International Journal of Latest Engineering and Management Research (IJLEMR)
ISSN: 2455-4847
www.ijlemr.com || Volume 07 - Issue 01 || January 2022 || PP. 31-52

Computational Analysis of Turbine Blade Cooling in Aircraft Gas Turbine Engines

Sivaraman S[1], Magesh Kumar M[2], Prakaas M[3], Reshwan Raj[4], Ashwin Nanda Kumar[5,]
[1,2] Assistant Professors, Dept. of Aeronautical Engineering,
Hindusthan College of engineering and technology,
Coimbatore, Tamilnadu, India

Abstract: The objective of this project is to analyse the Turbine blade cooling in aircraft gas turbine engine. Generally, Turbine blade in aircraft engines to high temperature of around 1600'c. At this elevated temperature, the turbine blades damaged due to the impact of the higher thermal stresses. But only in the turbine inlet temperature is high, large amount of thrust can be generated. On the other hand, on increasing the turbine inlet temperature, the blades may exceed its metallurgical limits. In order to prevent this, turbine blade cooling is mandatory. There are several methods for turbine blade cooling. In this project, we are going to analyse the turbine blade cooling effectiveness using cooling technique by varying certain parameters like hole arrangement, location orientation and shape.
Keywords: CATIA; Turbine Blade; ANSYS; Stress; Strain; CFD.

Introduction:

The purpose of turbine technology is to extract the maximum quantity of energy from the working fluid to convert 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, fast-moving gas as shown in figure 1[1]. This gas stream is used to power the compressor that supplies the air to the engine as well as providing excess energy that may be used to do other work,which is essential for expansion a compressor, is required. The quantity of the working fluid and speed required are more so generally a centrifugal or an axial compressor is required. The turbine drives the compressor so it is coupled to the turbine shaft.

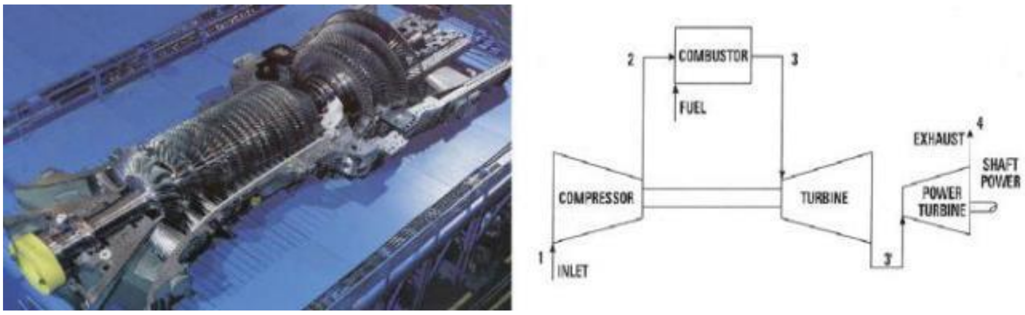


Figure 1: Gas Turbine Simple Open Cycle

If after compression the working fluid were to be expanded in a turbine, then assuming that there were no losses in either component, the power developed by the turbine can be increasedby increasing the volume of working fluid at constant pressure or alternatively increasing the pressure at constant volume.

Either of these may be done by adding heat so that the temperature of the working fluid is increased after compression. To get a higher temperature of the working fluid a combustion chamber is required where combustion of air and fuel takes place giving temperature rise to the working fluid. Gas turbines have been constructed to work on the following: -oil, natural gas, coal gas, producer gas, blast furnace and pulverized coal. The engine consists of three main parts.

- The Compressor section.
- The Combustion section (the combustor).
- The turbine (and exhaust) section.

The Turbine compressor usually sits at the front of the engine. There are two main types of compressors, the centrifugal compressor and the axial compressor. The compressor will draw in air and compress it before it is fed into the combustion chamber. In both types, the compressor rotates and it is driven by a shaft that passes through the middle of the engine and is attached to the turbine as shown below in figure 2 [1], [2].

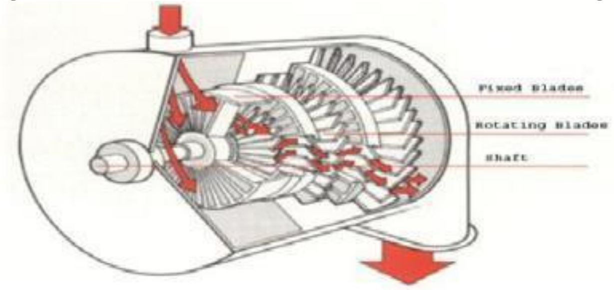


Figure 2: Turbine Blade Coupled to Centrifugal Compressor

Types of Gas Turbine
There are four main types of gas turbine: The turbojet, turbofan, turboprop and turbo shaft.

Fuel for Gas Turbine Power Plants:
Gas turbine fuel systems are similar for all Turbines. For the most common fuels, which are natural gas, LNG (liquid natural gas), and light diesel, the fuel system consists:
A fuel delivery system, Fuel nozzles, Fuel additives (to deal with vanadium), Fuel washing (to deal with sodium and potassium Salts) and Modifications to the fuel delivery system.

Natural Gas:
Natural gas comprises over 80% methane with minor amounts of ethane, propane, butane, and heavier hydrocarbons. It may also include carbon dioxide, nitrogen, and hydrogen. There is a plethora of blends of natural gas available worldwide.

Applications of Gas Turbine:
The following are the applications of gas turbine as shown in figure 3.

- **Land Applications:** Central power stations, Industrial and Industrial.
- **Space Applications:** Turbo jet and Turbo prop.
- Marine application [1].

Figure 3: Some Examples of Application Gas Turbine

Turbine Blade:
The rotor blades of the turbo machine are very critical components and reliable operation of the turbo machine as a whole depends on their repayable operation. The major cause of break down in turbo machine is the failure of rotor blade. The failure of the rotor blade may lead to catastrophic consequences both physically and economically. Hence, the proper design of the turbo machine blade plays a vital role in the proper functioning of the turbo machine as shown in figure 4.

Figure 4: Turbine Blade

A good design of the turbo machine rotor blading involves the following:
- Determination of geometric characteristics from gas dynamic analysis.
- Determination of steady loads acting on the blade and stressing due to them.
- Determination of natural frequencies and mode shapes.
- Determination of unsteady forces due to stage flow interaction.
- Determination of dynamic forces and life estimation based on the cumulative damage fatigue theories [3].

Production of Blades:

Blades may be considered to be the heart of turbine and all other member exist for the sake of the blades. Without blade there would be no power and the slightest fault in blade would mean a reduction in efficiency and costly repairs.

The following are some of the methods adopted for production of blades.

Rolling: Sections are rolled to the finished size and used in conjunction with packing pieces. Blades manufactured by this method do not fail under combined bending and centrifugal force.

Machining: Blades are also machined from rectangular bars. This method has more or less has the same advantage as that of first. Impulse blade is manufactured by this technique.

Forging: Blade and vane sections having aerofoil sections are manufactured by specialist techniques.

Extrusion: Blades are sometimes extruded and the roots are left on the subsequent machining. This method is not reliable as rolled sections, because of narrow limits imposed on the composition of blade material.

Turbine Blade Cooling:

Unlike steam turbine bladings, gas turbine blading's need cooling. The objective of the blade cooling is to keep the metal temperature at a safe level to ensure a long creep life and low oxidation rates. Although it is possible to cool the blades by liquid using thermosyphon and heat pipe principal, but the universal method of blade cooling is by cool air or working fluid flowing through internal passage in the blades. The mean rotor blade temperature is about 3500C below the prevailing gas temperature after efficient blade cooling as shown below in figure 5.

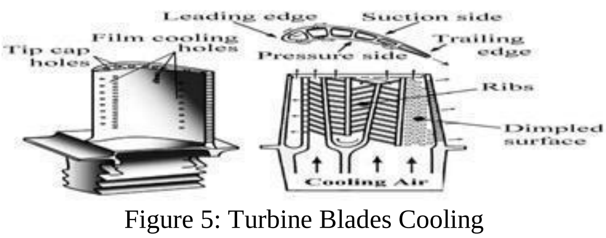


Figure 5: Turbine Blades Cooling

Due to corrosion and corrosion deposits turbine blades fail. To protect it from corrosion, the uses of pack-aluminized coatings are used. The main elements used are aluminium, nickel, and chromium.

Assumption System and Simulations:

Turbine Blade Materials:

Advancements made in the field of materials have contributed in a major way in building gas turbine engines with higher power ratings and efficiency levels. Improvements in design of the gas turbine engines over the years have importantly been due to development of materials with enhanced performance levels. Gas turbines have been widely utilized in aircraft engines as well as for land-based applications importantly for power generation. Advancements in gas turbine materials have always played a prime role – higher the capability of the materials to withstand elevated temperature service, more the engine efficiency; materials with high elevated temperature strength to weight ratio help in weight reduction. A wide spectrum of high-performance materials - special steels, titanium alloys and super alloys - is used for construction of gas turbines. The material available limits the turbine entry temperature (TET). The properties required are as follows (a) tensile strength (b) resistance to high frequency vibration fatigue stresses(c) low frequency thermal fatigue stresses (d) resistance to erosion and corrosion.

Stainless Steel Alloy:

In spite of this there is a group of iron-base alloys, the iron-chromium-nickel alloys known as stainless steels, which do not rust in sea water, are resistant to concentrated acids and which do not scale at temperatures up to 1100°C. It is this largely unique universal usefulness, in combination with good mechanical properties and manufacturing characteristics, which gives the stainless steels their raison d'être and makes them an indispensable tool for the designer. The usage of stainless steel is small compared with that of carbon steels but exhibits a steady growth, in contrast to the constructional steels. Stainless steels as a group are perhaps more heterogeneous than the constructional steels, and their properties are in many cases relatively unfamiliar to the designer. In some ways stainless steels are an unexplored world but to take advantage of these materials will require an increased understanding of their basic properties.

Titanium Alloy:

These titanium alloys are mainly used for substituting materials for hard tissues. Fracture of the alloys is, therefore, one of the big problems for their reliable use in the body. The fracture characteristics of the alloys are affected by changes in microstructure. Therefore, their fracture characteristics, including tensile and fatigue characteristics should be clearly understood with respect to microstructures. The fracture characteristics in the simulated body environment also be identified because the alloys are used as biomedical materials. The effect of living body environment on the mechanical properties is also very important to understand.

Alpha Structure (α Alloy): with alpha stabilizer elements present, these alloys possess excellent creep resistance. They are also used largely in cryogenic applications.

Alpha Beta Structure (α–β Alloy): this group contains both alpha and beta stabilizer elements. This is the largest group in the aerospace industry.

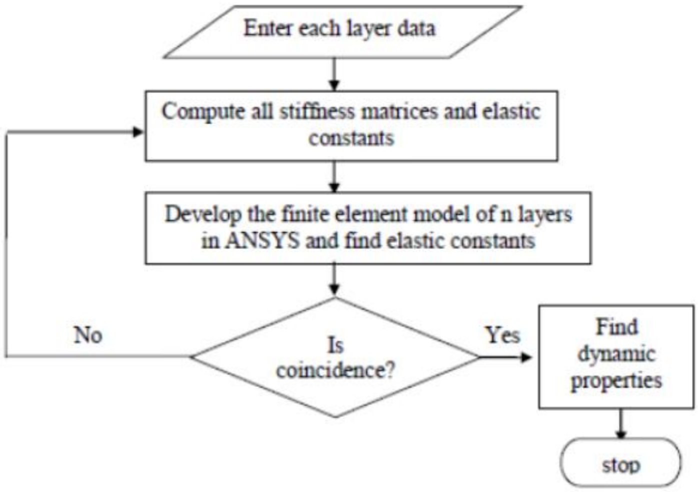
Beta Structure (β Alloy): with beta stabilizers this group has high harden ability and high strength, but also a higher density. Titanium alloys use in aero engines, Automotive, Airframes and road transport, Dental alloys, geothermal plant, Marine and Military hardware.

Properties	**Units**	**Titanium Alloy**	**Stainless Steel Alloy**	**Aluminum 2024 Alloy**
Density	kg/m3	4700	8025	2725
Thermal conductivity(K)	W/m.k0	10	33.5	180
Coefficient of thermal Expansion(α)	C0	8.8	14.5	23.3
Specific heat (CP)	J/kg.K0	544	448	880
Modules of elasticity (E)	GPa	205	200	73
Poisson ratio(μ)	…….	0.33	0.3	0.33
Melting point	0C	1649	1451	565
Ultimate tensile strength	MPa	1000	1050	470

Methodology:

The following methodology is being adopted to carry out the above-mentioned objectives:
1. The ANSYS achieved by aircraft landing gear and CAD model was designed by CATIA V5
2. Using ANSYS the overall load is computed and tried to validate with classical theory.
3. Using these equivalent properties of the composite the natural frequency computations are done.

Introduction to Catia:



CATIA which stands for computer aided three dimensional interactive applications is the most powerful and widely used CAD (computer aided design) software of its kind in the world. CATIA is owned/developed by Dassault system of France and until 2010, was marketed worldwide by IBM.

The Following general methodologies and best practices can be followed in the modelling of components in CATIA. The Below methodologies and best practices followed will help in capturing the design intent of the Feature that is to be Modelled and will make the design robust and easy to navigate through.

- Specification tree structuring
- Renaming appropriate features & bodies in specification tree
- Handling input data & foreign bodies
- Dimensioning & constraining in sketches
- Parameters and relations.

Specification Tree Structuring:

The SPECIFICATION TREE is the place where the histories of the features modeled are captured. So it is highly important to have an organized tree structure which gives ease for navigation of the features when any modification takes place.

The SPECIFICATION TREE in a structured manner. The Machining Body features are grouped under one body and base block features in another and so on with appropriate feature operations.

It is also important in structuring the reference and construction element in the tree in an orderly manner.

The points that would be often used (like the Global Origin Point 0, 0, 0,) can be created under Points GEOMETRICAL SET and any reference planes defining legal limits can be created in the planes GEOMETRICAL SET.

Handling Input Data and Foreign Bodies:

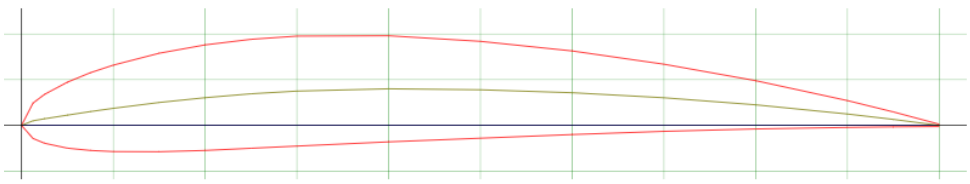
Any external data that are to be handled in the model can be grouped under a GEOMETRICAL SET called input data which can be used in the model when situation demands. Some foreign elements like planes, points, curves and surfaces that would be used in the modelling process. By grouping the foreign elements in a separate GEOMETRICAL SET, it is easy to identify them in the SPECIFICATION TREE.

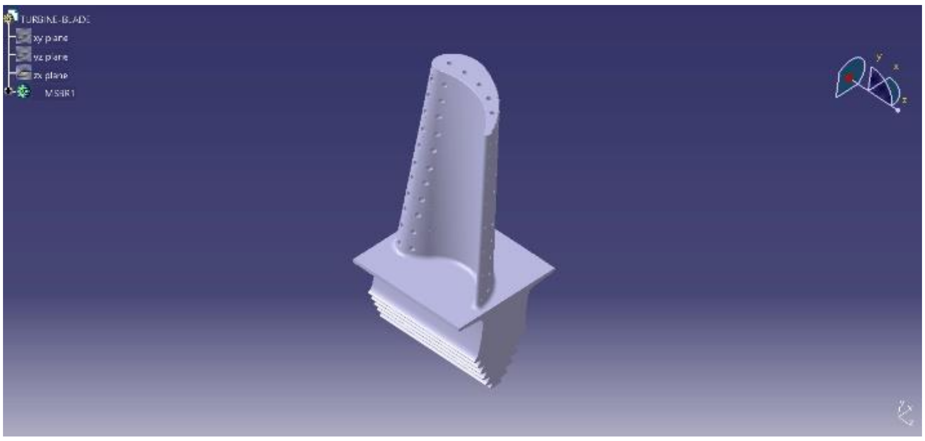
Dimensioning and Constraining in Sketches:

Planes should be intersected in the sketches and made as construction elements and should be used as dimension reference for geometries, this helps in identifying the dimension line clearly in a complex sketch. Equivalent dimension shouldbe used wherever possible to minimize modification time in the sketches. Usage of sketch analysis command is mandatory at the end of every sketch build which helps in diagnosing the sketch thereby identifying abnormalities. Robust design Intent can be Achieved with the Integration of Parameters and Relations.

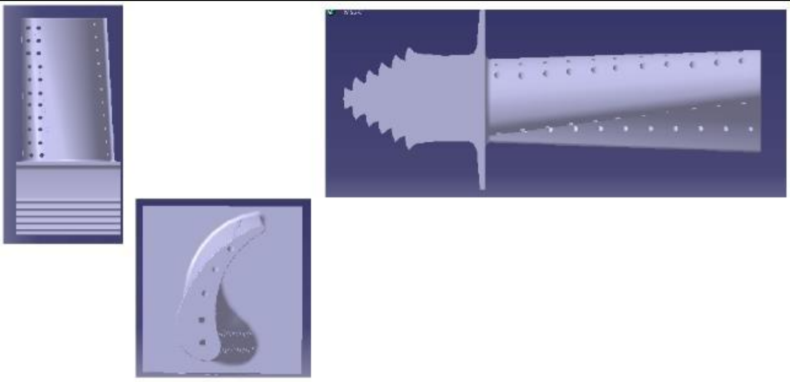
Air foil 4412 coordinates

NO.	X	Y	NO.	X	Y
1	1	0.0013	20	0.025	-0.0195
2	0.95	0.0147	21	0.05	-0.0249
3	0.9	0.0271	22	0.075	-0.0274
4	0.8	0.0489	23	0.1	-0.0286
5	0.7	0.0669	24	0.15	-0.0288
6	0.6	0.0814	25	0.2	-0.0274
7	0.5	0.0919	26	0.25	-0.025
8	0.4	0.098	27	0.3	-0.0226
9	0.3	0.0976	28	0.4	-0.018
10	0.25	0.0941	29	0.5	-0.014
11	0.2	0.088	30	0.6	-0.01
12	0.15	0.0789	31	0.7	-0.0065
13	0.1	0.0659	32	0.8	-0.0039
14	0.075	0.0576	33	0.9	-0.0022
15	0.05	0.0473	34	0.95	-0.0016
16	0.025	0.0339	35	1	-0.0013
17	0.0125	0.0244	36	0.025	-0.0195
18	0	0	37	0.05	-0.0249
19	0.0125	-0.0143	38	0.075	-0.0274

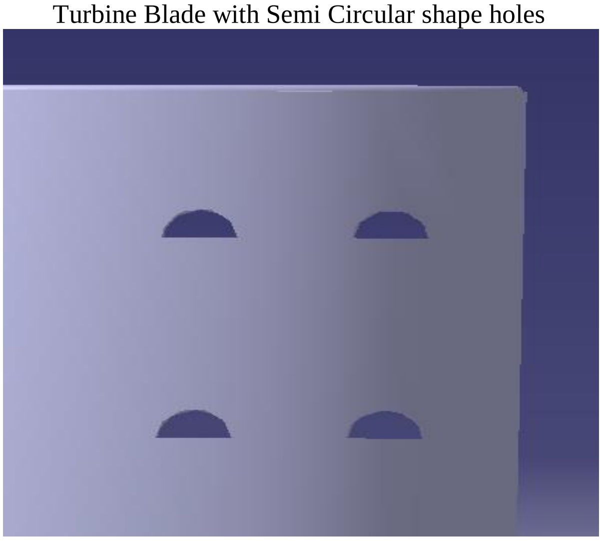




Turbine Blade with Circular shape holes



Turbine Blade with Semi Circular shape holes



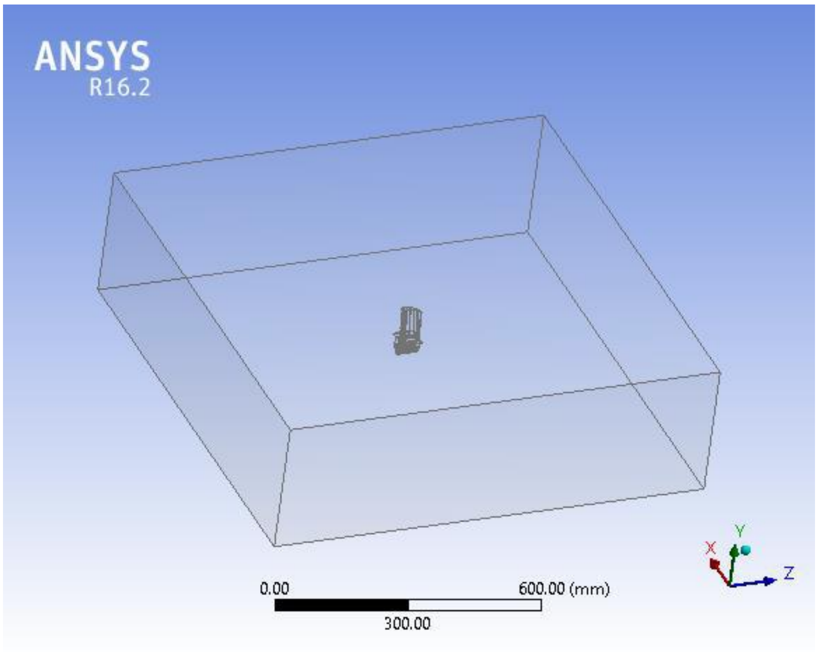
Computational Analysis:

ANSYS CFX software is a high-performance, general purpose fluid dynamics program that has been applied to solve wide-ranging fluid flow problems for over 20 years. ANSYS CFX is more than just a powerful CFD code. Integration into the ANSYS Workbench platform provides superior-directional connections to all major CAD systems, powerful geometry modification and creation tools with ANSYS Design Modeler, advanced meshing technologies in ANSYS Meshing, and easy drag-and-drop transfer of data and results to share between applications. For example, a fluid flow solution can be used in the definition of a boundary load of a subsequent structural mechanics simulation. A native two-way connection to ANSYS structural mechanics products allows capture of even the most complex fluid–structure interaction (FSI) problems in the same easy-to-use environment, saving the need to purchase, administer or run third-party coupling software. For more than 20 years, companies around the world have trusted ANSYS CFX technology to provide reliable and powerful computational fluid dynamics (CFD) solutions. ANSYS CFX combines advanced solver technology with a modern user interface and an adaptive architecture to make CFD accessible to both designers with general engineering knowledge and fluid dynamics specialists requiring in-depth model control and options. It is used in a vast array of industries to provide detailed insight into equipment and processes that increase efficiency, improve product longevity and optimize processes. The CFD analysis is based on the basic aerodynamic equations like energy equation, momentum equation and continuity equation.

ANSYS CFX software is fully integrated in to the ANSYS Workbench environment, the framework for the full suite of engineering simulation solutions from ANSYS. Its adaptive architecture enables users to easily set up anything from standard fluid flow analyses to complex interacting systems with simple drag-and-drop operations. Users can easily assess performance at multiple design points or compare several alternative designs. Within the ANSYS Workbench environment, applications from multiple simulation disciplines can access tools common to all, such as geometry and meshing tools. The step-by-step procedure undertaken in CFX is explained and demonstrated in detail below.

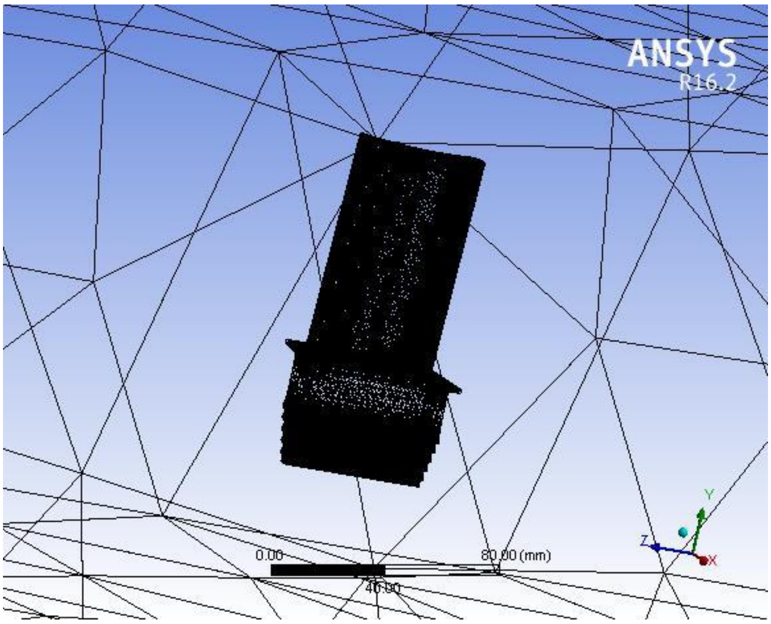
Geometry:

ANSYS Design Modeller software is specifically designed for the creation and preparation of geometry for simulation. It's easy-to-use, fully parametric environment with direct, bidirectional links to all leading CAD packages acts as the geometry portal for all ANSYS products to provide a consistent geometry source for all engineering simulations



Mesh:

Meshing is the process of dividing the whole component into number of elements so that whenever the load is applied on the component it distributes the load uniformly. To start meshing double click on mesh option that is below the geometry option as shown in the figure. After opening the meshing, you will find project tree in which there is an option called mesh. Click on the mesh so that you will find details of mesh at the bottom as shown in figure 4.4 in which the sizing values can be inputted.

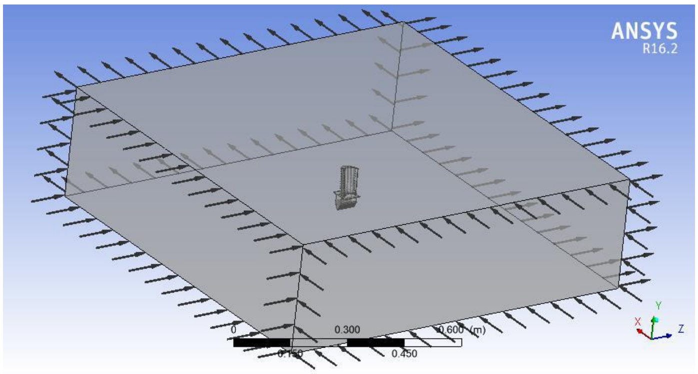


Object Name	Mesh
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	CFX
Relevance	-80
Sizing	
Use Advanced Size Function	On: Curvature
Relevance Center	Coarse
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (31.440 °)
Min Size	0.50 mm
Max Face Size	Default (160.580 mm)
Max Size	Default (321.160 mm)
Growth Rate	Default (1.2080)
Minimum Edge Length	5.e-002 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0.77
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	No
Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Shape Checking	CFD
Element Midside Nodes	Dropped
Straight Sided Elements	
Number of Retries	0
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Default (0.450 mm)
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default (0.250 mm)
Statistics	
Nodes	199168
Elements	1042587

Setup:

The ANSYS CFX physics pre-processor is a modern and intuitive interface for the setup of CFD analyses. In addition to a general mode of operation, predefined wizards are available to guide users through the setup of common fluid flow simulations. A powerful expression language gives users the ability to customize their problem definition in numerous ways, such as with complex boundary conditions, proprietary material models or additional transport equations. The adaptive architecture of CFX-Pre even allows users to create their own custom GUI panels to standardize input for selected applications, and thereby ensure adherence to established best practices. In this context as mentioned earlier the boundary conditions are defined. The material was selected as ideal gas and the initial pressure conditions were assigned to 0 atm. The basic domain physics chart is given aside. The boundary conditions were all changed to our accordance and this is briefed in the below given table.

Context, but the result is extracted most probably when the solution converges. The solution was taken on four grounds and based on this the result was extracted.



Domain - Default Domain	
Type	Fluid
Location	B562
Materials	
Air at 25 C	
Fluid Definition	Material Library
Morphology	Continuous Fluid
Settings	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Thermal Energy
Turbulence Model	k epsilon
Turbulent Wall Functions	Scalable

Table 4. Boundary Physics for CFX

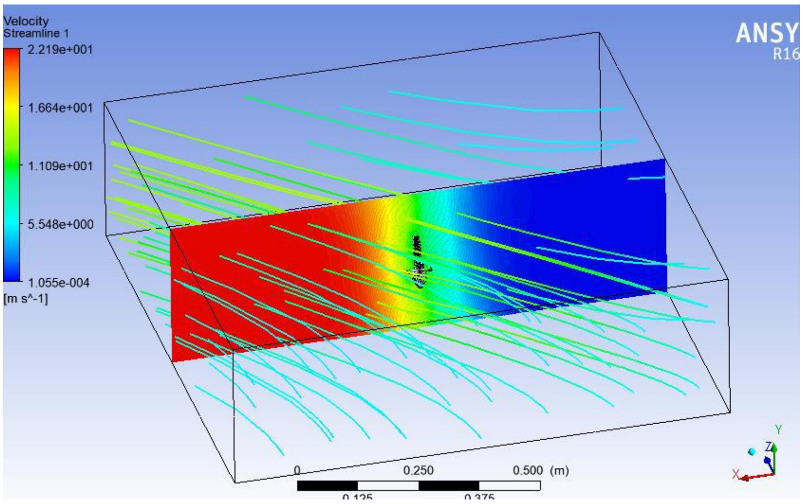
Domain	**Boundaries**	
	Boundary - Hot air inlet	
	Type	INLET
	Location	F564.562
	Settings	
	Flow Regime	Subsonic

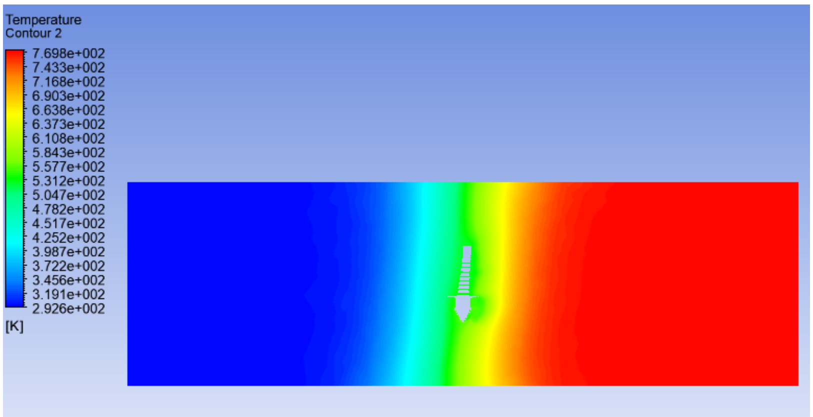
Default Domain	Heat Transfer	Static Temperature
	Static Temperature	5.0000e+02 [C]
	Mass And Momentum	Normal Speed
	Normal Speed	5.0000e+00 [m s^-1]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary - cool air inlet	
	Type	INLET
	Location	F566.562
	Settings	
	Flow Regime	Subsonic
	Heat Transfer	Static Temperature
	Static Temperature	2.0000e+01 [C]
	Mass And Momentum	Normal Speed
	Normal Speed	5.0000e+00 [m s^-1]
	Turbulence	Medium Intensity and Eddy Viscosity Ratio
	Boundary - cool outlet	
	Type	OUTLET
	Location	F563.562

	Settings		
	Flow Regime	Subsonic	
	Mass And Momentum	Average Static Pressure	
	Pressure Profile Blend	5.0000e-02	
	Relative Pressure	0.0000e+00 [kPa]	
	Pressure Averaging	Average Over Whole Outlet	
	Boundary - outlet		
	Type	OUTLET	
	Location	F568.562	
	Settings		
	Flow Regime	Subsonic	
	Mass And Momentum	Average Static Pressure	
	Pressure Profile Blend	5.0000e-02	
	Relative Pressure	0.0000e+00 [kPa]	
	Pressure Averaging	Average Over Whole Outlet	
	Boundary - Default Domain Default		
	Type	WALL	
	Settings		

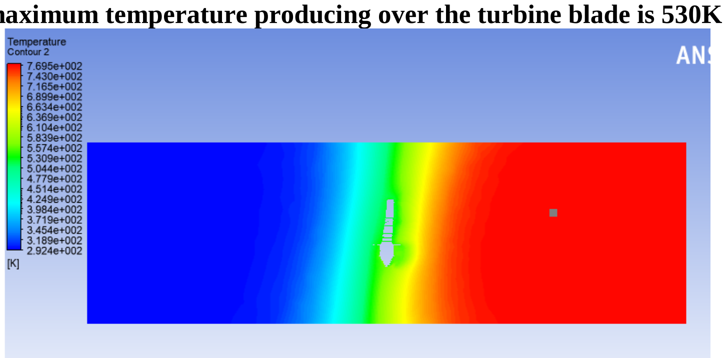
	Heat Transfer	Adiabatic
	Mass And Momentum	No Slip Wall

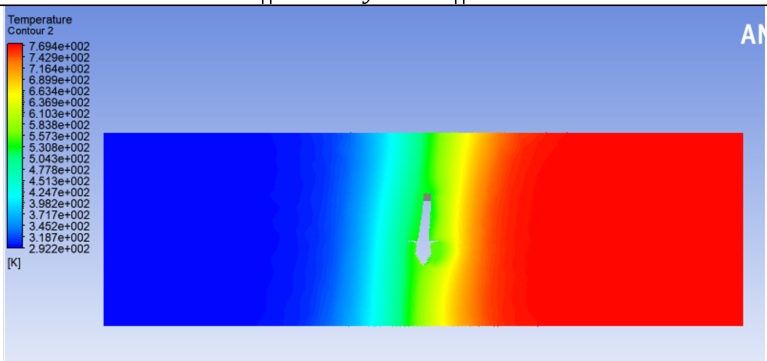
Turbine blade cooling over Circular section:
Velocity Flow over Turbine blade

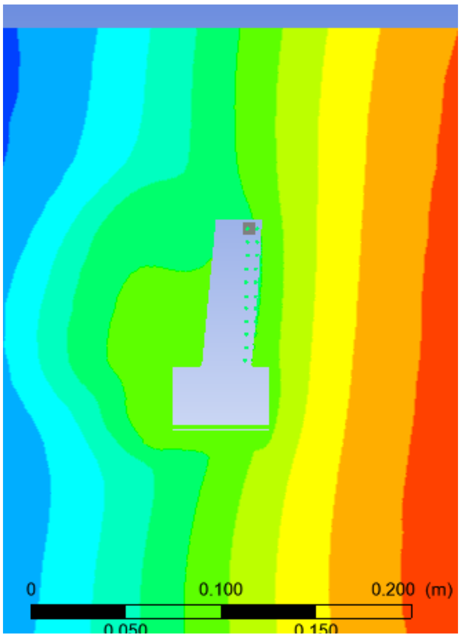




By this problem the maximum temperature producing over the turbine blade is 530K







Semi-circular Shapes holes

Temperature at semi-circular shapes Holes 526K

Finite Element Method

The finite element method (FEM) has now become a very important tool of engineering analysis. Its versality is reflected in its popularity among engineers and designers belonging to nearly all the engineering disciplines. Whether a civil engineer designing bridges, dams or a mechanical engineers designing auto engines, rolling mills, machine tools or an aerospace engineer interested in the analysis of dynamics of an aero plane or temperature rise in the heat shield of a space shuttle or a metallurgist concerned about the influence of a rolling operation on the microstructure of a rolled product or an electrical engineer interested in analysis of the electromagnetic field in electrical machinery-all find the finite element method handy and useful[9]. It is not that these problems remained unproved before the finite element method came into vogue; rather this method has become popular due to its relative simplicity of approach and accuracy of results. Traditional methods of engineering analysis, while attempting to solve an engineering problem mathematically, always try for simplified formulation in order to overcome the various complexities involved in exact mathematical formulation. In the modern technological Environment the conventional methodology of design cannot compete with the modern trends of Computer Aided Engineering (CAE) techniques [10]. The constant search for new innovative design in the engineering field is a common trend. To build highly optimized product, this is the basic requirement of today for survival in the global market. All round efforts were put forward in this direction. Software professional and technologists have developed various design packages.

Analysis in FEM

The finite element method is a numerical analysis technique for obtaining approximate solution to a wide variety of engineering problems. In engineering problems there are some basic unknowns. If they are found, the behaviour of the entire structure can be predicted. The basic unknowns or the field variable which are encountered in the engineering problems are displacement in solid mechanics. The finite procedure reduces such unknowns to a finite number by dividing the solution region into small part called element as shown in figure 6 and by expressing the unknown field variable in terms of assumed approximating functions within each element. The approximating functions are defined in terms of field variable specified called nodes or nodal point. Thus, in the finite element analysis the unknowns are field variables of the nodal points. Once this are found the field variable at any point can be found by using interpolation functions. The various step involved in the finite element analysis are

- Select suitable field variables and the elements.
- Discretize the continua.
- Select the interpolation function.
- Find the element properties.
- Assemble element properties to get global properties.
- Impose the boundary conditions.
- Solve the system equations to get the nodal unknowns.
- Make the additional calculation to get the required values [10], [11].

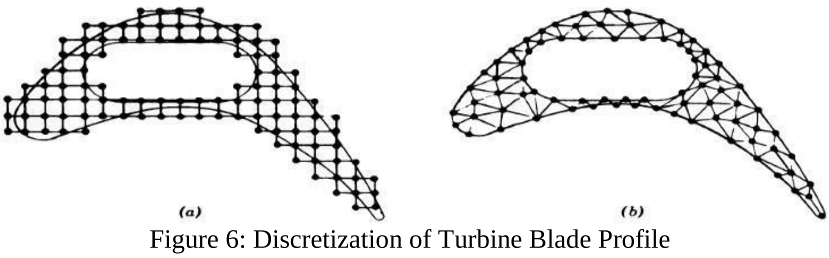


Figure 6: Discretization of Turbine Blade Profile

Element Types

The ANSYS element library contains more than 150 different element types. Each element type has a unique and prefix that identifies the element category: BEAM4, PLANE77, SOLID96, etc. The element type determines, among other things the degree of freedom set (which in turn implies the discipline-structural, thermal, magnetic, electric, quadrilateral, brick, etc.,).

Solid 185 3D 8-Nodes Structural Solid Element:

SOLID185 is used for 3-D modelling of solid structures. It is defined by eight nodes having three degrees of freedom at each node as shown below in figure 7, translations in the nodal x, y, and z directions. The element has plasticity, hyper elasticity, stress stiffening, creep, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials.

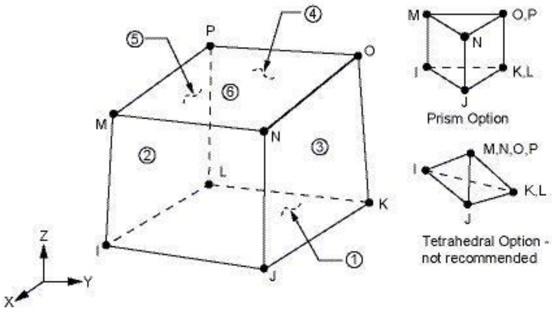


Fig 6: Solid 185 Structural Solid Geometry

Element name Solid 185
Nodes I, J, K, L, M, N, O, P
Degrees of Freedom UX, UY, UZ
Real Constants None

Material Properties EX, EY, EZ, PRXY, PRYZ, PRXZ (or NUXY, NUYZ, NUXZ), ALPX, ALPY, ALPZ (or CTEX, CTEY, CTEZ or THSX, THSY, THSZ), DENS, GXY, GYZ, GXZ, DAMP

Surface Loads Pressure
Body Loads Temperature

Special Features Plasticity, Hyper elasticity, Viscoelasticity, viscoplasticity, Elasticity, Other material, Stress stiffening, large deflection, large strain, Initial stress import and Nonlinear stabilization [14].

Convective Heat Transfer Coefficients over the Blade Surfaces:
The flows over suction and pressure side of rotor blade as shown in figure 11.

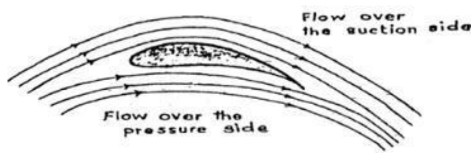


Figure: Gas Flows over Suction and Pressure Side of Rotor Blade

Convective Heat Transfer Coefficients on Suction side of Rotor Blades hs =379.92 w/m2 k. Convective Heat Transfer Coefficients on the Pressure side of rotor blade hp = 284.95 w/m2k [1], [2],[15].

Evaluation of Convective Heat Transfer Coefficient (hr)

Convective Heat Transfer Coefficient (hr) on the Two Rectangular Faces at inlet and Exit of Rotor Blades as shown in figure 12.

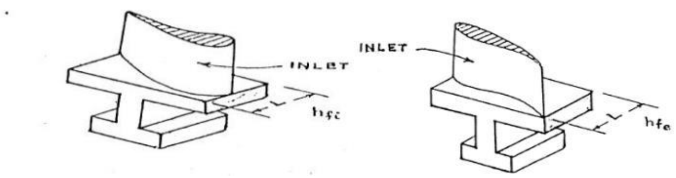


Fig 7: Inlet and Exit of the Rotor Blade

Convective heat transfer coefficients on the rectangular face at inlet hfi = 231.195 w/m2 K.
Convective heat transfer coefficients on the rectangular face at exist hfe = 224.73 w/m2K [1], [2],[15].

Structural Analysis of a Gas Turbine Rotor Blade

Element Type 1: Solid 185 3D 8-nodes Structural Solid Element
Element type 2: Solid70 3D 8-nodes Thermal Solid Element
Young's Modulus of Elasticity (E)
Poisson ratio (μ)
Density (ρ)
Coefficient of thermal expansion (α)

The aero foil profile of the rotor blade was generated on the XY plane with the help of key points defined by the coordinates as shows in table 2. Then a number of splines were fitted through the key points. A rectangle of dimensions 49*27 mm was generated.

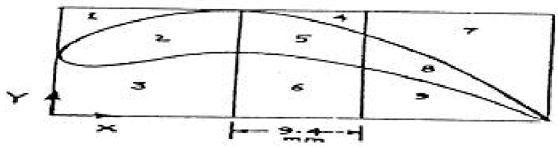


Fig 8: Boundary of Aero Foil Section

Using splines and lines 9 different areas were generated as volumes. In the shape and size option, the number of element edges along the lines surrounding the areas 1 to 9 was specified. In the attribute option element type 1 and material type 1 were assigned to the two areas. Areas 1 to m9 were extruded upwards in the positive Z direction through a height of 5mm. After extrusion, the rectangular block as shown as in figure 14was generated.

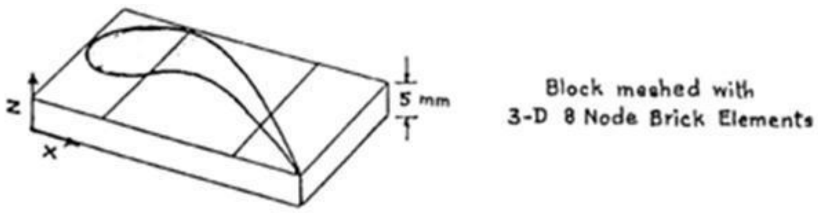


Fig 9: Turbine Rotor Block

Again, using the extrusion option, the shaded area as shown in figure 15 was extruded upwards through the blade height (117 mm) along the positive Z direction, Areas 4,5,6 were extruded downwards along the negative direction through a distance 14.5 mm. The model was generated as shown in figure15 by use in the pre-processor of ANSYS12.0.

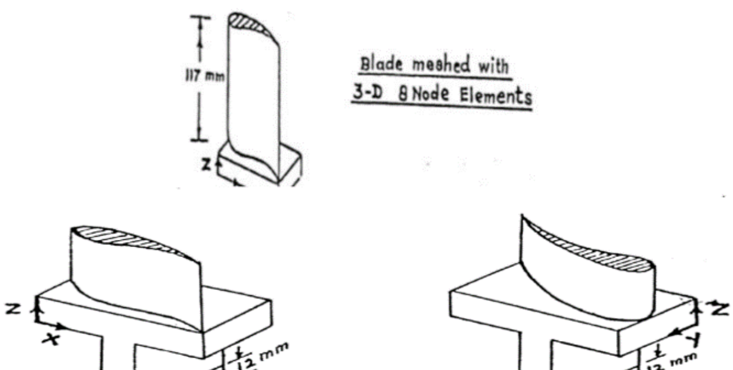


Fig 10: Volume of Rotor Blade

The shaded areas shown below in figure 16 were extruded along the X-direction through a distance of 3.8 mm using the mesh option all the areas were meshed with Brick 8-node 185 element as shown in figure 23[14].

structural Boundary Conditions to Be Applied on the Rotor Blade Model

Two structural boundary conditions namely displacement and force were applied on the rotor blade model as shown in figure 17.

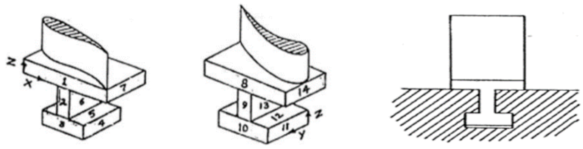


Fig 11: Structural Boundary Conditions on Rotor Blade

Ux = 0 for areas 4,5,6,7 and 11,12,13,14
Uy = 0 for areas 1, 2, 3 and 8,9,10
Uz = 0 for areas 5 and 12
U represents displacement and suffix X, Y; Z represents the direction in which the displacement was constrained.

In the solution part of Ansys the blade forces namely tangential, axial and centrifugal were applied on the node located at the centroid of the blade as shown in figure 10.

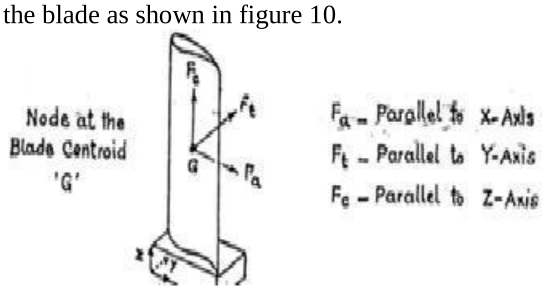
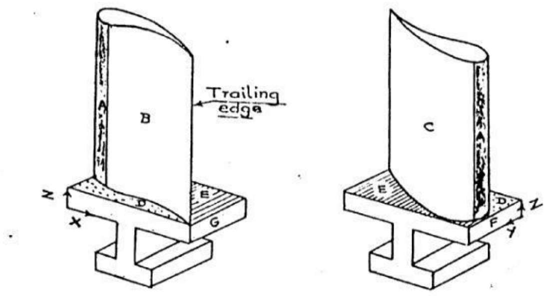


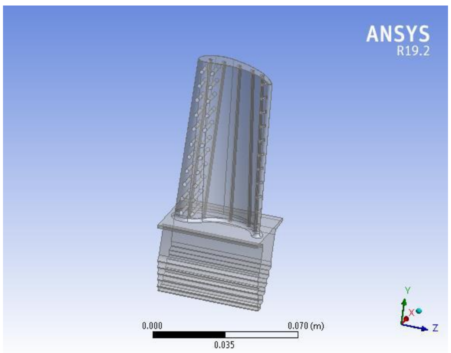
Fig 12: Static Loading on Rotor Blade

Thermal Boundary Conditions Applied on the Rotor Blade Model:

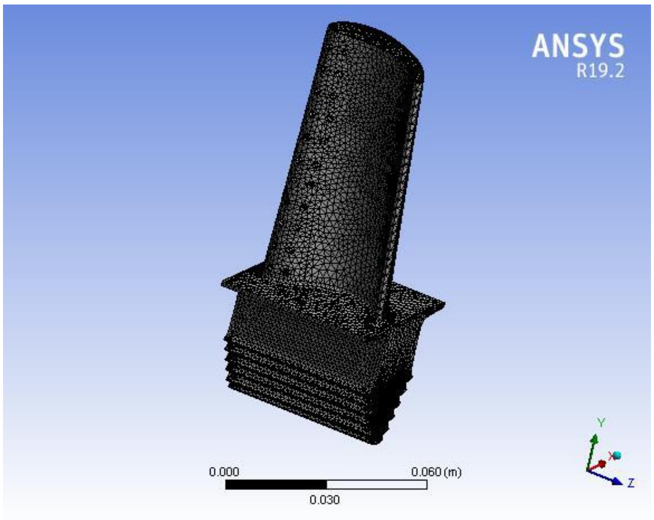
Heat flux = 0 from areas 1, 2… to 16. Areas 1, 2, 3 and 8,9,10 come in contact with similar areas on the adjacent rotor blades as shown in figure 17. Hence due to symmetry boundary conditions, these areas are assumed to be insulated. Areas 4, 5, 6 and 11,12,13,15 on account of their small dimensions are assumed to be insulated. In the convective boundary condition, the convective heat transfer coefficient (h) and temperature of surrounding gases (T) have to be specified on the areas subjected to convection as shown below in figure 19.



Thermal Analysis
Meshing



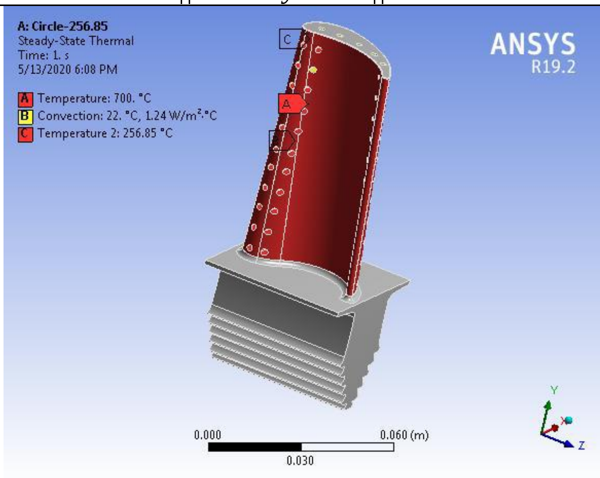
Density	4620 kg m^-3
Isotropic Secant Coefficient of Thermal Expansion	9.4e-006 C^-1
Specific Heat Constant Pressure	522 J kg^-1 C^-1
Isotropic Thermal Conductivity	21.9 W m^-1 C^-1
Isotropic Resistivity	1.7e-006 ohm m



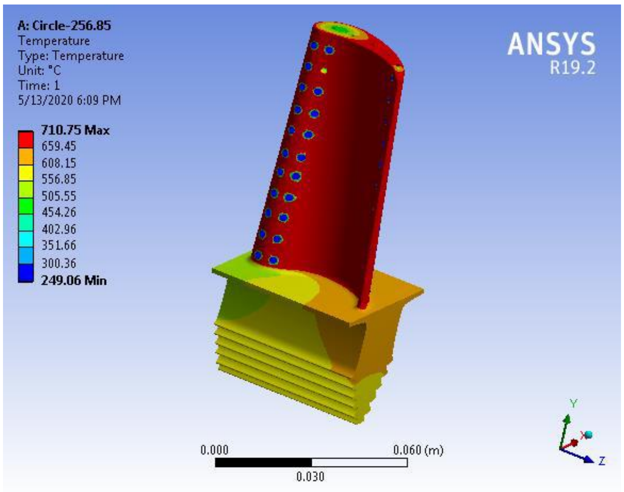
Object Name	***Mesh***
State	Solved
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Solver Preference	Program Controlled
Relevance	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	7
Mesh Defeaturing	Yes
Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Diagonal	0.13756 m
Average Surface Area	1.2627e-004 m²
Minimum Edge Length	1.2754e-005 m
Quality	

Check Mesh Quality	Yes, Errors	
Error Limits	Standard Mechanical	
Target Quality	Default (0.050000)	
Smoothing	Medium	
Mesh Metric	None	
Inflation		
Use Automatic Inflation	None	**RES**
Inflation Option	Smooth Transition	
Transition Ratio	0.272	
Maximum Layers	5	
Growth Rate	1.2	
Inflation Algorithm	Pre	
View Advanced Options	No	
Advanced		
Number of CPUs for Parallel Part Meshing	Program Controlled	
Straight Sided Elements	No	Due mini
Number of Retries	Default (4)	
Rigid Body Behavior	DimensionallyReduced	Min i
Triangle Surface Mesher	Program Controlled	Ma x
Topology Checking	No	**Semi**
Pinch Tolerance	Please Define	
Generate Pinch on Refresh	No	
Statistics		
Nodes	286643	
Elements	182283	

Boundary Condition:
The circular and semi, circular, semi elliptical hole temperature applied which got from Fluid analysis.



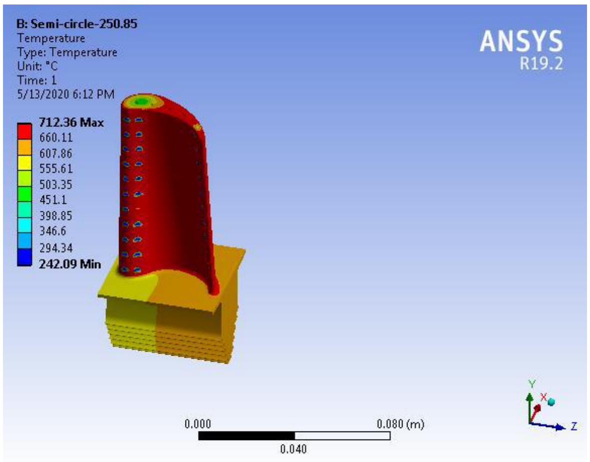
Result of Circular:



Due to circle hole the the temperature of minimum temperature is 249

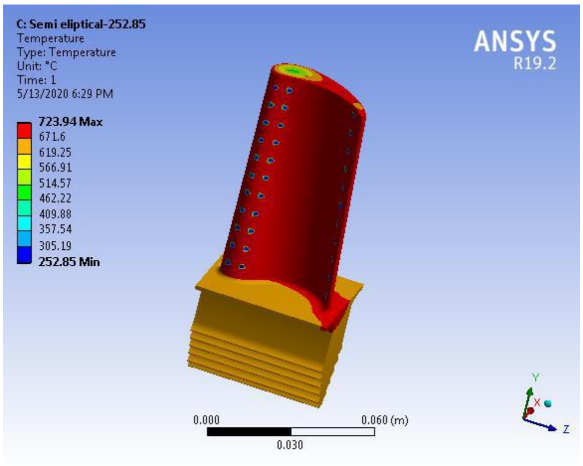
Minimum	249.06 °C
Maximum	710.75 °C

Semi-circular:



Minimum	242.09 °C
Maximum	712.36 °C

Semi elliptical: Minimum



Minimum	252.85 °C
Maximum	723.94 °C

Conclusion:

In this project using finite element analysis as a tool, the fluid and thermal analysis is carried out sequentially. The blade with different shape were used for analysis. The gas turbine blade is modelled in a 3D cad tool called CATIA V5-6R2017 by using extrude feature. Then gas turbine blade with different shape (circular, semi cylinder, semi elliptical has been modelled on the blade span. The blade with different shape were used for Fluid analysis in Ansys-CFX simulation tool. It is observed that as different shape holes changing the temperature distribution and observe the temperature changes for all shapes and which is applied into thermal analysis. The thermal analysis is carried out in ANSYS steady state thermal analysis TOOL. It is observed that blade with semi cercal has showing more amount of reducing temperature in inner part of section which means of 242 degrees Celsius. this shapes only having low amount of minimum temperature. Compare with other shapes.

Reference

[1]. S. Gowreesh, N. Sreenivasalu Reddy and N.V.Yogananda Murthy CONVECTIVE HEAT TRANSFER ANALYSIS OF A AEROGAS TURBINE BLADE USING ANSYS", International journal of Mechanics of solids, vol4, No.1, March 2009(ppt55 -62).

[2]. P.Kauthalkar, Mr.DevendraS.Shikarwar, and Dr.Pushapendra Kumar Sharma. "ANLYSIS OF THERMAL STRESES DISTRIBUTION PATTERN ON GAS TURBINE BLADE USING ANSYS", International journal of Engineering Education and technology, Vol.2, No.3, Nov 2010.

[3]. John.V, T.Ramakrishna. "THE DESIGN ANDANALYSIS OF GAS TURBINE BLADE", International Journal of Advanced Research and Studies, Vol 2, No.1, Dec 2012.

[4]. V.RagaDeepu, R.P.KumarRopichrla."DESIGN AND COUPLED FIELDANALYSIS OF FIRST STAGE GASTURBINE ROTOR BLADES", International journal of Mathematics and Engineering, Vol13, No.2, Pages: 1603 -1612.

[5]. Moyroud, F., Fransson, T. and Jacquet-Richardet, G. 2002. A comparison of two finite element reduction techniques for mistuned bladed-disks, Journal of Engineering for Gas Turbines and Power. 124: 942-953.

[6]. Giovanni, C., Ambra, G., Lorenzo, B. and Roberto, F. 2007 Advances in effusive cooling techniques of gas turbines, Applied Thermal Engineering. 27: 692-698.

[7]. Cun-liang, L., Hui-ren, Z., Jiang-tao, B. and Du-chun, X. 2010. Film cooling performance of converging slot-hole rows on a gas turbine blade, International Journal of Heat and Mass Transfer. 53: 5232-5241.

[8]. Zhang, JJ.,Esat, II. and Shi, YH. 1999. Load Analysis with Varying Mesh Stiffness, Computers and Structures. 70: 273-280.

[9]. Hildebrabd, FB. 1997. Introduction to Numerical Analysis, McGraw-Hill, New York.

[10]. MoussaviTorshizi, SE.,YadavarNikravesh, SM. and Jahangiri, A. 2009. Failure analysis of gas turbine generator cooling fan blades, Engineering Failure Analysis. 16: 1686-1695.

[11]. Cleeton, JPE., Kavanagh, RM. and Parks, GT. 2009. Blade cooling optimisation in humid-air and steam-injected gas turbines, Applied Thermal Engineering. 29: 3274-3283.