Numerical Investigation Of Gas Turbine Blade Cooling Methods

G.Sri Latha[1] , G.Leela Siva Rama Prasad [2]
[1]dept Of Mechanical Engineering
[2]assistant Professor, Dept Of Mechanical Engineering
[1, 2] DJR College of Engineering and Technology , Vijayawada , A.P

Abstract- *Cooling of gas turbine blades is a major consideration because they are subjected to high temperature working conditions. Several methods have been suggested for the cooling of blades and one such technique is to have radial holes to pass high velocity cooling air along the blade span. The forced convection heat transfer from the blade to the cooling air will reduce the temperature of the blade to allowable limits. One of the major challenges in this new century is the efficient use of energy resources as well as the production of energy from renewable sources. Undoubtedly, researchers from around the world have shown that global warming has been caused in part by the greenhouse effect which is largely due to the use of fossil fuels for transportation and electricity. There are several alternative forms of energy that have already been explored and developed such as geothermal, solar, wind and hydroelectric power. Moreover, the advancement in renewable energy technologies has been possible thanks to the vast amount of research performed by scientists and engineers in order to make them more efficient and most importantly, more affordable. The affordability and performance of renewable energy technologies is the key to ensure the availability to the mass market. In this project we implement several methods to Cool the gas turbine using data available in literature with a 3D model.*

I. TURBINE

A turbine, from the Greek τύρβη, *tyrbē*, ("turbulance"), is a rotary mechanical device that extracts energy from a fluid flow and converts it into useful work. A turbine is a turbomachine with at least one moving part called a rotor assembly, which is a shaft or drum with blades attached. Moving fluid acts on the blades so that they move and impart rotational energy to the rotor. Early turbine examples are windmills and waterwheels.

Gas, steam, and water turbines usually have a casing around the blades that contains and controls the working fluid. Credit for invention of the steam turbine is given both to the British engineer Sir Charles Parsons (1854–1931), for invention of the reaction turbine and to Swedish engineer Gustaf de Laval (1845–1913), for invention of the impulse turbine. Modern steam turbines frequently employ both reaction and impulse in the same unit, typically varying the degree of reaction and impulse from the blade root to its periphery.

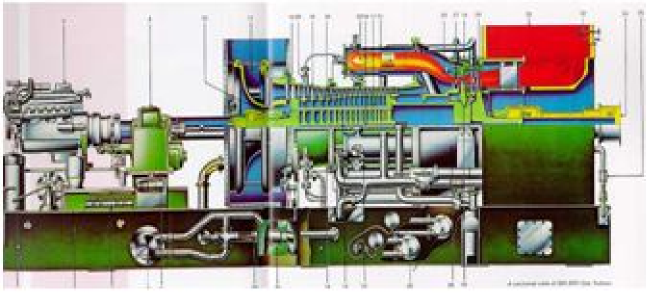
The word "turbine" was coined in 1822 by the French mining engineer Claude Burdin from the Latin *turbo*, or vortex, in a memoir, "Des turbines hydrauliques ou machines rotatoires à grande vitesse", which he submitted to the Académie royale des sciences in Paris.[3]Benoit Fourneyron, a former student of Claude Burdin, built the first practical water turbine.

II. OPERATION THEORY

A working fluid contains potential energy (pressure head) and kinetic energy (velocity head). The fluid may be compressible orincompressible. Several physical principles are employed by turbines to collect this energy:

Impulse turbines change the direction of flow of a high velocity fluid or gas jet. The resulting impulse spins the turbine and leaves the fluid flow with diminished kinetic energy. There is no pressure change of the fluid or gas in the turbine blades (the moving blades), as in the case of a steam or gas turbine, all the pressure drop takes place in the stationary blades (the nozzles). Before reaching the turbine, the fluid's *pressure head* is changed to *velocity head* by accelerating the fluid with a nozzle. Pelton wheelsand de Laval turbines use this process exclusively. Impulse turbines do not require a pressure casement around the rotor since the fluid jet is created by the nozzle prior to reaching the blading on the rotor. Newton's second law describes the transfer of energy for impulse turbines.

Reaction turbines develop torque by reacting to the gas or fluid's pressure or mass. The pressure of the gas or fluid changes as it passes through the turbine rotor blades. A pressure casement is needed to contain the working fluid as it acts on the turbine stage(s) or the turbine must be fully immersed in the fluid flow (such as with wind turbines). The casing contains and directs the working fluid and, for water turbines, maintains the suction imparted by the draft tube. Francis turbines and most steam turbinesuse this



concept. For compressible working fluids, multiple turbine stages are usually used to harness the expanding gas efficiently. Newton's third law describes the transfer of energy for reaction turbines.

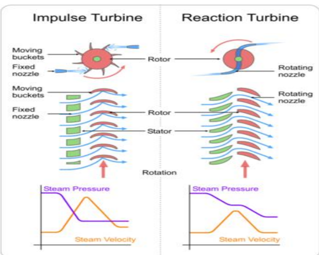


FIGURE : IMPULSE AND REACTION TURBINE

III. TYPES OF TURBINES

- Steam turbines are used for the generation of electricity in thermal power plants, such as plants using coal, fuel oil or nuclear power. They were once used to directly drive mechanical devices such as ships' propellers (for example the *Turbinia*, the first turbine-powered steam launch) but most such applications now use reduction gears or an intermediate electrical step, where the turbine is used to generate electricity, which then powers an electric motor connected to the mechanical load. Turbo electric ship machinery was particularly popular in the period immediately before and during World War II, primarily due to a lack of sufficient gear-cutting facilities in US and UK shipyards.
- Gas turbines are sometimes referred to as turbine engines. Such engines usually feature an inlet, fan, compressor, combustor and nozzle (possibly other assemblies) in addition to one or more turbines.
- Water turbines
    - Pelton turbine, a type of impulse water turbine.
    - Francis turbine, a type of widely used water turbine.
    - Kaplan turbine, a variation of the Francis Turbine.
    - Turgo turbine, a modified form of the Pelton wheel.
    - Cross-flow turbine, also known as Banki-Michell turbine, or Ossberger turbine.
- Wind turbine. These normally operate as a single stage without nozzle and interstage guide vanes. An exception is the Éolienne Bollée, which has a stator and a rotor.

IV. INTRODUCTION TO GAS TURBINE

Gas turbines in simple cycle mode

The gas turbine is the most versatile item of turbomachinery today.It can be used in several different modes in critical industries such as power generation, oil and gas ,process plants, aviation,as well domestic and smaller related industries.A gas turbine essentially brings together air that it compresses in its compressor module ,and fuel,that are then ignited. Resulting gases are expanded through a turbine. That turbine’s shaft continues to rotate and drive the compressor which is on the same shaft, and operation continues. A separator starter unit is used to provide the first rotor motion, until the turbine’s rotation is up to design speed and can keep the entire unit running. The compressor module, combustor module and turbine module connected by one or more shafts are collectively called the gas generator. The figure below illustrate a typical gas turbine sectional view.

A. General

A single-shaft gas turbine , is mounted on a platform or base which supports the basic gas turbine unit. The various assemblies , systems and components that comprise the compressor, combustion and turbine sections of the gas turbine are described in the text which follows.

B. Detail Orientation

By definition ,the air inlet of the gas turbine is the forward end ,while the exhaust is the aft end. The forward and aft ends of each component are determined in like manner wuth respect to its orientation within the complete unit.The right and left sides of the turbine or of a particular component are determined by standing forward and looking aft.

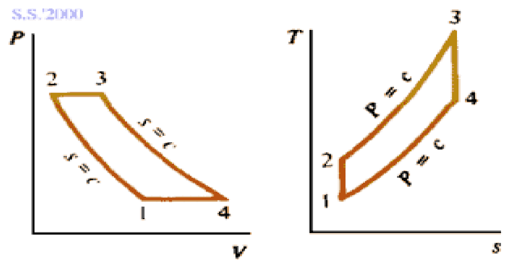


Fig. the basic gas turbine cycle [Brayton cycle]

COMPRESSOR SECTION

A. General

The axial-flow compressor section consists of the compressor rotor and the compressor casing .Within the compressor casing are the variable inlet guide vanes, the various stages of rotor and stator blading , and the exit guide vanes.

In the compressor , air is confined to the space between the rotor and stator where it is compressed in stages by a series of alternate rotating(rotor) and stationary(stator) airfoil-shaped blades.

The rotor blades supply the force needed to compress the air in each stage and the stator blades guide the air so that it enters the following rotor stage at the proper angle. The compressed air exits through the compressor dischargs casing to the combustion chambers.

B. Rotor

The compressor portion of the gas turbine rotor is an assembly ofwheels, a speed ring, ties bolts, the compressor rotor blades ,and a forward stub shaft.Each wheel hasslots broached around its periphery**.**

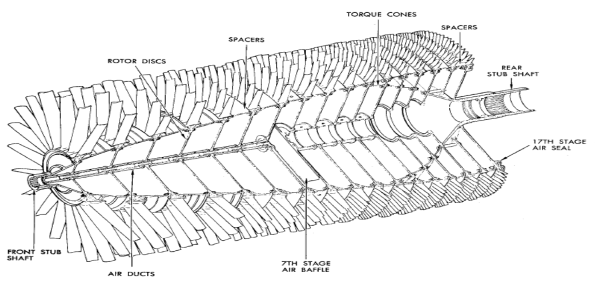


Fig. compressor rotor

The rotor blades and spacers are inserted into these slots and held in axial position by staking at each end of the slot.The wheels are assembled to each other with mating rabbets for concentricity control and are held together with tie bolts.Selective positioning of the wheels is made during assembly to reduce balance correction.

After assembly, the rotor is dynamically balanced. The forward stub shaft is machined to provide the thrust collar which carries the forward and aft thrust loads. The stub shaft also provides the journal for the No.1 bearing, the sealing surface for the No.1bearing oil seals and the compressor low-pressure air seal. The stage 17 wheel carries the rotor blades and also providesthe sealing surface for the high-pressure air seal and the compressor-to-turbine marriage flange.

C. Stator

1. General

The casing area of the compressor section is composed of three major sections . These are :

a. Inlet casing
b. Compressor casing
c. Compressor discharge casing

a. Inlet casing

The inlet casing is located at the forward end of the gas turbine .Its prime function is to uniformly direct air into the compressor. The inlet casing also supports the #1 bearing assembly. The #1 bearing lower half housing is integrally cast with the inner bell mouth. The upper half bearing housing is a separate casting, flanged and bolted to the lower half. The inner bell mouth is positioned to the outer bell mouth by nine airfoil-shaped radial struts.

b. Compressor casing

The forward compressor casing contains the stage 0 through stage 4 compressor stator stages . The compressor casing lower half is equipped with two large integrally cast trunnions which are used to lift the gas turbine when it is separated from its base. The aft compressor casing contains stage 5 through stage 12 compressor stator stages. Extraction ports in aft casing permit removal of 13^{th} stage compressor air. This air is used for cooling functions and is also used for pulsation control during startup and shutdown.

c. Compressor discharge casing

The compressor discharge casing is the final portion of the compressor section. It is the longest single casing, situated at mid point—between the forward and aft supports—and is, in effect, the keystone of the gas turbine structure.

4.3 Introduction to Solidworks Flow Simulation

With the full SolidWorks Flow Simulation product, you have the following advanced functionality:

4.3.1 Heat Transfer Analysis

- Calculate heat flow in the adiabatic walls approximation or in solid bodies.
- Specify different types of heat sources.
- Assign to models a broad range of solid materials that are stored in the engineering database.
- Define your own materials by assigning them values for physical properties such as thermal conductivity, heat capacity, etc.
- Calculate radiation heat. The engineering database contains radiative surfaces such as Blackbody Wall, Whitebody Wall, Grey Body with an arbitrary albedo, and a broad range of surfaces of real materials.

4.3.2 Fluids and Flow Types

- Analyze the flow of up to ten fluids of different types (liquids, gases/steam, real gases, non-Newtonian liquids and compressible liquids). The database contains numerous fluids with predefined properties.
- Analyze a problem with multiple fluids of different types, provided you separate the areas of the different fluids from each other using fluid subdomains.
- Analyze mutual dissolution of fluids. Mixing fluids must be of the same type.
- Define your own fluids.

Initial Settings

Before you start the calculation, Flow Simulation offers additional settings. If you set the initial condition values closer to the anticipated final parameters, calculation performance improves.

Initial fluid parameters You can set these parameters globally. For an assembly, you can set them locally for a subassembly or individual part.

- Temperature
- Pressure
- Flow velocity
- Fluid composition

Initial temperature Set the initial temperature of a solid.

Initial mesh parameters Set additional parameters that control how the analysis resolves the solid/fluid interface, curved surfaces, narrow channels, small solid features, etc. You can apply these settings globally or, for assemblies, to a subassembly or an individual part.

Boundary Conditions

You can set these boundary conditions.

For inlet and outlet:

- Mass volume
- Volume flow
- Velocity
- Mach number
- Static pressure
- Total pressure
- Environment pressure
- Wall pressure

For inlet only:

- Flow velocity profile, swirl, or vector
- Temperature
- Composition (for assemblies)
- Turbulence parameters

Black Box Entities

To reduce analysis time, Flow Simulation includes several pre-built "black boxes." Black boxes have tabulated integral input and output parameters and are included in calculations. Flow Simulation does not resolve them during an analysis.

Fan An idealized fan that is fully defined by its fan curve, which means the tabulated dependency of volume flow versus pressure drop. You can use the fan as an inlet, outlet, or internal fan. The database contains fan curves for selected industrial fans. You can also define fan curves yourself.

Heat Sink An idealized fan combined with the heat sink. Flow Simulation defines the heat sink by the fan curve and the heat resistance curve.

Thermoelectric Cooler An idealized Peltier cooling device defined by the maximum temperature difference it can develop.

Viewing Results

Flow Simulation includes these features to view the results:

Plots	Animations	Particle studies
3D-profile plots	Flow Trajectories	Reports
Cut Plots	Goals	Point, surface, and volume parameters
Surface Plots	Iso surfaces	XY Plots

You can also obtain the final value of any physical parameter, including flow rate, pressure drop, etc., at a given point, or the maximum, minimum, average, or weighted averaged over a surface or volume area.

V. FILM COOLING

The coolant, i.e. air, after passing through of the interior of the turbine blade is made to exit the blade through holes on the leading edge. This cool air comes out of the leading edge and forms a layer or a thin film protecting the blade from the hot gases.

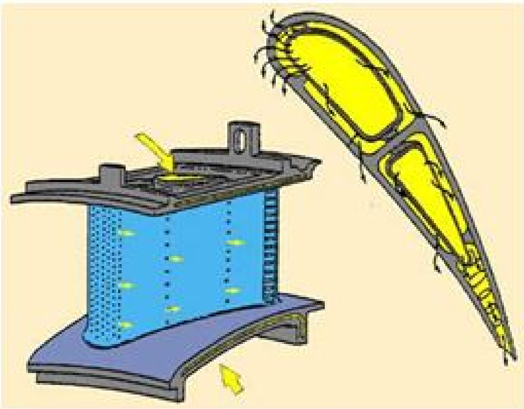


Fig.2Source: (from http://lttwww.epfl .ch/research/htprojects/fi lmcool.htm)

The primary process by which film cooling reduces the heat transfer to the wall is by reducing the gas temperature near the wall, i.e. reducing the driving temperature potential for heat transfer to the wall. As the coolant flows from the coolant holes, it mixes with the mainstream gas resulting in an increase in coolant temperature. A typical example of this is presented in figure 2 which shows measurements of the temperature profile along the centerline of a coolant jet as it flows downstream of the coolant hole.

$$\theta = \frac{T_\infty - T}{T_\infty - T_e}$$

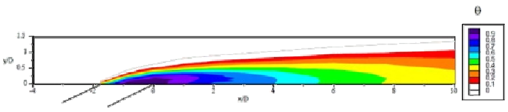


Fig. 3.Thermal profiles showing the coolant

The coolant temperature at the wall will be at the adiabatic wall temperature, Taw, and this temperature is generally assumed to be the driving temperature potential for heat transfer into the wall. Generally a normalized form of Taw, referred to as the adiabatic effectiveness or fi lm effectiveness, is used to characterize the film cooling performance.

The film effectiveness, η, is defined as follows:[3]

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

Where Tc,exit is the coolant temperature at the coolant hole exit. For perfect film cooling performance, the film effectiveness would have a value of η = 1.0, i.e. Taw would be equal to the coolant temperature at the exit of the hole; while a value of η = 0 would indicate that the film cooling has not reduced the gas temperature at the wall. In practice, η values decrease rapidly downstream of the coolant holes due to the strong turbulent dispersion of the coolant jet. The primary measure of film cooling performance is the film effectiveness, η, since this has a dominating effect on the net heat flux reduction.[3]

Ideally a film of coolant would be introduced to the surface of an airfoil using a slot angled almost tangential to the surface in order to provide a uniform layer of coolant that remain attached to the surface. However, long slots in the airfoil would seriously reduce the structural strength of the airfoil, and hence are not feasible. Consequently coolant is typically introduced to the airfoil surface using rows of holes. The film cooling performance is dependent on the hole geometry and configuration of the layout of the holes. Furthermore, various factors associated with the coolant and mainstream flows, and the airfoil geometry, also significantly affect the cooling performance.

Factors affecting Film cooling performance

The various factors influencing the performance of the film cooling are listed in table 1 and some of the are discussed in greater detail further.

Coolant/Mainstream Conditions	Hole Geometry and Configuration	Airfoil Geometry
Mass flux ratio*	Shape of the hole*	Hole location - leading edge - main body - blade tip - endwall
Momentum flux ratio*	Injection angle and compound angle of the coolant hole *	
Mainstream turbulence*	Spacing between holes, *P/d*	Surface curvature*
Coolant density ratio	Length of the hole, *l/d*	Surface roughness*
Approach boundary layer	Spacing between rows of holes and number of rows	
Mainstream Mach number		
Unsteady mainstream flow		
Rotation		

Table 1 Factors Affecting Film Cooling PerformanceSource: Bogard, Airfoil film cooling

Mainstream Effects on Film Cooling Performance

There are a number of mainstream factors that can affect film cooling performance including approach boundary layers, turbulence levels, Mach number, unsteadiness, and rotation [4]. Because of the very high levels of mainstream turbulence exiting the combustor and entering the turbine section, turbulence levels have the largest effect on film cooling performance. High mainstream turbulence levels degrade film cooling performance by increasing heat transfer coefficients and generally decreasing film effectiveness.

Film Cooling with Shaped Holes

Improved film effectiveness can be achieved if the exit of the hole is expanded so that coolant is slowed through a diffuser. There are two advantages for such a “shaped hole”: the coolant exit velocity is reduced and a broader jet cross-section is presented to the mainstream flow. Both these characteristics will reduce the tendency for the coolant jet to separate. This results in good film effectiveness levels for shaped holes.

Airfoil Surface Effects on Film Cooling Performance

Surface curvature and surface roughness are significant factors affecting film cooling performance. Clearly for turbine airfoils strong convex curvature exists around the leading edge and along the suction side of the airfoil. Sometimes strong concave curvature is encountered on the pressure side of the airfoils. Surface roughness varies with the length of operation of the engine; new airfoils are relatively smooth, but after some period of operation the surfaces can become quite rough due to erosion, spalation of thermal barrier coatings, and deposition of contaminants. [3]

Surface roughness degrades film cooling performance by increasing the heat transfer coefficient and potentially reducing film effectiveness. Heat transfer coefficients can be increased by as much as 50% to 100% [5]. The decrease in film effectiveness at the optimum blowing ratio was primarily due to the roughness upstream of the coolant holes. The upstream roughness doubled the boundary layer thickness and significantly increased turbulence levels which resulted in more separation of the coolant jets and increased dispersion of the coolant.

INTERNAL COOLING

A typical cooled turbine blade is shown in figure 4. As shown in the figure, the vane is hollow, so cooling air can pass through the vane internally. The coolant is extracted from the internal channel for impingement and pin fin cooling. Jet impingement is a very aggressive cooling technique which very effectively removes heat from the vane wall. However, this technique is not readily applied to the narrow trailing edge. The vane trailing edge is cooled using pin-fins (an array of short cylinders). The pin-fins increase the heat transfer area while effectively mixing the coolant air to lower the wall temperature of the vanes. After impinging on the walls of the airfoil, the coolant exits the vane and provides a protective film on the vane’s external surface. Similarly, the coolant travelling through the pin-fin array is ejected from the trailing edge of the airfoil. [6]

IMPINGEMENT COOLING

Impingement cooling is commonly used near the leading edge of the airfoils, where the heat loads are the greatest. With the cooling jets striking (impinging) the blade wall, the leading edge is well suited for impingement cooling because of the relatively thick blade wall in this area.

Impingement can also be used near the mid-chord of the blade. Figure 4 shows jet impingement located throughout the cross-section of an inlet guide vane. Several aspects must be considered when developing efficient cooling designs. The effect of jet-hole size and distribution, cooling channel cross-section, and target surface shape all have significant effects on the heat transfer coefficient distribution. Jet impingement near the mid-chord of the blade is very similar to impingement on a flat plate; however, the sharp curvature at the leading edge of the vane must be considered when utilizing impingement in this region.

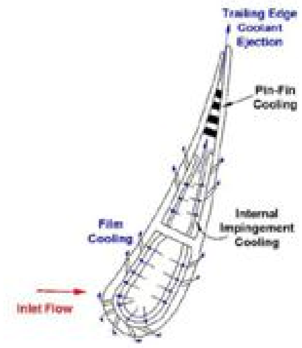


Fig. 4 Turbine Vane Cross-Section with Impingement and Trailing Edge Pin-Fin Cooling
Source: Han and Wright

Rotational Effect on Jet Impingement Cooling

It has been concluded by various studies that rotation of the blades decreases the impingement heat transfer, but the effective heat transfer is better than a smooth rotating channel. The effect of rotation is least when jet direction has an angle of 45° to rotation direction. However, a maximum of 40% reduction in heat transfer is noted when jet direction is perpendicular to rotation direction. This may be because the Coriolis force creates a swirl action on the spent flow and also deflects the jet when jet direction is parallel to rotation direction. [6]

PIN-FIN COOLING

Due to manufacturing constraints in the very narrow trailing edge of the blade, pin-fin cooling is typically used to enhance the heat transfer from the blade wall in this region. The pins typically have a height-to-diameter ratio between ½ and 4. In a pin-fin array heat is transferred from both the smooth channel end wall and the numerous pins. Flow around the pins in the array is comparable to flow around a single cylinder. As the coolant flows past the pin, the flow separates and wakes are shed downstream of the pin. In addition to this wake formation, a horseshoe vortex forms just upstream of the base of the pin, and the vortex wraps around the pins. This horseshoe vortex creates additional mixing, and thus enhanced heat transfer.

Many factors must be considered when investigating pin-fin cooling. The type of pin-fin array and the spacing of the pins in the array effect the heat transfer distribution in the channel. The pin size and shape also have a profound impact on the heat transfer in the cooling passage. Because pin-fins are commonly coupled with trailing edge ejection (as shown in figure 2), the effect of this coolant extraction must also be considered.

Pin Array and Partial Length Pin Arrangement

There are two array structures commonly used. One is the inline array and the other is the staggered array. Figure 5 shows a typical experimental test model with a staggered array of pin-fins.

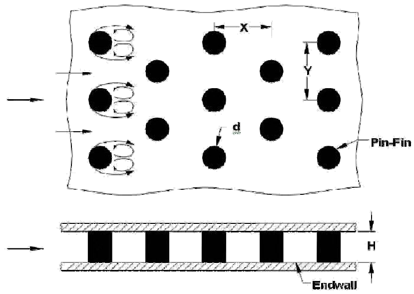


Fig. 5 . A Typical Test Model and Secondary Flow for Pin-Fin Cooling Studies
Source: Source: Han and Wright

A closer spaced array (smaller x/D) shows a higher heat transfer coefficient. Their observations of various researches have clearly indicated that addition of pin-fins significantly enhances the heat transfer coefficient. However, the addition of pins also increases the pressure drop in the flow channel. The average Nusselt number in a channel with short pin-fins is primarily dependent on the Reynolds number of the flow, and a weaker dependence is shown for the pin spacing. [6]

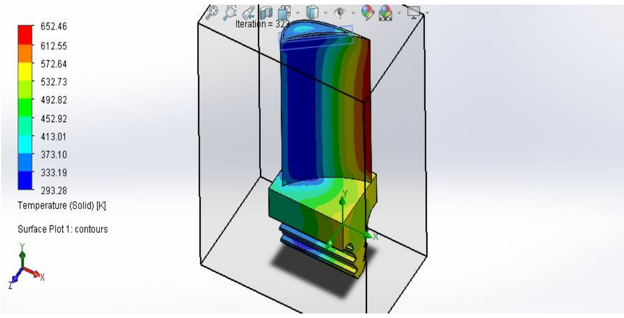
Effect of Pin Shape and Array Orientation

Straight cylinders in staggered array formation have the highest heat transfer followed by filleted cylinders in the staggered formation. It is interesting to note that the fillet cylinder inline formation has better heat transfer than the straight cylinders in the inline formation. Though a staggered array gives higher heat transfer coefficients, performance of the inline straight cylinders is best among the group and the fillet cylinders in staggered formation are the worst. [6]

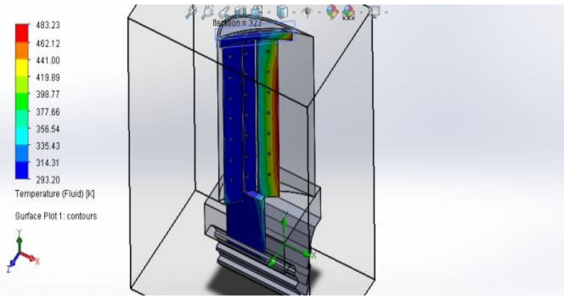
The cube-shaped pins have the highest mass transfer coefficients among the shapes considered and round pins have the lowest mass transfer coefficients. Corresponding pressure loss coefficients are higher for the cube and diamond shaped pins relative to the circular pins.

VI. RESULTS

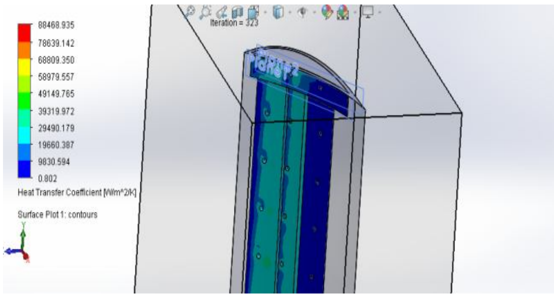
Pin Fin Impegmentation



Temperature of the Blade with cooling

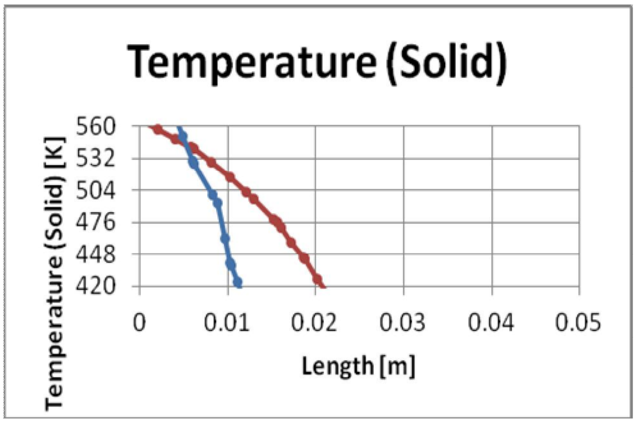


Temperature Of fluid

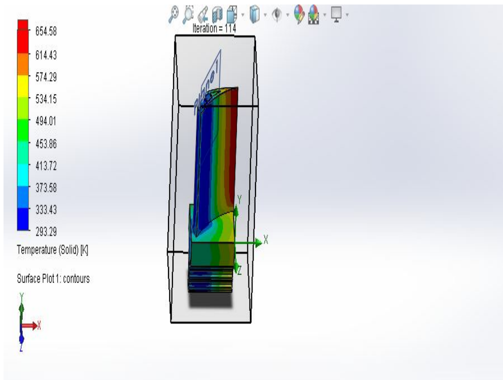


Heat transfer Coefficient

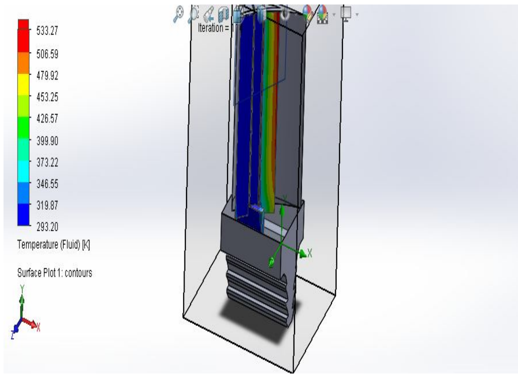
Graphs of temperature



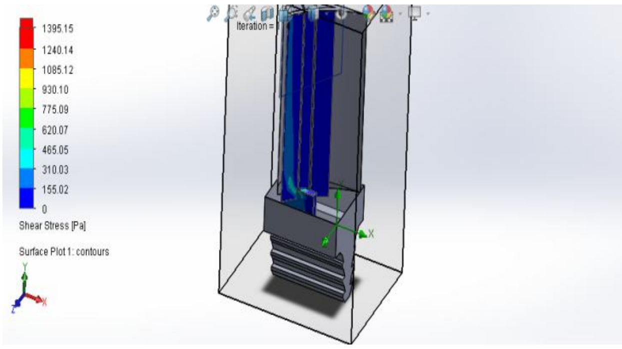
Film Cooling



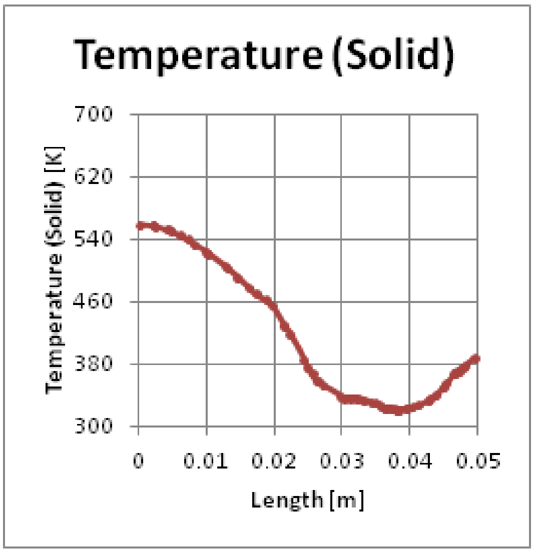
Temperature Cooling



Temperature Fluid



Shear Stress



Graph Length Vs Temperature

VII. RESULTS SUMMARY

In Pin fin Impegmentation the Temperature when assumed as a 700k free stream temperature the total temperature reduced in to 654K in most of the part and average temperature is set to be 380k where as in film cooling the temperature is 656k and 420k Solid and fluid temperature From the above design and analysis The possible conclusion can be made Is the pin fin impigmentation is the best way of cooling the gas turbine blade.

REFERENCES

[1] Koff, Bernard L. (2003). "Gas Turbine Technology Overview - A Designer's Perspective". AIAA/ICAS International Air and Space Symposium and Exposition: The Next 100 Years. 14–17 July 2003, Dayton, Ohio. AIAA 2003-2722.Flack, p. 429.Bogard, Airfoil film cooling

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

[3] J.L. Rutledge, D. Robertson, and D.G. Bogard, "Degradation of Film Cooling Performance on a Turbine Vane Suction Side Due to Surface Roughness," ASME GasTurbine Expo, GT2005-69045, 2005; also see note 19 (Bogard).

[4] Han and Wright, Enhanced Internal Cooling of Turbine Blades and Vanes F.T. Willett and A.E. Bergles, "Heat Transfer in Rotating Narrow Rectangular Ducts with Heated Sides Oriented at 60- Degree to the R-Z Plane," ASME Paper No. 2000-GT-224 (2000); F.T. Willett and A.E. Bergles, "Heat Transfer in Rotating Narrow Rectangular Pin-Fin Ducts," Experimental Thermal and Fluid Science 25 (2002): 573-582.

[5] J.H. Wagner, B.V. Johnson, and F.C. Kopper, "Heat Transfer in Rotating Serpentine Passages With Smooth Walls," ASME Journal of Turbomachinery 113 (1991): 321-330;

[6] S. Dutta and J.C. Han, "Rotational Effects on the Turbine Blade Coolant Passage Heat Transfer," Annual Review of Heat Transfer 9 (1997): 269-314.

[7] 69. J.H. Wagner, B.V. Johnson, R.A. Graziani, and F.C. Yeh, "Heat Transfer in Rotating Serpentine Passages With Trips Normal to the Flow," ASME Journal of Turbomachinery 114 (1992): 847-857.

[8] B.V. Johnson, J.H. Wagner, G.D. Steuber, and F.C. Yeh, "Heat Transfer in Rotating Serpentine Passages with Trips Skewed to the Flow," ASME Paper No. 92-GT-191, ASME Journal of Turbomachinery. 116 (1992): 113-123