

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Thermal Analysis of Gas Turbine Blade

CH.Rami Reddy¹, K.Chendrasekhar²

M.Tech Student, QIS College of Engineering and Technology, Ongole, Andhra Pradesh, India,

Associate Professor Department of Mechanical engineering, QIS College of Engg & Technology Ongole, A.P, India

ABSTRACT: A turbine blade is the individual component which makes up the turbine section of a gas turbine or steam turbine. The blades are responsible for extracting energy from the high temperature, high pressure gas produced by the combustor. The turbine blades are often the limiting component of gas turbines. In general all solid and nonsolid model will deform when certain amount of thermal or structural loads applied within the environmental condition. In order to find the changes of the product or component, a analysis software is used. Ansys is an analytic software to find changes in deformation, Product life, Failures, heat flux(change of heat flow with respect to time and distance) and CFD (flow of air or water or any gas or liquid in the body).In this project the model is Designed with respect to all the available constraints using an advanced cad software's like catia . incatia all the individual parts are created in part module and assemble each other in assemble module. Later the product file is converted to ".stp" file format (standard exchange of product file) and imported to ansys workbench to find deformation and analytic value with respect to the model or product definitions. Using static structural analysis total deformation, stress and strain value, product life and failures etc., can be calculate by using Ansys workbench. In Thermal analysis is heat flux is calculated with respect to temperature changes, heat flow and Convection.

Ansys software helps to find the accurate or approximate solutions

I. INTRODUCTION



Figure 1.1 Turbine Blade From A Turbo-Union RB199jet Engine.

A turbine blade is the individual component which makes up the turbine section of a gas turbine or steam turbine. The blades are responsible for extracting energy from the high temperature, high pressure gas produced by the combustor. The turbine blades are often the limiting component of gas turbines.[1] To survive in this difficult environment, turbine blades often use exotic materials like superalloys and many different methods of cooling, such as internal air channels, boundary layer cooling, and thermal barrier coatings. Blade fatigue is a major source of failure in steam turbines and gas turbines. Fatigue is caused by the stress induced by vibration and resonance within the operating range of machinery. To protect blades from these high dynamic stresses, friction dampers are used.[2] Blades of wind turbines and water turbines are designed to operate in different conditions, which typically involve lower rotational speeds and temperatures.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

1.1. Introduction

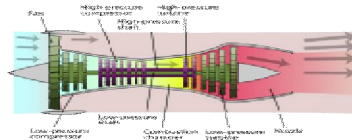


Figure 1.1 Diagram Of A Twin Spool Jet Engine.

The high-pressure turbine is connected by a single spool to the high-pressure compressor (purple)- and the low-pressure turbine is connected to the low-pressure compressor by a second spool (green). In a gas turbine engine, a single turbine section is made up of a disk or hub that holds many turbine blades. That turbine section is connected to a compressor section via a shaft (or "spool"), and that compressor section can either be axial or centrifugal. Air is compressed, raising the pressure and temperature, through the compressor stages of the engine. The temperature is then greatly increased by combustion of fuel inside the combustor, which sits between the compressor stages and the turbine stages. The high-temperature and high-pressure exhaust gases then pass through the turbine stages. The turbine stages extract energy from this flow, lowering the pressure and temperature of the air and transfer the kinetic energy to the compressor stages along the spool. This process is very similar to how an axial compressor works, only in reverse.^[3]

The number of turbine stages varies in different types of engines, with high-bypass-ratio engines tending to have the most turbine stages.^[citation needed] The number of turbine stages can have a great effect on how the turbine blades are designed for each stage. Many gas turbine engines are twin-spool designs, meaning that there is a high-pressure spool and a low-pressure spool. Other gas turbines use three spools, adding an intermediate-pressure spool between the high- and low-pressure spool. The high-pressure turbine is exposed to the hottest, highest-pressure air, and the low-pressure turbine is subjected to cooler, lower-pressure air. The difference in conditions leads to the design of high-pressure and low-pressure turbine blades that are significantly different in material and cooling choices even though the aerodynamic and thermodynamic principles are the same.^[4] Under these severe operating conditions inside the gas and steam turbines, the blades face high temperature, high stresses, and potentially high vibrations. Steam turbine blades are critical components in power plants which convert the linear motion of high-temperature and high-pressure steam flowing down a pressure gradient into a rotary motion of the turbine shaft.^[5]

1.2. Materials^[edit]

A key limiting factor in early jet engines was the performance of the materials available for the hot section (combustor and turbine) of the engine. The need for better materials spurred much research in the field of alloys and manufacturing techniques, and that research resulted in a long list of new materials and methods that make modern gas turbines possible

^[8] One of the earliest of these was Nimonic, used in the British Whittle engines. The development of superalloys in the 1940s and new processing methods such as vacuum induction melting in the 1950s greatly increased the temperature capability of turbine blades. Further processing methods like hot isostatic pressing improved the alloys used for turbine blades and increased turbine blade performance.^[8] Modern turbine blades often use nickel-based superalloys that incorporate chromium, cobalt, and rhodium.^{[6][10]}

Aside from alloy improvements, a major breakthrough was the development of directional solidification (DS) and single crystal (SC) production methods. These methods help greatly increase strength against fatigue and creep by aligning grain boundaries in one direction (DS) or by eliminating grain boundaries altogether (SC).^[8]

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

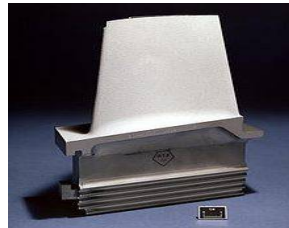


Figure 1.2 A Turbine Blade With Thermal Barrier Coating.

Another major improvement to turbine blade material technology was the development of thermal barrier coatings (TBC). Where DS and SC developments improved creep and fatigue resistance, TBCs improved corrosion and oxidation resistance, both of which became greater concerns as temperatures increased. The first TBCs, applied in the 1970s, were aluminide coatings. Improved ceramic coatings became available in the 1980s. These coatings increased turbine blade temperature capability by about 200 °F (90 °C).^[8] The coatings also improve blade life, almost doubling the life of turbine blades in some cases.^[11]

Most turbine blades are manufactured by investment casting (or lost-wax processing). This process involves making a precise negative die of the blade shape that is filled with wax to form the blade shape. If the blade is hollow (i.e., it has internal cooling passages), a ceramic core in the shape of the passage is inserted into the middle. The wax blade is coated with a heat-resistant material to make a shell, and then that shell is filled with the blade alloy. This step can be more complicated for DS or SC materials, but the process is similar. If there is a ceramic core in the middle of the blade, it is dissolved in a solution that leaves the blade hollow. The blades are coated with a TBC, and then any cooling holes are machined.^[12]

1.3. Cooling

At a constant pressure ratio, thermal efficiency increases as the maximum temperature increases. But, high temperatures can damage the turbine, as the blades are under large centrifugal stresses and materials are weaker at high temperature. So, turbine blade cooling is essential.^[24]

1.3.1. Methods Of Cooling

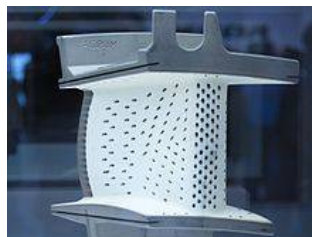


Figure 1.3 Laser-Drilled Holes

Cooling of components can be achieved by air or liquid cooling. Liquid cooling seems to be more attractive because of high specific heat capacity and chances of evaporative cooling but there can be problem of leakage, corrosion, choking, etc. which works against this method.^[24] On the other hand, air cooling allows the discharged air into main flow without any problem. Quantity of air required for this purpose is 1–3% of main flow and blade temperature can be reduced by 200–300 °C.^[24] There are many types of cooling used in gas turbine blades; convection, film, transpiration cooling, cooling effusion, pin fin cooling etc. which fall under the categories of internal and external cooling. While all

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

methods have their differences, they all work by using cooler air (often bled from the compressor) to remove heat from the turbine blades.

II. LITERATURE REVIEW

K. Sornes [1] in his paper have discussed about various types of blade design used in Hydro-Microturbines. The two most common small scale hydro-kinetic turbine concepts are axial flow turbine and cross flow turbines. The axial concept has a rotational axis of rotor which is parallel to the incoming water stream. The cross flow concept on the other hand, has a rotational axis of rotor which is parallel to water surface. The advantage of cross flow turbines over axial flow turbine is that they can rotate uni-directional even with bi-directional flow. Cross flow turbine can be divided into two groups, namely Vertical axis (Axis vertical to water plane) and In-plane axis (Axis on the horizontal plane of water surface). From which we are going to discuss namely six traditional types of blade design.

CROSS FLOW TURBINES

In cross flow turbine the water passes through the turbine transversely, or across the turbine blade. It provides additional efficiency. Cross flow turbines are often constructed as two turbines of different capacity that share the same shaft [2]. The different types of Cross-flow turbines are explained below. In-plane axis J. Khan [3] in his research paper have discussed about In-plane axis. In-plane axis is better known as floating water wheels. These are mainly drag based devices and inherently less efficient than their lift based counter parts. The large amount of material usage is another problem for such turbines. Darrieus turbines with In-plane axis may also fall under this category. But such systems are left common and suffer from bearing and power take off problems.

Squirrel cage Darrieus G. J. M. Darrieus [4] has presented two major types of Darrieus mechanisms. They differ on how they handle the centrifugal force impose on the blade of the turbine, one is called Squirrel cage variant which consists of two disks at top and bottom with the airfoil running straight up and down between their rims. This allows the centrifugal force to be handled by the relatively sturdy construction of the disks. The advantage of turbine is to be able to progressively get into rotation. The disadvantage of it is the low reynold number.

H-Darrieus S. Roy et al [5] in his work have described about H-darrieus turbine. H-Darrieus are breed of vertical axis wind turbine designed by George Darrieus in 1920's. They are capable of producing much power than most typical wind turbine. H-Darrieus rotor is a lift type device having two or three blades designed as airfoils. The blades are attached vertically to centre shaft through support arms. The support to vertical axis helps rotor to maintain its shape. One major disadvantage of H-type Darrieus turbine is that since lift forces drives them that must be brought to a minimum speed before the forces generated as sufficient to propel the turbine. The starting torque coefficient is zero and at low tip speed ratio it is even negative. Therefore, a special motor is required to start the rotor. With increase of height to diameter ratio, velocity magnitude difference from inlet up to rotor increases up to height to diameter ratio 1.0 and then decreases loss of performance for turbine with increases of height to diameter ratio. It can be concluded that velocity difference from inlet up to rotor is responsible for power stroke of blades during its clockwise direction. The Tip Speed Ratio of H-Darrieus turbine is high, hence, it rotate faster [6].

Darrieus L. J. Hagen et al [7] in his research paper has summarized about Darries turbine. This design of turbine was patented by Georges Jean Marie Darrieus, a French aeronautical engineer in 1931. Darrieus turbine is a vertical axis turbine. It has streamlined blades turning around an axis perpendicular to the flow. The turbine consists of number of curved airfoil blades mounted on a vertical rotating shaft or framework. The curvature of the blade allows the blade to be stressed only in tension at high rotating speeds. It is powered by the phenomenon of lift. There are major difficulties in protecting the Darrieus turbine from extreme speed of fluid and in making it self-starting. In Darrieus blades the airfoils are arranged so that they are symmetrical and have zero rigging angle, that is, the angle that the airfoil are set relative to the structure on which they are mounted. This arrangement is equally effective no matter which direction the flow of fluid is flowing in contrast to the conventional type, which must be rotated to face into the fluid. One problem with the design is that the angle of attack changes as the turbine spins, so each blade generates its maximum torque at

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

two points on its cycle, that is, at front and back of the turbine. The Tip Speed Ratio of Darrieus turbine is high approximately same as H-Darrieus, hence, it rotate faster. Its self-starting capabilities are low.

Gorlov A. M. Gorlov [8] have patented information about Gorlov turbine. The Gorlov turbine was invented by Alexander Gorlov in 1995. This turbine is also known as 'Cross flow helical turbine'. It is similar to Darrieus straight blade style turbine, except airfoil blade profile is swept in a helical profile along its span. One of the advantage of helical blade is that it improves self-starting of Gorlov turbine compared to Darrieus turbine. As helical blade sweeps along circumference of rotation of turbine some portion of blade profile is located at optimum angle of attack even in static or slowly rotating conditions, which allows for more uniform starting torque that depend upon turbine azimuthal position. Also owing to helical blade shape is reduction of torque oscillation during rotation. A Darrieus turbine tends to experience torque oscillation resulting from circumferential void space between discrete blade positions. On the other hand Helical turbine with full blade wrap around its circumference does not experience this problem. Uniform blade coverage, neglecting end effects and wake dynamics, ideally give the turbine torque, although in reality some variations are likely to occur. Gorlov reported from experimental testing that maximum efficiency for Gorlov turbine is around 35%.

RECENT TREND IN HYDRO-MICROTURBINES

Darrieus and Savonius Hybrid Turbine There are various modification done in Hydro-microturbines to improve certain parameters such as efficiency, tip speed ratio, self-starting ability, torque etc. For example, hybrid of Darrieus and Savonius turbines are available. Double Savonius which is recently been developed have greater efficiency than Savonius turbines. Darrieus turbine, which is the typical lift-type vertical axis wind turbine, has many advantages over horizontal axis types. In particular, this turbine is suitable for stand-alone power systems on isolated islands and in mountainous regions where the power supply using utility grids is very difficult. Thus, this paper discusses the suitable hybrid configuration of Darrieus lift-type rotor and Savonius drag-type rotor for stand-alone wind turbine-generator systems using our dynamic simulation model [10].

III. INTRODUCTION CATIAV5R20

CATIA is the leading solution for product success. It addresses all manufacturing organizations. CATIA can be applied to a wide variety of industries- from aerospace- automotive- and industrial machinery- to electronics- shipbuilding- plant design- and consumer goods. Today- CATIA is used to design anything from an airplane to jewelry and clothing. With the power and functional range to address the complete product development process- CATIA supports product engineering- from initial specification to product-in-service- in a fully-integrated manner. It facilitates reuse of product design knowledge and shortens development cycles- helping enterprises to accelerate their response to market needs

3.2. Modeling Of Slap And Slider For Friction Surface

3.2.1. A Part 1: Slap

To create go to catia software and click on start – mechanical design and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in fig.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

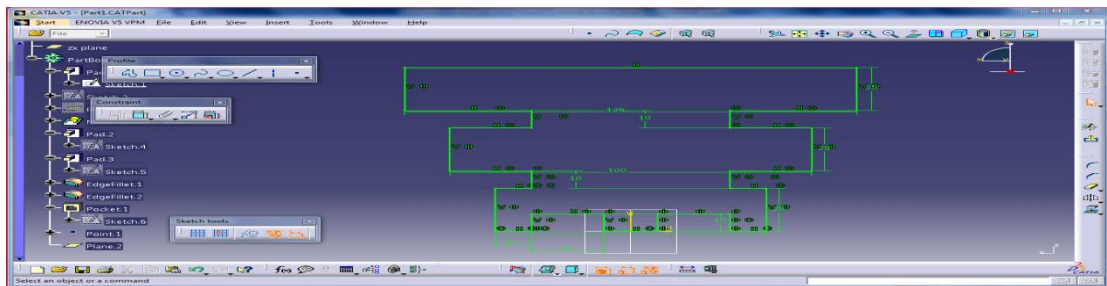


Figure 3.1 Sketcher

After completing sketcher go to workbench and select exit work bench. Then we enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify length as shown in fig.

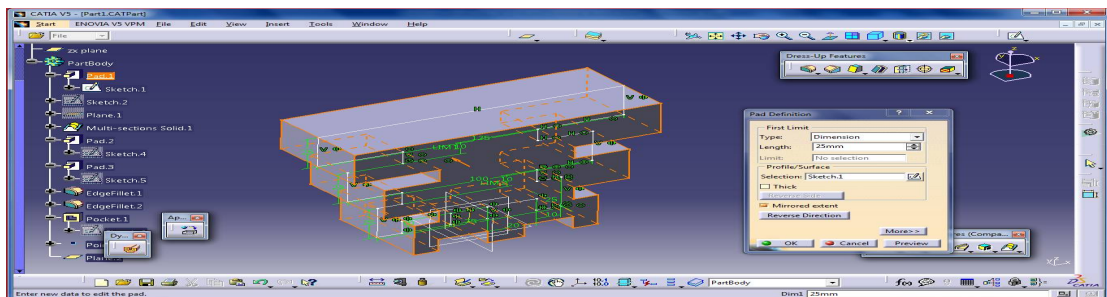


Figure 3.2 Pad Tool For Base Locker

To create go to catia software and click on start – mechanical design and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in fig.

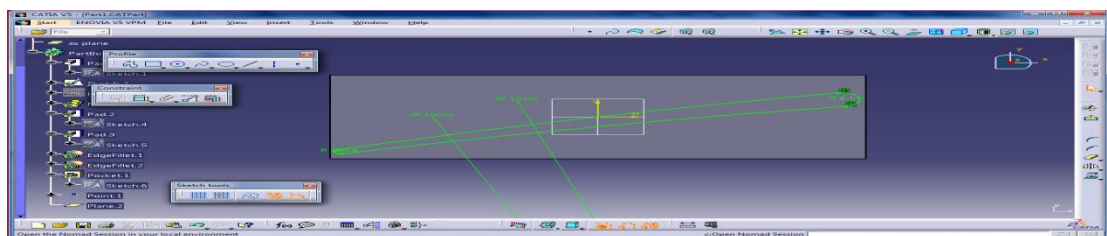


Figure 3.3 Sketcher For Curve Shaft Blade

To create go to catia software and click on start – mechanical design and select part design module. In part design module then go to sketcher tool and select sketcher tool and select required plane. Then we enter into sketcher module and go to profile and create a profile structure as shown in fig.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

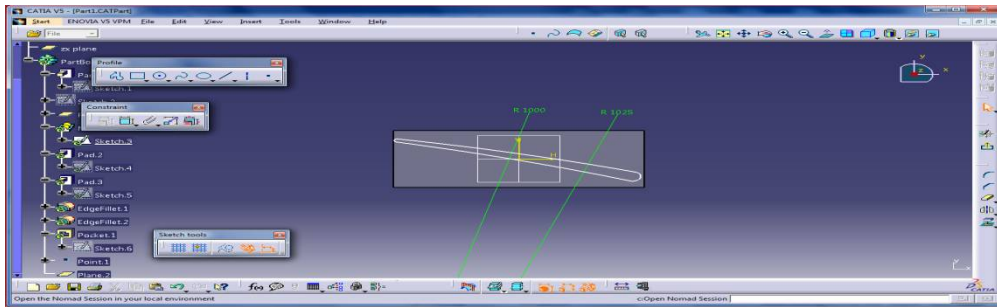


Figure 3.4 Sketcher For Curve Shaft Blade

After completing sketcher go to workbench and select exit work bench. Then we enter into part module again. In part module, go to sketcher based feature and select pad tool. In pad definition, select above sketch as selection profile and specify length as shown in fig.

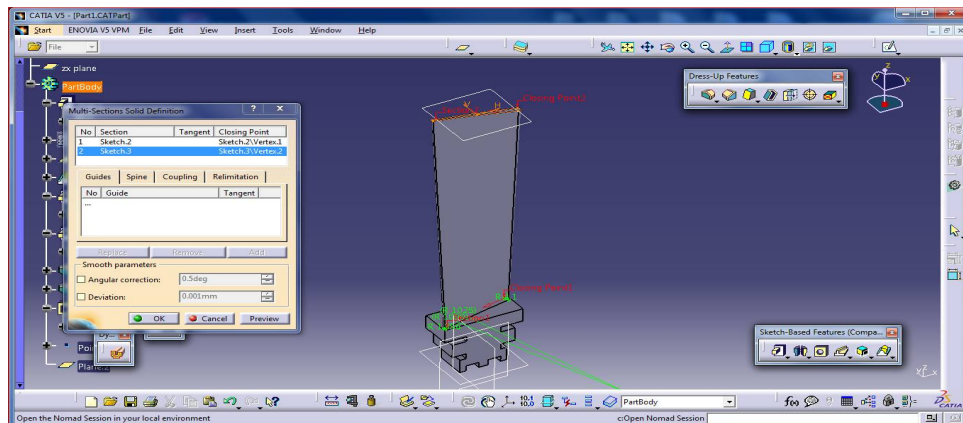


Figure 3.5 Final Shape Of Steam Turbine Blade

IV. INTRODUCTION OF ANSYS

Ansys a product of Ansysinc. Is a world's leading, widely distributed and popular commercial CAE package. It is widely used by designers/analysis in industries such as aerospace, automotive, manufacturing, nuclear, electronics, biomedical, and much more. Ansys provides simulation solution that enables designers to simulate design performance directly on the desktop. In this way, it provides fast efficient and cost efficient product development from design concept stage to performance validation stage of product development cycle.

4.1. Project Objectives

After complete of this chapter,

- The basic concept and general working of Fea
- Understand the advantage and limitations of Fea
- Understanding the analysis type
- Understanding important terms and definition of Fea

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

4.2. Introduction OfFea

The finite element analysis (Fea) is computing technique that is used to obtain approximately. Solution to boundary value problems. It uses a numerical method called finite element method(fem).in Fea involves the computer model of a design that is loaded and analysed for specific results, such as stress, deformation, deflection, natural frequencies, mode shapes, temperatures distribution and soon.

The concept of Fea can be explained through a basic measurement of dimensions. In Fea simulation the loading condition of design and determination the design responses in those conditions. It can be used in new product design as well as in exiting product refinement. A model is divided into a finite number of regions/ divisions call elements, these elements can be of predefinition shapes, such as triangular, quadrilateral, hexahedron, tetrahedron and soon. This predefined shapes(elements) helps to find the sum of responses of all elements in model gives the total responses of complete model

V. INTRODUCTION TO ANSYS WORKBENCH

5.1. Project Objective

In this chapter, we will be able to define:

- Understand the types of system
- Understand different types of cells
- Understand the graphic user interface of the workbench window
- Start a new project in Ansys workbench windows
- Add the first and subsequent analysis system to a project
- Set units for the project

Ansys workbench, developed by Ansysinc., usa, is a computer aided finite element modelling and finite element analysis tool (caem and caFea). In the graphical user interface gui of Ansys workbench the user can generate 3-dimensional and Fea models, perform analysis and generate results of analysis. We can perform a variety of tasks ranging from design assessment to finite element analysis to complete product optimisation analysis by using Ansys workbench. Ansys also enable the combination of standalone analysis system into a project and to manage the project workflow.

In Ansys workbench this are the list of analysis can be determined:

- Modal analysis
- Static structural analysis
- Static structural analysis
- Steady state thermal analysis
- Static thermal analysis
- Fluid flow (CFD)

5.2. Starting Ansys Workbench 16.0

To start Ansys workbench 16.0, choose start- programs/ all programs- Ansys 16.0 - workbench 16.0 from the taskbar. Alternatively, we can start Ansys workbench by double click on the workbench 16.0.

The workbench windows helpstreamline an entire project to be carried out in Ansys workbench 16.0. In this window, one can create, manage, and view the workflow of the entire project create by using standard analysis system. The workbench windows mainly consist of the menu bar, standard toolbar, the toolbar windows, project schematic windows, and the status bar.

VI. STATIC STRUCTURAL ANALYSIS

6.1. Project Objective

In this project, we will be able to define total deformation and stress, etc

- Create the static structural analysis system
- Apply different types of materials

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

- Applying of boundary conditions
- Apply a different type of constraints
- Apply different loads
- Generate the results as per required
- Generate project reports

In this project, we imported the geometry of the component show the dimensions for the component with respect to the load applications. The material to be applied on the model is stainless steel. Next, you will run the analysis under two conditions and evaluate the total deformation, directional deformation, equivalent stress, maximum principal stress, and minimum principal stress.

The static structural analysis is one of the important analyses in Ansys workbench. It is available as static structural analysis system under the analysis system toolbox in the toolbox window, this system analyses the structural components for displacements (deformation), stresses, strains, and forces under different loading conditions. The loads in this analysis system are assumed not to have damping characteristics (time dependent). Steady loading and damping conditions are assumed in this type of analysis system.

6.1.1. Pre-Processing

The pre-processing of an analysis system involves specifying the material, generating a mesh, and defining boundary conditions.

In Ansys workbench, the various tools related to boundary conditions are available in the environment contextual toolbar, which is displayed when you select the static structural node in the tree outline. In order to provide a support to the model, you need to choose the required tool from the supports drop-down. Similarly, to add a load, choose the desired tool from the loads drop-down in the environment contextual toolbar. Also, when you choose any tool from the environment contextual toolbar; the corresponding entity is placed under the static structural node in the tree outline.

The main purpose of an analysis is to evaluate the results. After the boundary condition is set and loads are applied, you need to specify the desired outcomes of the analysis. In Ansys workbench, you can analyze various parameters such as deformation, stresses, strains, and so on. To do so, you need to specify the results required and then evaluate them. You can use the tools available in the solution contextual toolbar to specify results, refer to figure 9-4. Alternatively, right-click on the solution node in the tree outline and then use the desired option from the shortcut menu displayed.

In order to evaluate deformations, stresses, strains, and so on, choose the desired options from the drop-downs available in the solution contextual toolbar.

6.1.2. Solution

In an analysis, after pre-processing (meshing, specifying material, and specifying boundary condition) is done, the next step is to solve the analysis. In Ansys workbench, you will use the solve tool from the standard toolbar to run the solver. The solver runs in the background of a software and acquires results of an analysis, based on the specified boundary conditions.

6.1.3. Post-Processing

After the analysis is complete, you need to generate the report in the mechanical window. To do so, choose the report preview tab from the bottom of the graphics screen; the Ansys report generation in progress message is displayed on the screen. After sometime, this message vanishes and the report is generated.

6.2. Starting A New Project And Creating The Model

The first step is to start a new project in the workbench window.

1. Start Ansys workbench.
2. Choose the save button from the standard toolbar; the save as dialog box is displayed.
3. Double-click on static structural in the toolbox window; the static structural analysis system is added in the project schematic window.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

4. Rename the static structural analysis system (if).
5. Double-click on the geometry cell; the design modeler window along with the Ansys workbench dialog box is displayed.
6. In the Ansys workbench dialog box, set the unit to millimeter. Now, create the model on the xy plane
7. Exit the design modeler window to display the workbench window.

6.3. Adding The Material To The Engineering Data Figure Workspace

After creating the holes in the model, you now need to apply the material to it. The material to be applied is stainless steel.

1. Double-click on the engineering data cell of the spring plate analysis system; the engineering data workspace is displayed in the workbench window.
2. Choose the engineering data sources toggle button from the standard toolbar; the engineering data sources window is added to the engineering data workspace.
3. In the engineering data sources window, select the general materials library to display the outline of general materials window.
4. In the outline of general materials window, choose the plus symbol corresponding to aluminum alloy; the material is added to the engineering data in the outline window of the engineering data workspace.
5. Again, choose the engineering data sources toggle button from the standard toolbar to switch to the default view of engineering data workspace.
Choose the return to project button from the standard toolbar to display the

6.4. Generating The Mesh

After the model is created in the design modeler window, you need to generate the mesh for the model in the mechanical window.

1. In the project schematic window, double-click on the model cell in the static structural analysis system; the mechanical window is displayed.
2. Select mesh in the tree outline to display the details of "mesh" window.
3. In the details of "mesh" window, expand the sizing node, if it is not already expanded. Also, notice that default is displayed in the element size edit box.

The element size edit box is used to specify the size of an element. The element size specified in this edit box is according to the size of the geometry. However, this edit box will not be visible when the

- On: proximity and
- On: proximity and curvature

Options are selected from the use advanced size function drop-down list. When default is displayed in the element size edit box, it indicates that a default value, based on the size of the geometry, is already specified by the software.

4. Choose the generate mesh tool from the mesh drop-down in the mesh contextual toolbar; the mesh is generated.

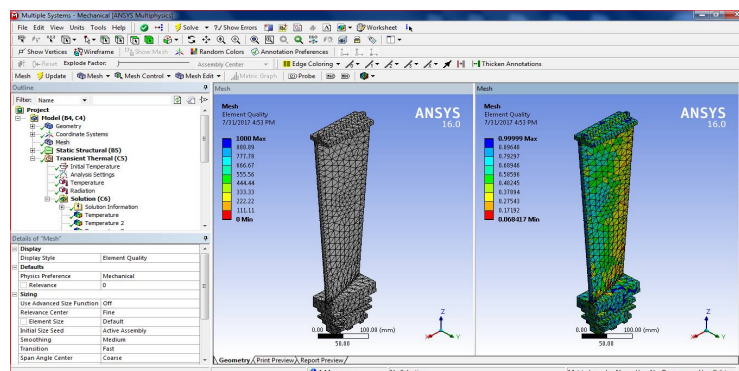


Figure 6.1 Mesh Generated With Default Mesh Controls

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

5. Expand the statistics node in the details of "mesh" window to display the total number of elements created. On doing so, you will find that the total number of elements.

6.5. Specifying The Boundary Conditions

After you mesh the model, it is required to specify the boundary and loading conditions.

1. In the mechanical window, select the static structural node from the tree outline; the details of "static structural" window is displayed along with the environment contextual toolbar.

2. Choose the fixed support tool from the supports drop-down in the environment contextual toolbar; fixed support is added under the static structural node. Also, the details of "static structural" window is displayed

3. Click on the geometry selection box to display the apply and cancel buttons

4. Choose the face tool from the select toolbar to enable selection of faces.

5. Select faces on the model. Next, choose the apply button in the geometry selection box; the selected faces turn purple indicating that fixed support is applied

6. Choose the force tool from the loads drop-down in the environment contextual toolbar; force is added under the static structural node in the tree outline. Also, the details of "pressure" window is displayed

7. Click on the geometry selection box to display the apply and cancel buttons, if they are not already displayed.

8. Next, select the circular face on the right of the model, as shown in figure.

9. Choose the apply button from the geometry selection box; the cylindrical face turns red indicating that the force load is applied.

10. In the details of "pressure" window, expand the definition node, if it is not already expanded.

11. Select vector from the define by drop-down list, if it is not already selected.

12. In the details of "force" window, click on the right arrow next to the magnitude edit box; a fly-out is displayed.

13. Choose constant from the fly out, if it is not already chosen, as shown in figure. The constant option is chosen when the force applied remains constant with respect to time,

14. In the magnitude edit box, enter number.

15. Click on the direction selection box to display the apply and cancel buttons.

As the application of force under consideration is vertically downward, you need to define the direction by selecting edges for the force vector.

16. Select any vertical edge on the model, as shown in figure 949, to specify the direction of force application

17. Next, choose the apply button from the direction selection box; a downward force is specified for the analysis.

After you have selected the edge for specifying the direction, you can flip the direction

18. Specified by choosing the flip button available in the graphics screen.

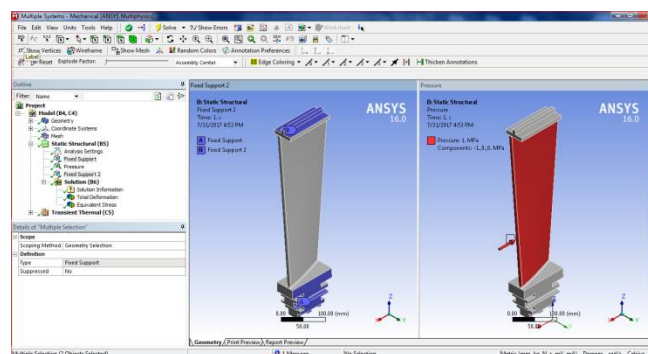


Figure 6.2The Violet And Red Color Face Of The Model Displaying The Fixed Support And Pressure

6.6. Solving The Fea Model And Analyzing The Results

1. Select the solution node in the tree outline; the solution contextual toolbar is displayed. Also, the details of "solution" window is displayed.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

2. Choose the total tool from the deformation drop-down of the solution contextual toolbar; total deformation is added under the solution node.
3. Now, choose the directional tool from the deformation drop-down; directional deformation is added under the solution node.
4. Choose the equivalent (von-misses) tool from the stress drop-down in the solution contextual toolbar; The equivalent or von-misses stress is the criteria by which the effect of all the directional stresses acting at a point is considered. It helps in finding out whether the model will fail or bear the stress at that particular point.
5. Choose the fatigue tool from the contact tool drop-down in the solution contextual toolbar
6. Again go to the fatigue tool from the solution and right click on it. Select life and damages for the model.

6.6.1. Results For Stainless Steel

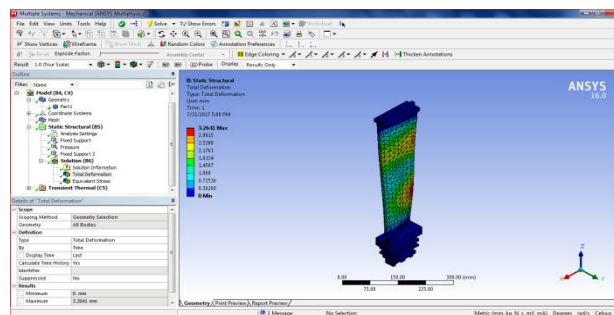


Figure 6.3 The Max And Min Values Of Total Deformation

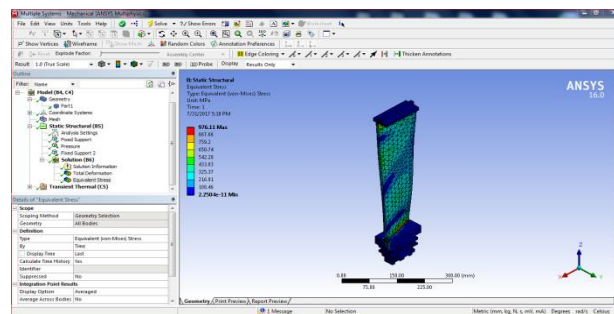


Figure 6.4 The Maxi And Mini Values Of Equivalent Stress In Color Bands

6.6.2. Results For Titanium Alloy

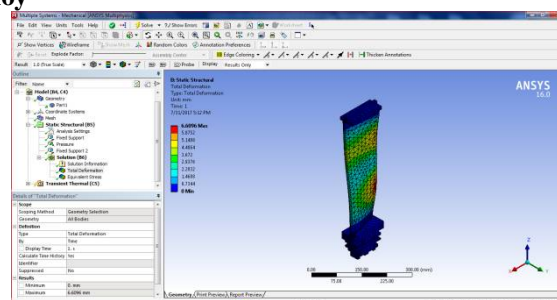


Figure 6.5 The Max And Min Values Of Total Deformation

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

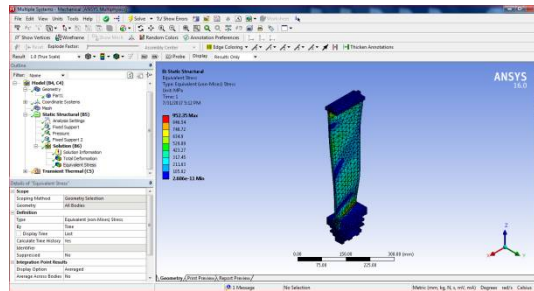


Figure 6.6 The Maxi And Mini Values Of Equivalent Stress In Color Bands

6.6.3. Results For Aluminum Alloy

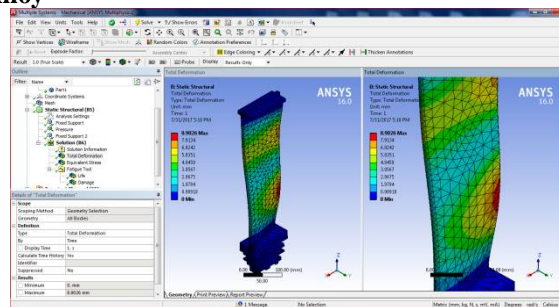


Figure 6.7 The Max And Min Values Of Total Deformation

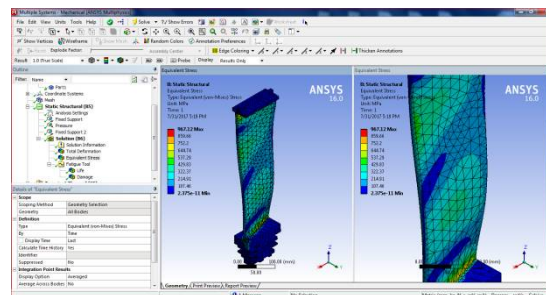


Figure 6.8 The Maxi And Min Values Of Equivalent Stress In Color Bands

1. Select all other parameters from the solution node; the respective view is displayed in the graphics screen. The table given next lists all the results obtained from the analysis. Show the corresponding graphical representation of values obtained.
2. Close the existing mechanical window; the workbench window is displayed.
A body is called to be deformed if its shape is changed temporarily or permanently. The temporary change of shape is known as elastic deformation and a permanent change of shape is known as plastic deformation. In Ansys workbench, you can determine deformation in terms of total and directional deformations.
Total deformation is the total change of shape in a given working condition. You can view the total deformation induced in any component by using the total tool from the deformation drop-down in the solution contextual toolbar. Directional deformation is the total change of shape in a particular axis, due to given working conditions. You can view directional deformation by using the directional tool from the deformation drop-down in the solution contextual toolbar.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Total deformation is the summation of all directional deformations produced in a certain region of the model. The following equation describes the total deformation:

If deformation in the x-axis u_x
Deformation in the y-axis u_y
Deformation in the z-axis u_z

Then total deformation, U will be given as follows:

$$U = (u_x^2 + u_y^2 + u_z^2)^{1/2}$$

The legend has colors arranged in a band from top to bottom. Depending upon the type of analysis and the parameters evaluated, each color will indicate a different value. A typical legend displayed when total deformation is selected from the tree outline.

The blue color in the legend indicates the minimum value of total deformation. In this case, it displays 0 which means there is no deformation at that region.

Exit the static structural analysis - mechanical window

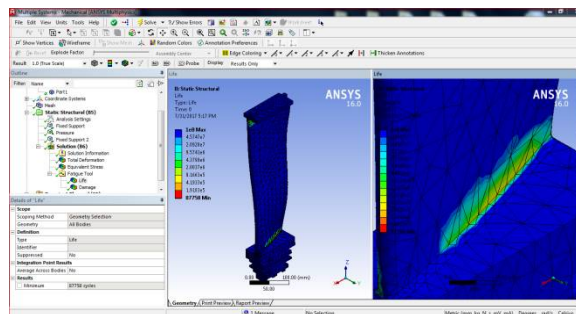


Figure 6.9 The Maximum And Minimum Values Of Life In Color Bands

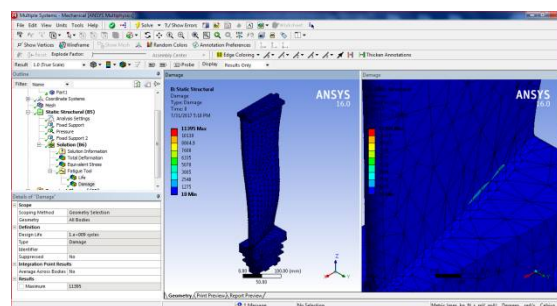


Figure 6.10 The Maximum And Minimum Values Of Damage In Color Band

VII. TRANSIENT THERMAL ANALYSIS

After completing of this chapter, we can able to Understand the types of thermal analysis

- Understand the mechanical interface used for thermal analysis.
- Understand different terms used in thermal analysis.
- Perform steady-state thermal analysis.
- Perform transient thermal "analysis."
- Understand temperature distribution.
- Run problem

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Before a model is set for production, it passes through several stages. Thermal analysis is one of them and plays an important role in product development. Various products such as engines, refrigerators, heat exchangers, and so on are designed based on the results of this analysis. Thermal analysis is used to determine the temperature distribution and related thermal quantities in the model. In this analysis, all heat transfer modes, namely conduction, radiation, and radiation are analyzed.

This analysis is used in many engineering industries such as automobile, piping, electronic, power generation, and so on. In Ansys workbench, two types of thermal analysis can be carried out, namely steady-state and transient thermal analysis.

The following are the basic steps required to perform the thermal analysis:

- Set the analysis preference.
- Create or import solid model.
- Define element attributes (element types, real constants, and material properties).
- Mesh the solid model.
- Specify the analysis type, analysis options, and the loads to be applied.
- Solve the analysis problem.
- Post-process results.

7.1. Types Of Thermal Analysis

In Ansys workbench, two types of thermal analysis can be carried out, namely steady-state thermal analysis and transient thermal analysis.

7.1.1. Steady-State Thermal Analysis

In the steady-state thermal analysis, the thermal load does not vary with time and remains constant throughout the period of application. This analysis considers only steady loads and does not consider any thermal load that varies with time. In the steady-state thermal analysis, the system is studied under steady thermal loads with respect to time. These thermal loads include radiation, radiation, heat flow rates, heat fluxes (heat flow per unit area), heat generation rates (heat flow per unit volume), and constant temperature boundaries.

The steady-state thermal analysis may be either linear or nonlinear, with respect to material properties that depend on temperature. The thermal properties of most of the materials do vary with temperature, therefore the analysis usually is nonlinear. Including radiation effects or temperature dependent radiation in a model also makes the analysis nonlinear.

The steps to solve a problem related to the thermal analysis are the same as that of the structural analysis, except a few steps such as selecting the element type, applying the load, and post processing results. .

7.2. Project Objective

In this project we will open the existing above project. Next, your project with a different name and run the steady-state thermal analysis on the model shown in figure to find out the effect of temperature on the whole body of the model. Also, we will evaluate the total heat flux and the directional heat flux with respect to the direction heat flux with respect to axis. The parameters required to run the analysis are given next.

7.2.1. Boundary Conditions:

Apply the temperature load of 80 °c on the front flat face of the model as shown in figure. Apply the radiation load on the inner faces of the model, as shown in figure. The inner faces are exposed to air. The ambient temperature is 22 °c.

Material: structural steel (default)

The following steps are required to complete this tutorial.

7.2.2. Applying Thermal Boundary Conditions

After the model is meshed, you now need to apply the boundary condition for the analysis.

1. Select the transient thermal analysis node in the tree outline to display the environment contextual toolbar.

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

2. From the environment contextual toolbar, choose the temperature tool; temperature is added under the steady-state thermal node. Also, the details of “temperature” window is displayed.
3. In the details of “temperature” window, select the geometry selection box to display the apply and cancel
4. Select the front face of the model
5. Choose the apply button from the geometry selection box in the details of “temperature” window; the face is selected for applying the temperature boundary condition.
6. In the details of “temperature” window, specify tabular data in the magnitude edit box.
7. Select temperature from the steady-state thermal node in the tree outline; the graph and tabular data windows are displayed, as shown in figures.

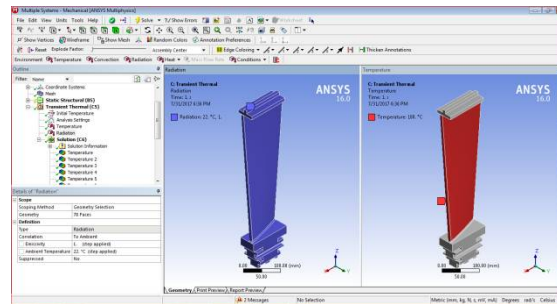


Figure 7.1 Temperature And Radiation The Steady-State Thermal

7.3. Analyzing The Results

After the thermal boundary conditions are applied, it is now important to analyze the behavior of the model with respect to the boundary conditions applied.

1. Right-click on the solution node in die tree outline to display a shortcut menu.
2. From this shortcut menu, choose insert > thermal > temperature; temperature is added under the solution node in the tree outline. Also, the details of “temperature” window is display
3. In this window, select the geometry selection box to display the apply and cancel buttons.
4. Choose the body tool from the select toolbar and then select the model, as shown in figr
5. Choose the apply button from the geometry selection box in the details of “temperature” window; 1 body is displayed in the geometry selection box.
6. Choose thermal > temperature from the solution contextual toolbar; temperature 2 is added under the solution node in the tree outline. Also, the details of “temperature 2” window is displayed.
7. In the details of “temperature 2” window, select the geometry selection box to display the apply and cancel buttons.
8. Choose the face tool from the select toolbar and then select the cylindrical face of the model
9. Choose the apply button from the details of temperature 2 window, 1 face is displayed in the geometry selection box in the details of temperature 2 window.
10. Choose the solve tool from the standard toolbar; the Ansys workbench solution status message box is displayed and the temperature distribution for the temperature 2 boundary condition is displayed in the graphics screen.
11. Select temperature under the solution node in the tree outline; temperature distribution in the model is displayed in the graphics screen, as shown in figure.
12. Similarly, select temperature 2 under the solution node in the ‘tree outline; the temperature distribution in the model is displayed in the graphics screen,
13. Select the solution node in the tree outline to display the solution contextual toolbar.
14. Choose total temperature from the thermal drop-down in the solution contextual toolbar;

7.3.1. For Stainless Steel

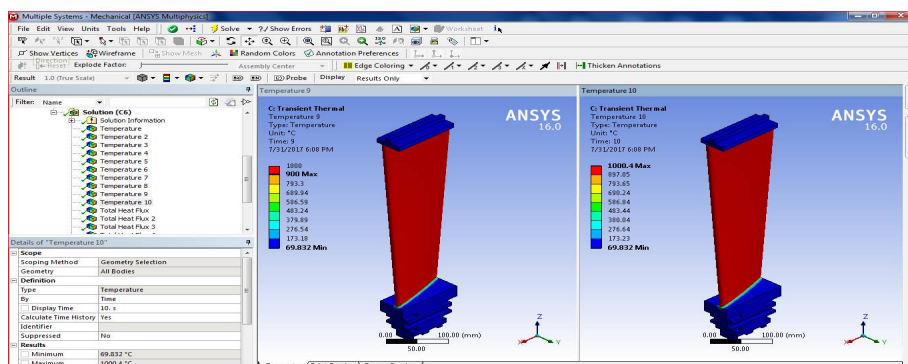
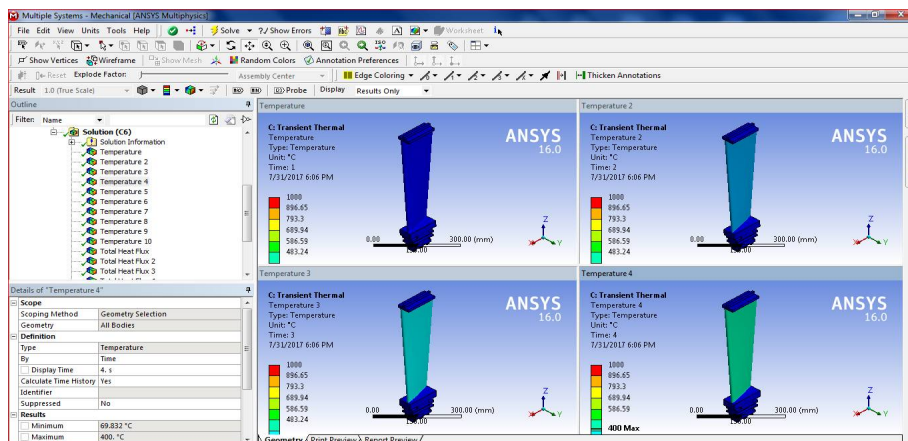
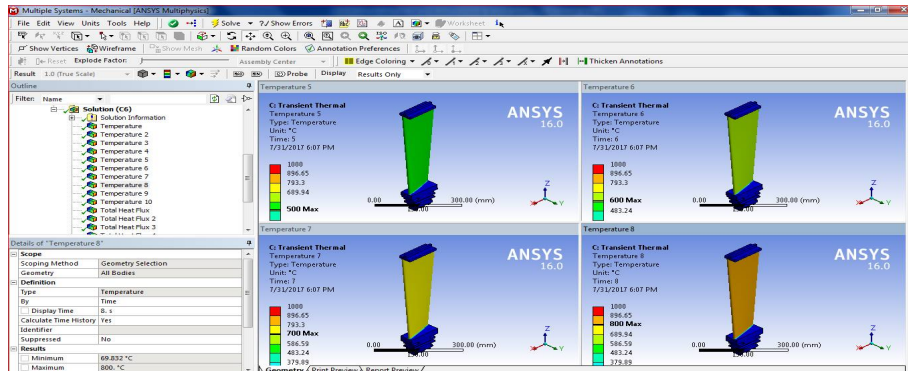


Figure 7.2 Total Temperature Thunderbolt Is Added Under Solution Node

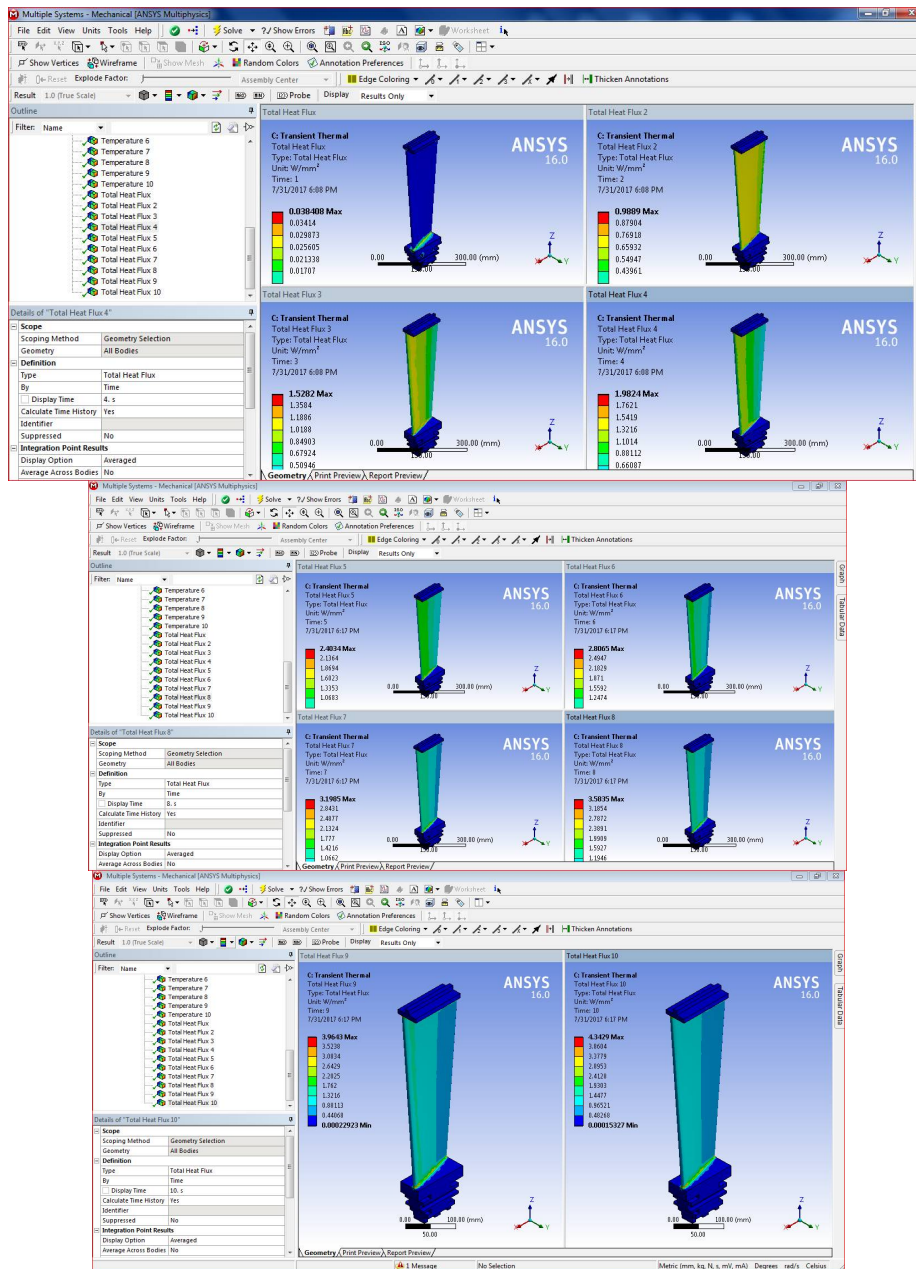


Figure 7.3 Total Heat Flux Thunderbolt Is Added Under Solution

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Figure 7.4 Total Temperature Thunderbolt Is Added Under Solution Node

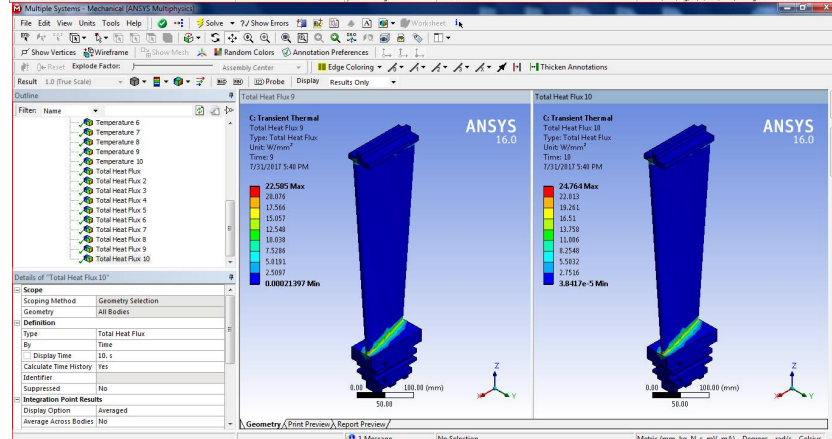
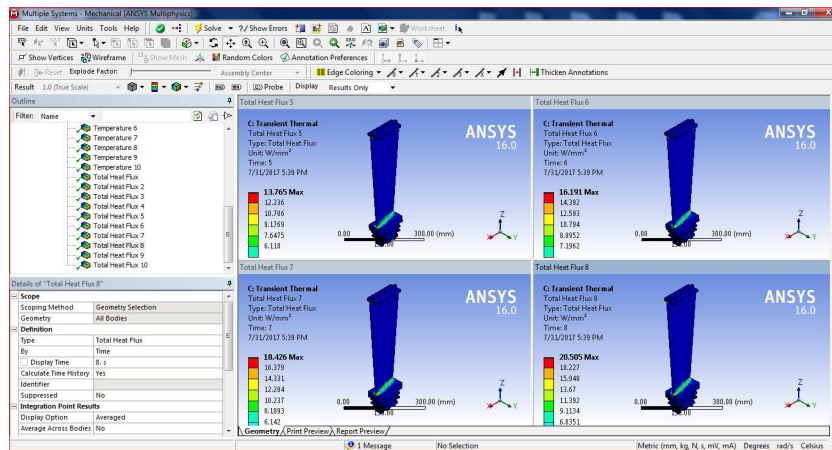
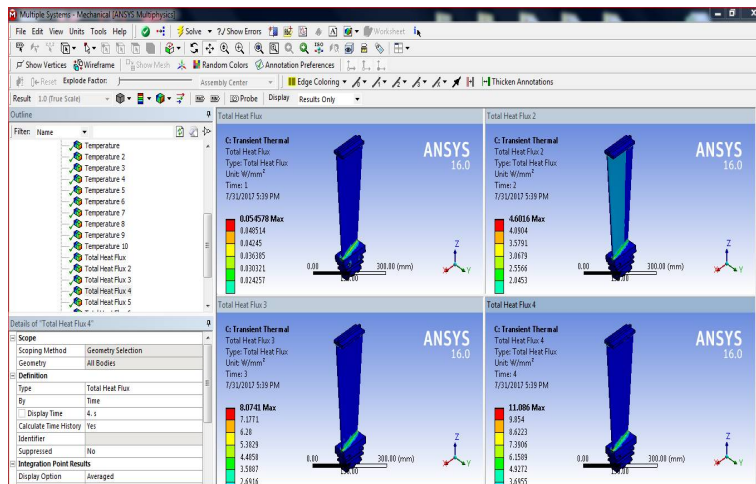


Figure 7.5 Total Heat Flux Thunderbolt Is Added Under Solution

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

15. In the details of “total heat flux” window, select the geometry selection box to display the apply and cancel buttons.
16. Choose the body tool from the select toolbar and then select the model,
17. Choose the apply button from the geometry selection box in the details of “total heat flux” window; 1 body is displayed in the geometry selection box.
18. Similarly, choose heat flux from the thermal fly out in the solution contextual toolbar;
It is obvious from the above results that the total heat flux is maximum in the region where the model is marked red and minimum where it is marked blue. Each color contour depicts a value and can be seen in the legend in the graphics screen.
19. Exit the mechanical window to display the workbench window.

VIII. FLOW OF STREAM IN BETWEEN BLADES

8.1. File Report

Case	CFX 1
File Path	C:\Users \Desktop\thermal\THERMAL ANALYSIS OF STREAM TURBINE BLADE SHAFT\b_files(dp0)\CFX-1\CFX\Fluid Flow CFX_002.res
File Date	31 July 2017
File Time	11:00:07 PM
File Type	CFX5
File Version	16.0

File Information for CFX

Table 1 File Information ForCfx

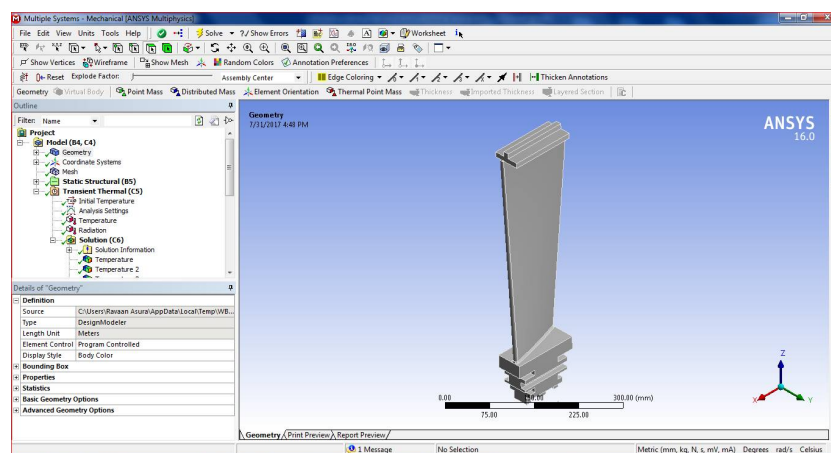


Figure 8.1 Modal

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

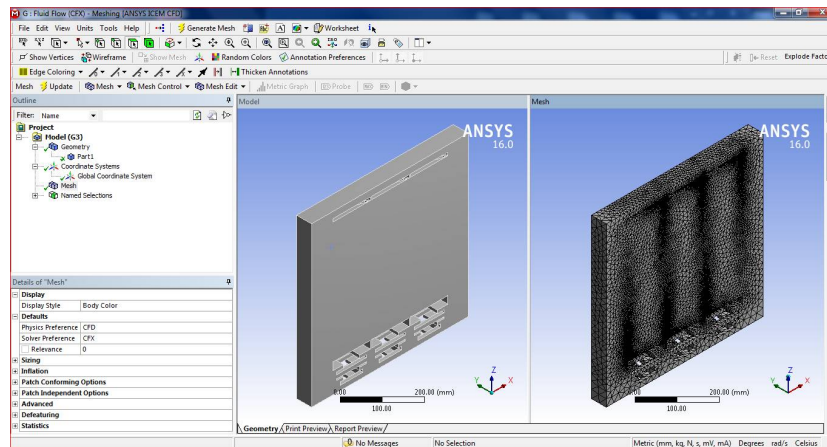


Figure 8.2 Vacuum Area And Meshing

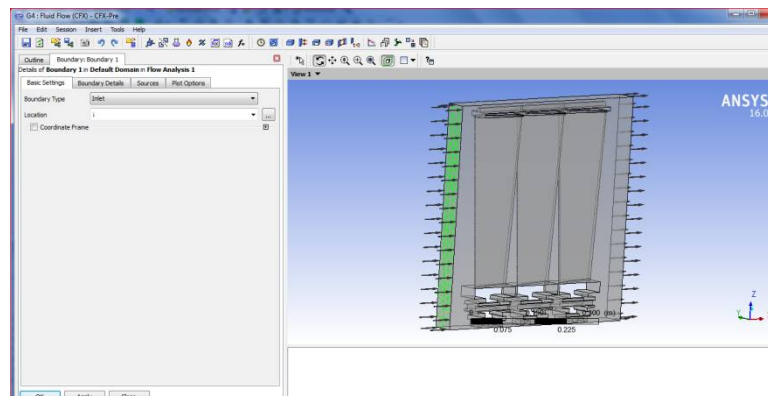


Figure 8.3 Boundary Conditions- Inlet

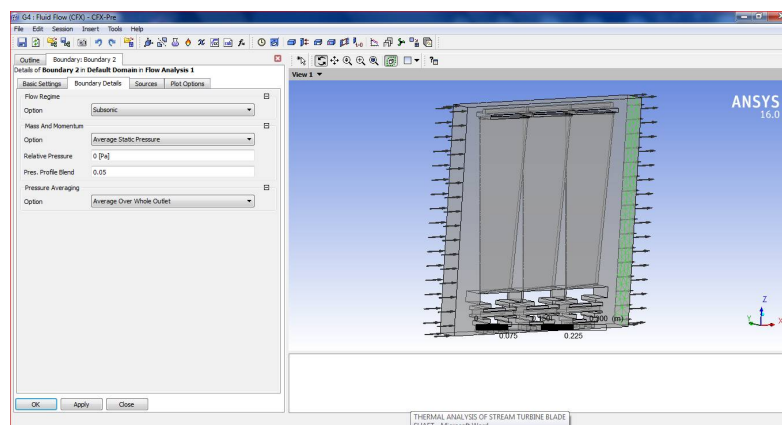


Figure 8.4 Boundary Condition –Outlet

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

8.2. Mesh Report

Mesh Information for CFX

Domain	Nodes	Elements	Tetrahedra	Wedges	Pyramids	Hexahedra	Polyhedra
Default Domain	622163	3391579	3391579	0	0	0	0

Table 2 Mesh Information ForCfx

Mesh Statistics for CFX

Domain	Minimum Face Angle	Maximum Face Angle	Maximum Edge Length Ratio	Maximum Element Volume Ratio	Connectivity Range	
Default Domain	9.04224 [degree]	134.043 [degree]	4.95697	18.4991	2	44

Table 3 Mesh Statistics ForCfx

8.3. Physics Report

Domain Physics for CFX

Domain - Default Domain	
Type	Fluid
Location	B4
<i>Materials</i>	
Air at 25 C	
Fluid Definition	Material Library
Morphology	Continuous Fluid
<i>Settings</i>	
Buoyancy Model	Non Buoyant
Domain Motion	Stationary
Reference Pressure	1.0000e+00 [atm]
Heat Transfer Model	Isothermal
Fluid Temperature	2.5000e+01 [C]
Turbulence Model	k epsilon
Turbulent Wall Functions	Scalable

8.4. Solution Report

Boundary Flows for CFX

Location	Type	Mass Flow	Momentum		
			X	Y	Z
Boundary 1	Boundary	1.3109e+01	1.6550e+04	-4.3632e-05	-1.2240e-03
Boundary 2	Boundary	-1.3111e+01	-8.5730e+03	4.1369e+01	-8.1449e+02
Boundary 3	Boundary	0.0000e+00	-7.5715e+03	-1.0604e+04	-6.9899e+03
Default Domain Default	Boundary	0.0000e+00	-4.0573e+02	1.0564e+04	7.7937e+03

Table 4 Boundary Flows ForCfx

Forces and Torques for CFX

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

Location	Type	X	Y	Z
Boundary 3	Pressure Force	7.3251e+03	1.0601e+04	6.9705e+03
	Viscous Force	2.4639e+02	2.8501e+00	1.9359e+01
	Total Force	7.5715e+03	1.0604e+04	6.9899e+03
	Pressure Torque	-4.6226e+03	9.9669e+02	1.2868e+03
	Viscous Torque	1.6135e+00	6.7567e+01	-3.9334e-01
	Total Torque	-4.6210e+03	1.0643e+03	1.2864e+03
Default Domain Default	Pressure Force	-4.5024e-14	-1.0564e+04	-7.8188e+03
	Viscous Force	4.0573e+02	1.6992e-01	2.5106e+01
	Total Force	4.0573e+02	-1.0564e+04	-7.7937e+03
	Pressure Torque	4.5808e+03	2.6006e+02	-1.2499e+03
	Viscous Torque	1.1943e+00	1.1764e+02	-1.6480e+00
	Total Torque	4.5820e+03	3.7770e+02	-1.2515e+03

Table 5 Forces And Torques For Cfx

8.5. Pictures

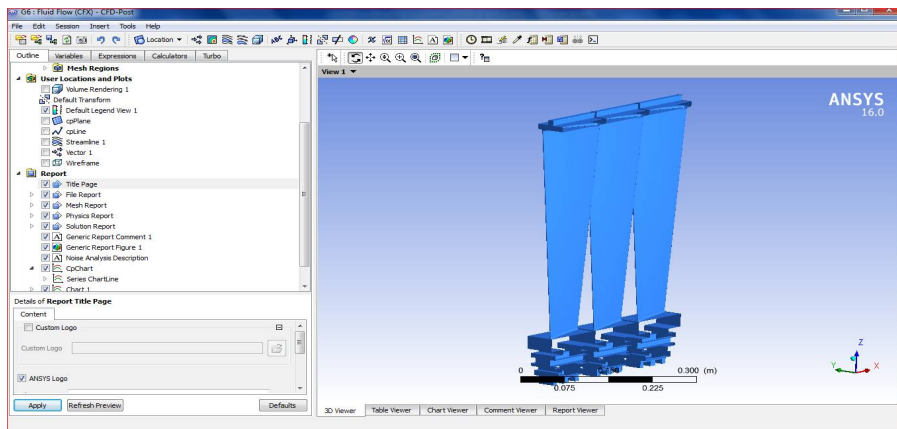


Figure 8.5 Modal In Fluent Resultant

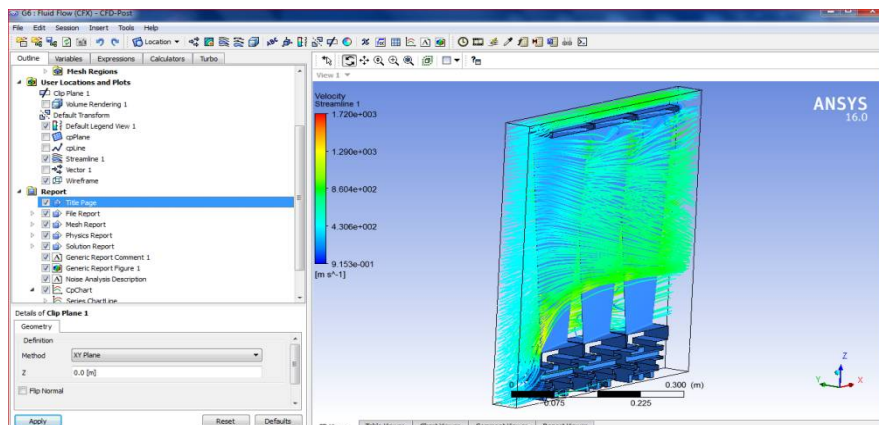


Figure 8.6 Flow Of The Exhaust Gas

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

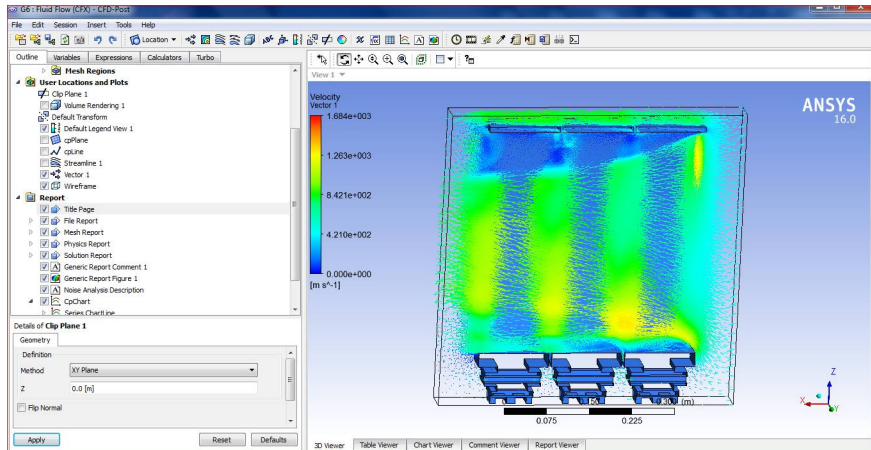


Figure 8.7 Vector Directions

8.6. Noise Analysis

This analysis has been supplied to assist in the evaluation of tonal noise levels generated by low speed fans (Mach number less than 0.4). The equations were obtained from available literature; however some equations may have alternate definitions. It is **your responsibility** to verify the accuracy of these definitions.

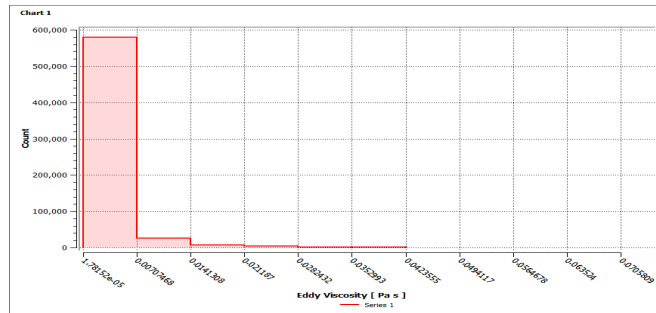


Figure 8.8 Chart

IX. RESULTS AND GRAPHS

A Static Structural Analysis

	Total Deformation Mm	Equivalent Stress Mpa
Stainless Steel	3.2641	976.11
Titanium Alloy	6.6096	952.35
Aluminum Alloy	8.9026	967.12

Table 6 Deformations And Stress By Pressure

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

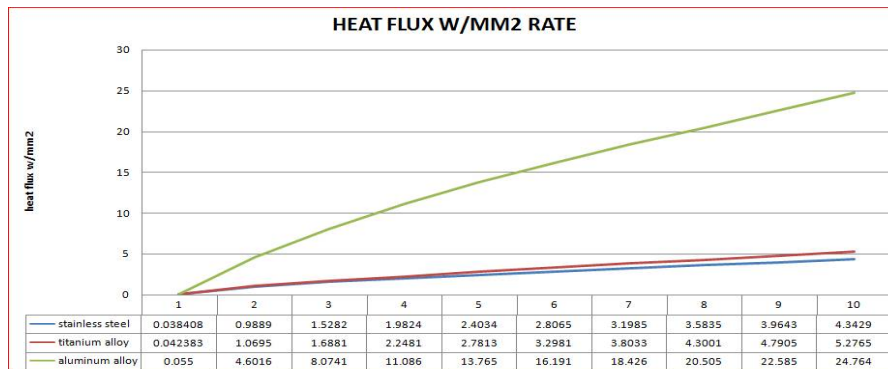
Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

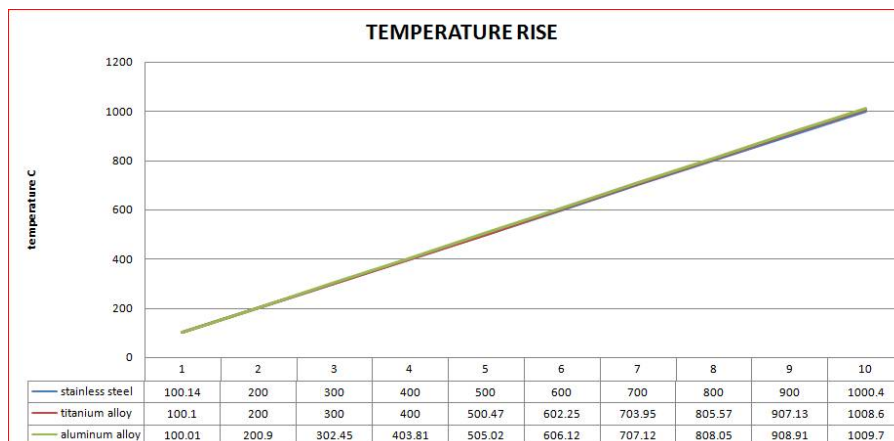
B Transient Thermal Analysis

S.No	Heat Flux W/Mm2			Temperature C		
	Stainless Steel	Titanium Alloy	Aluminum Alloy	Stainless Steel	Titanium Alloy	Aluminum Alloy
1	0.038408	0.042383	0.055	100.14	100.1	100.01
2	0.9889	1.0695	4.6016	200	200	200.9
3	1.5282	1.6881	8.0741	300	300	302.45
4	1.9824	2.2481	11.086	400	400	403.81
5	2.4034	2.7813	13.765	500	500.47	505.02
6	2.8065	3.2981	16.191	600	602.25	606.12
7	3.1985	3.8033	18.426	700	703.95	707.12
8	3.5835	4.3001	20.505	800	805.57	808.05
9	3.9643	4.7905	22.585	900	907.13	908.91
10	4.3429	5.2765	24.764	1000.4	1008.6	1009.7

Table 7 Temperatures And Heat Fluxes By Temperature Rise From 100 To1000 C



Graph 1 Heat Flux W/Mm2



Graph 2 Temperature C

International Journal of Innovative Research in Science, Engineering and Technology

(A High Impact Factor, Monthly, Peer Reviewed Journal)

Visit: www.ijirset.com

Vol. 8, Issue 5, May 2019

X. CONCLUSION

I have finally acquired knowledge over **Steam turbines** and its manufacturing process which had led me know many unknown things that actually be followed during the production procedure and also about the various materials used.

In chapter 3 i used catia software to redesign and later imported into ansys software's.

In chapter 4 , 5 and 6 are deals with the ansys software. In ansys i used find the deformation and stress for stainless, al alloy and ti alloy and thermal and heat flux.

From chapter 7 the temperature as high for al alloy and heat flux is also high for the al alloy

In chapter 8 flow of the stream as been determined

REFERENCES

1. *"Flight Operations Briefing Notes - Supplementary Techniques : Handling Engine Malfunctions"* (PDF). Airbus.
2. [propeller efficiency Archived](#) May 25, 2008, at the [Wayback Machine](#).
3. *"Centenary of the First Gas Turbine to Give Net Power Output: A Tribute to Ægidius Elling"*. The American Society of Mechanical Engineers. 14 June 2004. Retrieved 26 April 2015.
4. Maxime Guillaume, "Propulseur par réactionsurl'air," French patent no. 534,801 (filed: 3 May 1921; issued: 13 January 1922). Available on-line (in French) at: <http://v3.espacenet.com/origdoc?DB=EPODOC&IDX=FR534801&F=0&QPN=FR534801> .
5. *"Chasing the Sun - Frank Whittle"*. PBS. Retrieved 2010-03-26.
6. *"History - Frank Whittle (1907 - 1996)"*. BBC. Retrieved 2010-03-26.
7. Frank Whittle, "Improvements relating to the propulsion of aircraft and other vehicles," British patent no. 347,206 (filed: 16 January 1930). Available on-line at: <http://v3.espacenet.com/origdoc?DB=EPODOC&IDX=GB347206&F=0&QPN=GB347206> .
8. [The History of the Jet Engine - Sir Frank Whittle - Hans Von Ohain](#) Ohain said that he had not read Whittle's patent and Whittle believed him. (Frank Whittle 1907-1996).
9. Warsitz, Lutz: *THE FIRST JET PILOT - The Story of German Test Pilot Erich Warsitz* (p. 125). Pen and Sword Books Ltd., England, 2009
10. *"ch10-3"*. Hq.nasa.gov. Retrieved 2010-03-26.
11. Mattingly, Jack D. (2006). *Elements of Propulsion: Gas Turbines and Rockets*. AIAA Education Series. Reston, VA: American Institute of Aeronautics and Astronautics. p. 6. ISBN 1-56347-779-3.
12. *"Trent XWB infographic"*. Retrieved 2015-10-15.
13. [Hill & Peterson 1992](#), pp. 190.
14. [Mattingly 2006](#), pp. 12-14.
15. Sweetman, Bill (2005). *The Short, Happy Life of the Prop-fan*. *Air & Space Magazine*. 1 September 2005.
16. ^ Jump up to:^a & Mattingly, p. 14
17. *Flack, Ronald D. (2005). *Fundamentals of Jet Propulsion with Applications*. Cambridge Aerospace Series. New York, NY: Cambridge University Press. p. 16. ISBN 978-0-521-81983-1.
18. Benson, Tom. [Ramjet Propulsion](#). NASA Glenn Research Center. Updated: 11 July 2008. Retrieved: 23 July 2010.
19. Heiser, William H.; Pratt, David T. (1994). *Hypersonic Airbreathing Propulsion*. AIAA Education Series. Washington, D.C.: American Institute of Aeronautics and Astronautics. pp. 23–4. ISBN 1-56347-035-7.
20. [X-51 Waverider makes historic hypersonic flight](#). United States Air Force. 26 May 2010. Retrieved: 23 July 2010.
21. *"Rocket Thrust Equation"*. Grc.nasa.gov. 2008-07-11. Retrieved 2010-03-26.
22. **Jump up**^ Jet Propulsion for Aerospace Applications Second Edition 1964, Hesse and Mumford, Pitman Publishing Corporation, Library of Congress Catalog Card Number 64-18757,p.48