fig 4.3 Max equivalent stress is 93.912 mpa for the applied load condition which is lesser then yield stress

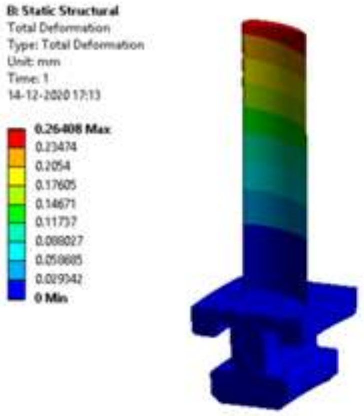
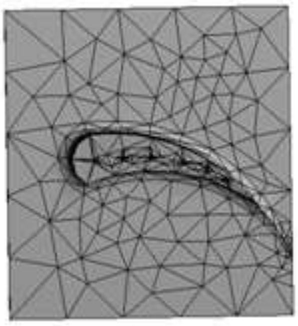


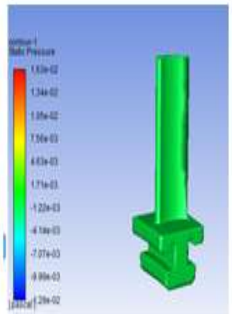
Fig 4.4, Max total deformation is 0.131 mm in horizontal view

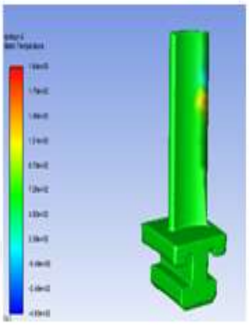
Geometry

The blade profile was produced utilizing ANSYS-15.0 Design Modeler programming. The cross-section of the edge was made by importing 374 points and afterward drawing a spline utilizing those points. The spline was then extruded to a length of 140 mm & hole diameter is 1mm and the distance between two hole is 8mm.



The Steps involved in the meshing process as per the figure 3-7 is as follows .Select mesh > mesh control > method. In the method dialogue box select all geometry and give the method option as sweep method. Select manual source and target option. Select symmetry surfaces of one side as the source and opposite side symmetry surfaces as target. Select the free face mesh type as Quad/tri. Define the number of divisions as 117.Select mesh > mesh control > sizing. In the face sizing dialogue box select the symmetry faces and define the element size as 1mm. Select mesh > mesh control >sizing.In the edge sizing dialogue box select the outer edges of the symmetry surfaces except edges at inlet and outlet. Define the element size as 1mm.Select mesh > mesh control >sizing.In the edge sizing dialogue box select the outer edges of aerofoil and the edges at the split of air domain near the trailing edge of the turbine blade. Define the element size as 0.2mm.Select mesh > mesh control > sizing. In the edge sizing dialogue box select the edges at the holes of turbine blade. Define the number of divisions as 40. Select mesh > mesh control >inflation.In the inflation dialogue box select geometry as the symmetry surfaces except the surface at the turbine blade at one end. Select the edges of air domain at the interface air and blade. Select the inflation option as total thickness. Number of layers as 27,





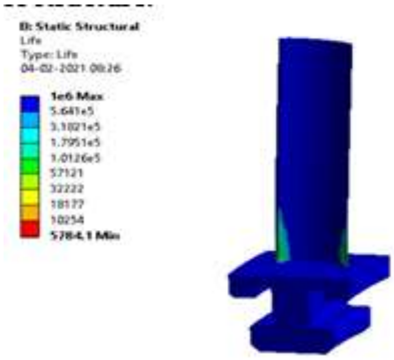
In order to have a comprehensive understanding of the flow and to compare different flow scenarios, the results of the simulation have been presented in different formats. A number of trend graphs were also drawn to show the variation over the blade volume.

- The trend of the average temperature of the blade cross-section at intervals of 10 mm was plotted with respect to distance from the inlet.
- Isometric views of the temperature contours of the blade for blade channel.
- Temperature contours of the bladeat outlet.
- Temperature contours of the fluid flow domain of the blades.
- The trend of average Nusselt Number of surfaces respect to distance from the inlet.
- The trend of average heat transfer coefficient of surfaces with respect to distance from the inlet.

V. FATIGUE ANALYSIS:

In materials science, fatigue is the wearying of a material caused by recurrently applied loads. It is the broad-minded and restricted structural damage that occurs when a material is subjected to cyclic loading. The nominal maximum stress values that cause such damage may be much less than the strength of the material classically quoted as the ultimate tensile stress limit, or the yield stress limit.

Fatigue occurs when a material is imperilled to replication loading and unloading. If the loads are above a certain threshold, microscopic cracks will begin to form at the stress concentrators such as the surface, persistent slip bands (PSBs), interfaces of constituents in the case of composites, and grain interfaces in the case of metals. Eventually a crack will reach a critical size, the crack will propagate suddenly, and the structure will fracture. The shape of the structure will significantly affect the fatigue life; square holes or sharp corners will lead to elevated local stresses where fatigue cracks can initiate. Round holes and smooth transitions or fillets will increase the fatigue strength of the structure.



VI. CONCLUSIONS

The film-cooling-channels deliver cooling effect by increasing the heat-transfer. Additional, mass-flow-rate was increased to determine an optimal mass-flow-rate but it could not be found as the trend was unsettled. Likewise, average heat-transfer coefficient, average skin-friction coefficient as well as when comparing with above two case studies temperature and heat flux is reduction in with cut out gas turbine blades hence by using the film-cooling-channels deliver cooling effect through enhancing heat-transfer. Additional, mass-flow-rate was increased to find an optimal mass-flow-rate but it could not be found as the trend was unsettled. Also, average heat-transfer coefficient, average skin-friction coefficient and average Nusselt-number is directly proportional to mass-flow-rate as well as they increase monotonously with growing mass-flow-rate

REFERENCES

[1]. http://www.andrew.cmu.edu/

[2]. Park, J.S., C.K., and Lie, Han, J.C., 1984. “ Pressure Drop in Blade Cooling Channels with Turbulence Promoters and heat Transfer.” A&M University

[3]. Hay, LaMpard& N, D.,1996, “Leave of Turbine Cooling Holes: A Review.” ASME paper 96-GT-492

[4]. Eckert, R. J., Ramsey and Eriksen, V. L., Goldstein, E. R. G., J. W., 1970, “Injection Through Inclined Circular Tubes Film Cooling,” Israel Journal of Technology, No. 1–2, Vol. 8, pp. 145–154.

[5]. Mehendale, A.B., Han, Ou and J.C., S., “Heat Transfer turbulence through Influence of high mainstream leading edge,” ASME Journal of Heat Transfer, Vol. 113, pp. 843-850, November 1991.

[6]. Shizawa, T., Honami, S., and Uchiyama, A., 1994, “Measurements of Surface Temperature and Velocity/ Temperature Field Within the Jet: Behavior of the Laterally Injected Jet in Film Cooling,” ASME Journal of Turbomachinery, Vol. 116, pp. 106–112.

[7]. Sen, B., 1996, Schmidt, D. L., “Compound Angle Holes with Film Cooling: Adiabatic Effectiveness,” ASME Journal of Turbomachinery, Vol.118, pp. 807–813.

[8]. Bons, J.P., MacArthur, C.D., and Rivir, R.B., 1996, “The Effect of High Free-Stream Turbulence on Film Cooling Effectiveness.” ASME Journal of Turbomachinery, Vol.118, pp.814-825.

[9]. Ekkad, S. V., Zapata, D., and Han, J. C., 1997, “Film Effectiveness Over a Flat Surface With Air and CO2 Injection Through Compound Angle Holes Using a Transient Liquid Crystal Image Method,” ASME Journal of Turbomachinery, Vol.119, pp. 587–593.

[10]. Schwarz, S., Goldstein, R., and Eckert, E., 1990, ‘’The Influence of Curvature on Film Cooling Performance,’’ ASME Journal of Turbomachinery, Vol. 112, pp. 472-478.

[11]. Harison, K. L., 2006, ’’Computational and Experimental Study of Film Cooling Performance Including Shallow Trench Configurations,’’ M.S. Thesis, Univ. of Texas, Austin, Texas.

[12]. www.matweb.com