

Design and Analysis of Pelton Turbine by Ansys

Azgerpasha Shaik, Yogesh Avula, Venkata Ramana T

Abstract: The Pelton wheel is an impulse type water turbine. It was invented by Lester Allan Pelton in the 1870s. The Pelton wheel extracts energy from the impulse of moving water, as opposed to water's dead weight like the traditional overshot water wheel. Many variations of impulse turbines existed prior to Pelton's design, but they were less efficient than Pelton's design. Water leaving those wheels typically still had high speed, carrying away much of the dynamic energy brought to the wheels. Pelton's paddle geometry was designed so that when the rim ran at half the speed of the water jet, the water left the wheel with very little speed; thus his design extracted almost all of the water's impulse energy—which allowed for a very efficient turbine. In general all solid and non solid 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 softwares like Creo parametric, solid works, catia and solid edge. 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 valve with respect to the model or product definitions. In this project the product was undergone various types of analysis to find frequencies with respect to gravity or mass by using Modal analysis and by Using static structural analysis total deformation, stress and strain valve, product life and failures etc., can be calculate by using Ansys workbench. Ansys software helps to find the accurate or approximate solutions

Keywords: flux (change of heat flow with respect to time and distance) and CFD, "STP"

I. INTRODUCTION OF PELTON WHEEL

The Pelton wheel is an impulse type water turbine. It was invented by Lester Allan Pelton in the 1870s. The Pelton wheel extracts energy from the impulse of moving water, as opposed to water's dead weight like the traditional overshot water wheel. Many variations of impulse turbines existed prior to Pelton's design, but they were less efficient than Pelton's design. Water leaving those wheels typically still had high speed, carrying away much of the dynamic energy brought to the wheels. Pelton's paddle geometry was designed so that when the rim ran at half the speed of the water jet, the water left the wheel with very little speed; thus his design extracted almost all of the water's impulse energy—which allowed for a very efficient turbine.

Revised Version Manuscript Received on June 22, 2017.

Azgerpasha Shaik, Mechanical Department, Guru Nanak Institute of Technology (GNIT), Hyderabad, India, E-mail: azger.mecheng@gmail.com

Yogesh Avula, Mechanical Department, Guru Nanak Institute Of Technology (GNIT), Hyderabad, India, E-mail: yogesh.avula@gmail.com

Venkata Ramana T, Mechanical Department, Guru Nanak Institute of Technology (GNIT), Hyderabad, India, E-mail: ramana302i@gmail.com



Figure 1 Old Pelton Wheel



Figure 2 Assembly of A Pelton Wheel

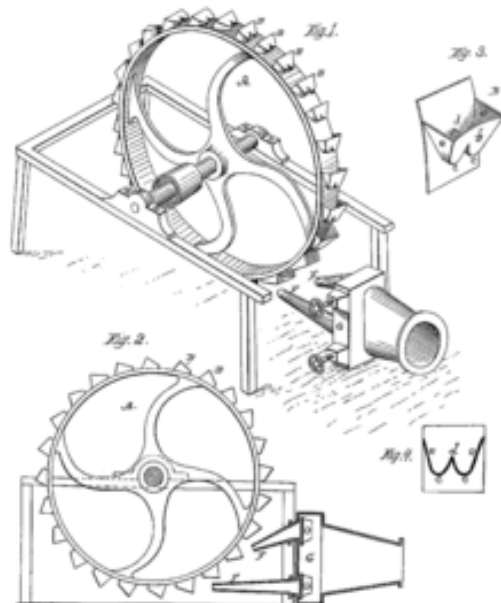


Figure 3 Pelton's Original Patent (October 1880).

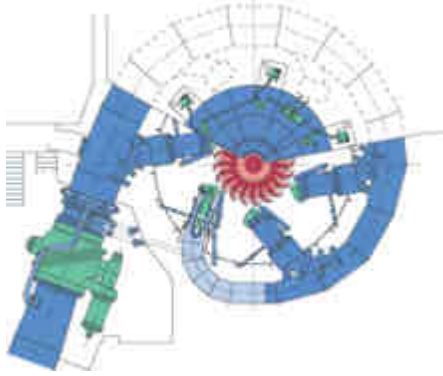


Figure 4 Sectional View of A Pelton Turbine Installation.



Figure 5 Bucket Detail On A Small Turbine.

A. Function

Nozzles direct forceful, high-speed streams of water against a rotary series of spoon-shaped buckets, also known as impulse blades, which are mounted around the circumferential rim of a drive wheel—also called a runner (see photo, 'Old Pelton wheel..'). As the water jet impinges upon the contoured bucket-blades, the direction of water velocity is changed to follow the contours of the bucket. Water impulse energy exerts torque on the bucket-and-wheel system, spinning the wheel; the water stream itself does a "u-turn" and exits at the outer sides of the bucket, decelerated to a low velocity. In the process, the water jet's momentum is transferred to the wheel and hence to a turbine. Thus, "impulse" energy does work on the turbine. For maximum power and efficiency, the wheel and turbine system is designed such that the water jet velocity is twice the velocity of the rotating buckets. A very small percentage of the water jet's original kinetic energy will remain in the water, which causes the bucket to be emptied at the same rate it is filled, (see conservation of mass) and thereby allows the high-pressure input flow to continue uninterrupted and without waste of energy. Typically two buckets are mounted side-by-side on the wheel, which permits splitting the water jet into two equal streams (see photo). This balances the side-load forces on the wheel and helps to ensure smooth, efficient transfer of momentum of the fluid jet of water to the turbine wheel.

Because water and most liquids are nearly incompressible, almost all of the available energy is extracted in the first stage of the hydraulic turbine. Therefore, Pelton wheels have only one turbine stage, unlike gas turbines that operate with compressible fluid. It is used for generating electricity.

B. Applications

Pelton wheels are the preferred turbine for hydro-power, when the available water source has relatively high hydraulic head at low flow rates, where the Pelton wheel geometry is most suitable. Pelton wheels are made in all sizes. There exist multi-ton Pelton wheels mounted on vertical oil pad bearings in hydroelectric plants. The largest units can be over 400 megawatts. The smallest Pelton wheels are only a few inches across, and can be used to tap power from mountain streams having flows of a few gallons per minute. Some of these systems use household plumbing fixtures for water delivery. These small units are recommended for use with 30 meters (100 ft) or more of head, in order to generate significant power levels. Depending on water flow and design, Pelton wheels operate best with heads from 15–1,800 meters (50–5,910 ft), although there is no theoretical limit.

C. Design Rules

The specific speed parameter is independent of a particular turbine's size.

Compared to other turbine designs, the relatively low specific speed of the Pelton wheel, implies that the geometry is inherently a "low gear" design. Thus it is most suitable to being fed by a hydro source with a low ratio of flow to pressure, (meaning relatively low flow and/or relatively high pressure).

The specific speed is the main criterion for matching a specific hydro-electric site with the optimal turbine type. It also allows a new turbine design to be scaled from an existing design of known performance.

$$\eta_s = n\sqrt{P} / \sqrt{\rho(gH)^{5/4}}$$

where:

- n = Frequency of rotation (rpm)
- P = Power (W)
- H = Water head (m)
- ρ = Density (kg/m³)

The formula implies that the Pelton turbine is *geared* most suitably for applications with relatively high hydraulic head H , due to the 5/4 exponent being greater than unity, and given the characteristically low specific speed of the Pelton.

D. Turbine Physics and Derivation

1) Energy And Initial Jet Velocity

In the ideal (frictionless) case, all of the hydraulic potential energy ($E_p = mgh$) is converted into kinetic energy ($E_k = mv^2/2$) (see Bernoulli's principle). Equating these two equations and solving for the initial jet velocity (V_i) indicates that the theoretical (maximum) jet velocity is $V_i = \sqrt{2gh}$. For simplicity, assume that all of the velocity vectors are parallel to each other. Defining the velocity of the wheel runner as: (u), then as the jet approaches the runner, the initial jet velocity relative to the runner is: ($V_i - u$). The initial jet velocity of jet is V_i

2) Final Jet Velocity

Assuming that the jet velocity is higher than the runner velocity, if the water is not to become backed-up in runner,

then due to conservation of mass, the mass entering the runner must equal the mass leaving the runner. The fluid is assumed to be incompressible (an accurate assumption for most liquids). Also it is assumed that the cross-sectional area of the jet is constant. The jet speed remains constant relative to the runner. So as the jet recedes from the runner, the jet velocity relative to the runner is: $-(V_i - u) = -V_i + u$. In the standard reference frame (relative to the earth), the final velocity is then: $V_f = (-V_i + u) + u = -V_i + 2u$.

3) Optimal Wheel Speed

We know that the ideal runner speed will cause all of the kinetic energy in the jet to be transferred to the wheel. In this case the final jet velocity must be zero. If we let $-V_i + 2u = 0$, then the optimal runner speed will be $u = V_i/2$, or half the initial jet velocity.

4) Torque

By Newton's second and third laws, the force F imposed by the jet on the runner is equal but opposite to the rate of momentum change of the fluid, so:

$$F = -m(V_f - V_i) = -\rho Q[(-V_i + 2u) - V_i] = -\rho Q(-2V_i + 2u) = 2\rho Q(V_i - u)$$

where (ρ) is the density and (Q) is the volume rate of flow of fluid. If (D) is the wheel diameter, the torque on the runner is: $T = F(D/2) = \rho Q D (V_i - u)$. The torque is at a maximum when the runner is stopped (i.e. when $u = 0$, $T = \rho Q D V_i$). When the speed of the runner is equal to the initial jet velocity, the torque is zero (i.e. when $u = V_i$, then $T = 0$). On a plot of torque versus runner speed, the torque curve is straight between these two points, $(0, \rho Q D V_i)$ and $(V_i, 0)$.

5) Power

The power $P = Fu = T\omega$, where ω is the angular velocity of the wheel. Substituting for F , we have $P = 2\rho Q(V_i - u)u$. To find the runner speed at maximum power, take the derivative of P with respect to u and set it equal to zero, $[dP/du = 2\rho Q(V_i - 2u)]$. Maximum power occurs when $u = V_i/2$. $P_{\max} = \rho Q V_i^2/2$. Substituting the initial jet power $V_i = \sqrt{2gh}$, this simplifies to $P_{\max} = \rho gh Q$. This quantity exactly equals the kinetic power of the jet, so in this ideal case, the efficiency is 100%, since all the energy in the jet is converted to shaft output.^[4]

6) Efficiency

A wheel power divided by the initial jet power, is the turbine efficiency, $\eta = 4u(V_i - u)/V_i^2$. It is zero for $u = 0$ and for $u = V_i$. As the equations indicate, when a real Pelton wheel is working close to maximum efficiency, the fluid flows off the wheel with very little residual velocity.^[4] In theory, the energy efficiency varies only with the efficiency of the nozzle and wheel, and does not vary with hydraulic head.^[5] The term "efficiency" can refer to: Hydraulic, Mechanical, Volumetric, Wheel, or overall efficiency.

CHAPTER -2

II. 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.

It helps to acceleration and streamlines the product development process by helping designers to resolve issues relation to structural thermal fluid flow electromagnetic effect a combination of these phenomena acting together and soon.

In Ansys, the basics of Fea concepts, modelling and the analysing of engineering problem using Ansys workbench. In addition, describe of importance tools and concepts given whenever required .this following simulation streams of Ansys.

1. Structural analysis
 - Static structural analysis
 - Modal analysis
 - Transient structural analysis
2. Thermal analysis
 - Steady state thermal analysis
 - Transient thermal analysis

CHAPTER -3

III. MODAL ANALYSIS

A. Project Objective

In this project, we imported a model which is created in one of the cad software's, as shown in figure. The dimensions of the model are given in figure. You will generate the mesh with default global mesh control settings and find six natural frequencies and their respective mode shapes. The material used is structural steel.

The following steps are required to complete this tutorial:

1. Opening above saved project..
2. Generate the mesh.
3. Specify the boundary conditions.
4. Solve the analysis.
5. Retrieve the analysis results.
6. Play the animation.
7. Save the model.
8. Creating the model

After creating the project, you now need to work in the design modeler to create the model.

1. Double click on the geometry cell; the design modeler window along with the Ansys workbench dialog box is displayed.
2. Select the millimeter radio button in the Ansys workbench dialog box and then choose the ok button to specify millimeter as the unit for creating the sketch.

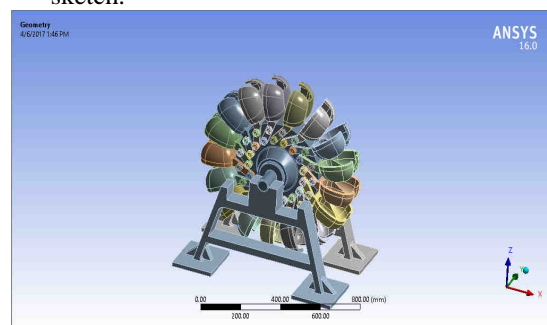


Figure 22 Modal Geometry

B. Generating the Mesh For The Model

Now, you need to generate the mesh of the model.;

1. Double click on model cell will enter into mechanical window. Also, you will notice that in the outline window, the mesh node is displayed in the tree outline with a yellow thunderbolt attached to it.
2. Click on mesh in the tree outline; the details of “mesh” window is displayed.
3. In the details of “mesh” window, expand the sizing node, if not already expanded
4. In the sizing node in the details of “mesh” window, (inter 2-5 in the element size edit box.
5. Right-click on mesh in the tree outline and then choose the preview > surface mesh from the shortcut menu displayed; the preview of the mesh for the model is displayed.
6. Choose the generate mesh tool from the mesh drop-down in the mesh contextual toolbar; the mesh is generated, as shown in figure.

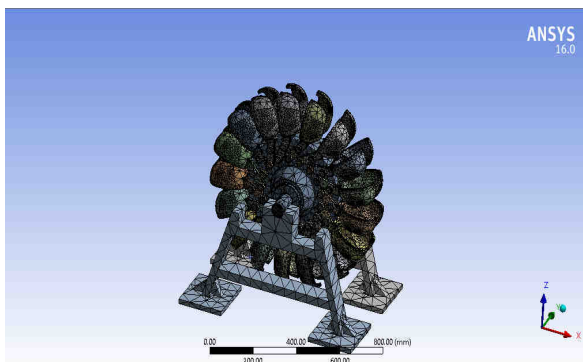


Figure 23 Mesh

C. Setting the Boundary Conditions

After the mesh is generated, you need to set the boundary conditions under which the analysis is to be performed.

1. Right-click on modal node in the tree outline and then choose insert > fixed support from the shortcut menu displayed; fixed support with a question symbol is added under the modal node in the tree outline. Also, the details of “fixed support window is displayed.
2. In the details of “fixed support” window, click on the geometry cell to display the apply and cancel buttons, if not already displayed.
3. Select the side face of the model, as shown in figure.
4. Next, choose the apply button from the geometry selection box in the details of “fixed support” window, fixed support is applied to the selected face.

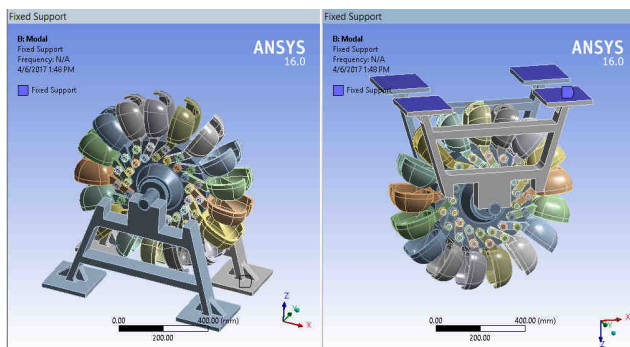


Figure 24 Fixed Supports

D. Solving the Modal Analysis

After specifying the boundary conditions in the mechanical window, you need to set the variables to define the results and solve the analysis.

1. Select analysis settings under the modal node in the tree outline; the details of “analysis settings” window are displayed.
2. In the details of “analysis settings” window, expand the options node, if it is not already expanded.
3. Enter 8 in the max modes to find edit box, if not already specified by default. Also make sure that no is selected in the limit search to range drop-down list, refer to figure.
4. Expand the solver controls node in the details of “analysis settings” window, if it is not already expanded.
5. In the damped drop-down list, select the no option, if not already selected.
6. Right-click on the solution node in the tree outline and then choose the solve option from the shortcut menu displayed; the analysis is solved.
7. Select the solution node in the tree outline; the graph and tabular data windows are displayed, refer to figure.

E. Retrieving Analysis Results

After the analysis is solved, you need to find the mode shapes.

1. Right-click in the graph window, a shortcut menu is displayed
2. Choose select all from this shortcut menu to select all the data available in the graph window, as shown in figure,
3. After the columns-in the graph window are selected, right-click again to display a shortcut menu.
4. Choose the create mode shape results option from the shortcut menu displayed, refer to figure; total deformation results are added under the solution node in the tree outline with the names: total deformation, total deformation 2, - - - total deformation 8.

Also, you will notice that there are yellow thunderbolts attached to each one of them indicating that these results need to be evaluated. The number of modes under the solution node depend upon the value specified in the max modes to find edit box in the details of “analysis settings” window.

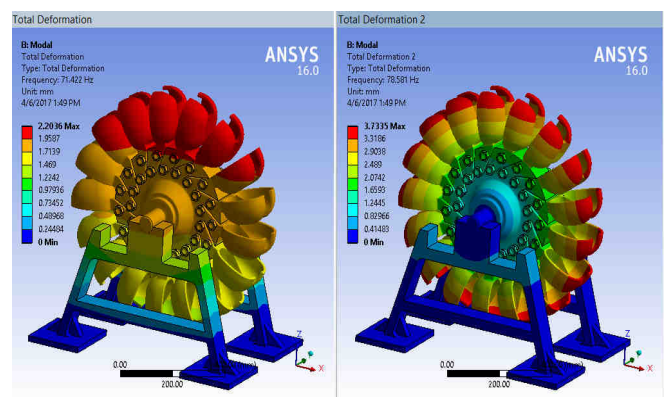


Figure 25 Total Deformation 1st, & 2nd

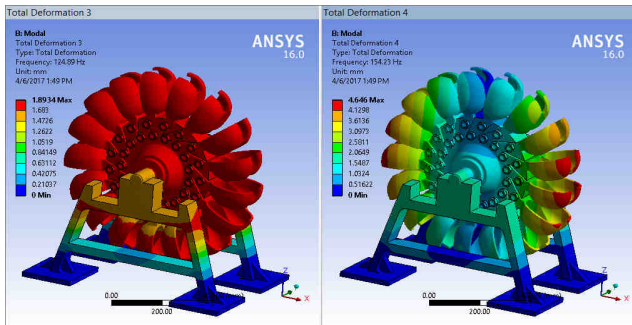


Figure 26 Total Deformation 3rd, & 4th

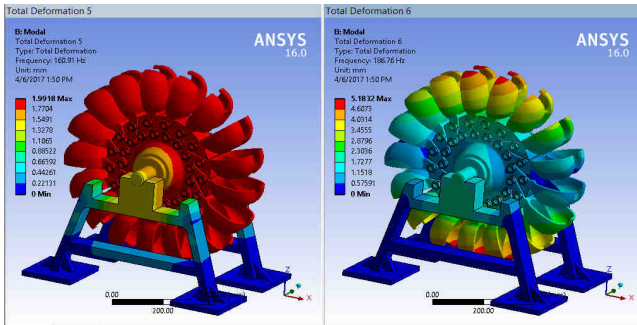


Figure 27 Total Deformation 5th & 6th

- Right-click on the solution node again and then choose the evaluate all results from the shortcut menu displayed; all the six results are ready to be viewed.
- Select total deformation under the solution node in the tree outline; the first mode is displayed in the graphics screen, as shown in figure.
- Select total deformation 2 under the solution node; the second mode shape is displayed in the graphics screen, as shown in figure
- Similarly, select other results from the solution node to view the corresponding mode shape in the graphics screen.
- Go to solution and right click on solution- summary of results from workbench

Results	Minimum	Maximum	Units	Reported Frequency (Hz)
Total Deformation	0.	2.2036	mm	71.422
Total Deformation 2	0.	3.7335	mm	78.581
Total Deformation 3	0.	1.8934	mm	124.89
Total Deformation 4	0.	4.646	mm	154.23
Total Deformation 5	0.	1.9918	mm	160.91
Total Deformation 6	0.	5.1832	mm	186.76

Results Summary
[1]

CHAPTER 4

IV. STATIC STRUCTURAL ANALYSIS

A. Project Overview

In this project, you will create the model. The dimensions to create the model and its boundary and loading conditions are also given in the same figure. Run a static structural analysis on the model and evaluate the total deformation and the directional deformation. Determine directional deformation along the x, y, and z axes. After evaluating the results, interpret them

- A. Start a new project and create the model.

- B. Generate the mesh.
- C. Set the boundary and loading _conditions.
- D. Solve the model.
- E. Duplicate the existing analysis system.
- F. Interpret results.
- G. Save the project.

B. Starting A New Project And Creating The Model

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

- Start Ansys workbench.
- Choose the save button from the standard toolbar; the save as dialog box is displayed.
- Double-click on static structural in the toolbox window; the static structural analysis system is added in the project schematic window.
- Rename the static structural analysis system (if).
- Double-click on the geometry cell; the design modeler window along with the Ansys workbench dialog box is displayed.
- In the Ansys workbench dialog box, set the unit to millimeter. Now, create the model on the xy plane
- Exit the design modeler window to display the workbench window.

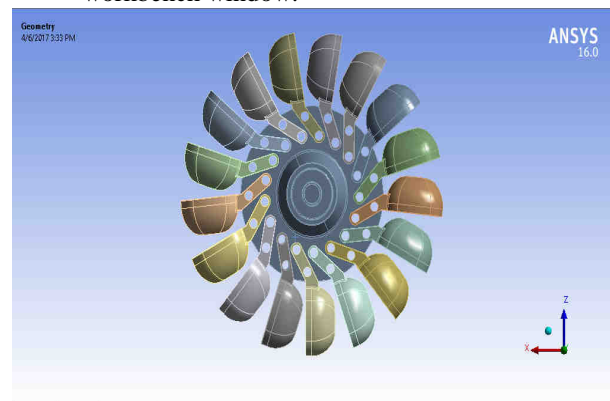


Figure 33 Modal In Static Structural Analysis

C. 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.

- Double-click on the engineering data cell of the spring plate analysis system; the engineering data workspace is displayed in the workbench window.
- Choose the engineering data sources toggle button from the standard toolbar; the engineering data sources window is added to the engineering data workspace.
- In the engineering data sources window, select the general materials library to display the outline of general materials window.
- 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.

- 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

D. Project Schematic Window.

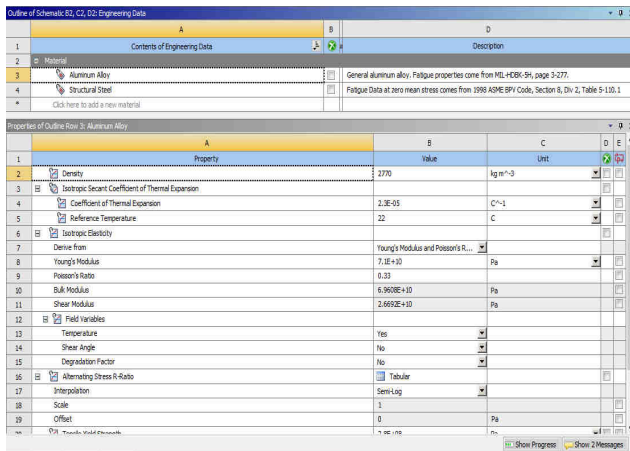


Figure 34 Specifying A Material

E. 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.

- In the project schematic window, double-click on the model cell in the static structural analysis system; the mechanical window is displayed.
- Select mesh in the tree outline to display the details of "mesh" window.
- 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.

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

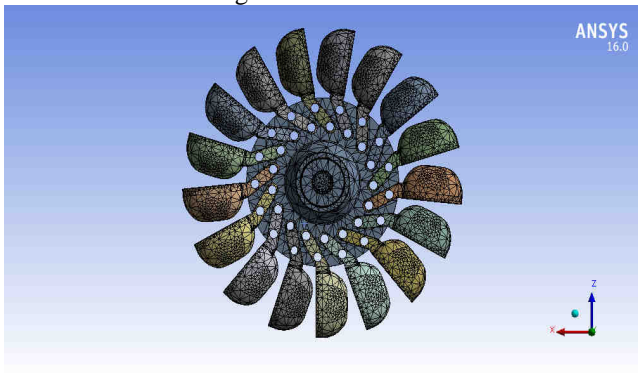


Figure 35 Mesh Generated With Default Mesh Controls

- 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.

F. Specifying the Boundary Conditions

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

- 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.
- 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
- Click on the geometry selection box to display the apply and cancel buttons
- Choose the face tool from the select toolbar to enable selection of faces.

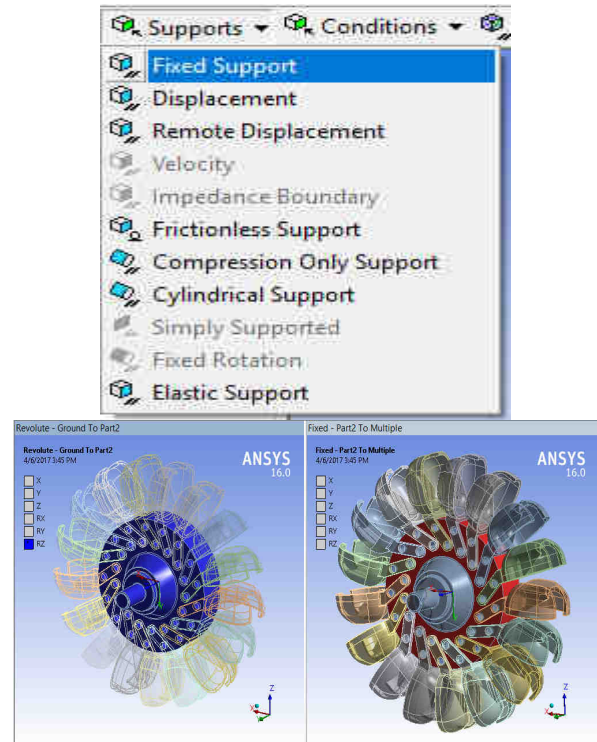


Figure 36 choosing the revolute connection tool fixed connection tool

- click on body to body connection and select cylindrical body to body connection type. select required face in mobile and reference in details of cylindrical body to body connection type.
- click on body to ground connection and select revolve body to ground connection type. select required face in mobile and reference in details of cylindrical body to ground connection type.
- 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

8. Click on the geometry selection box to display the apply and cancel buttons, if they are not already displayed.
9. Next, select the circular face on the right of the model, as shown in figure.
10. Choose the apply button from the geometry selection box; the cylindrical face turns red indicating that the force load is applied.
11. In the details of "pressure" window, expand the definition node, if it is not already expanded.

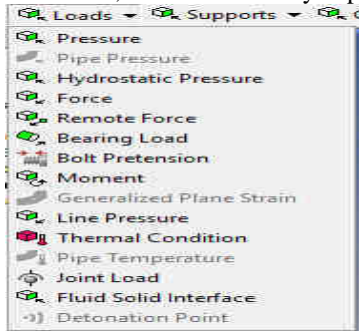


Figure 38 The Loads Drop-Down

12. Select vector from the define by drop-down list, if it is not already selected.
13. In the details of "force" window, click on the right arrow next to the magnitude edit box; a fly-out is displayed.
14. 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,
15. In the magnitude edit box, enter number.
16. 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.

17. Select any vertical edge on the model, as shown in figure 949, to specify the direction of force application
18. 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
19. Specified by choosing the flip button available in the graphics screen.

G. Solving the FEA Model and Analyzing the Results

After the boundary and load conditions are specified for the model, you need to solve the analysis. After solving, you will get the total and directional deformations due to the given condition. Also, you will get equivalent stress, maximum principal, and minimum principal stresses.

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

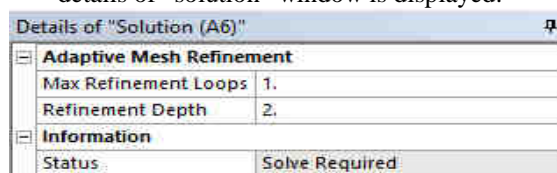


Figure 39 The Details of Solution Window

2. Choose the total tool from the deformation drop-down of the solution contextual toolbar; total deformation is added under the solution node.

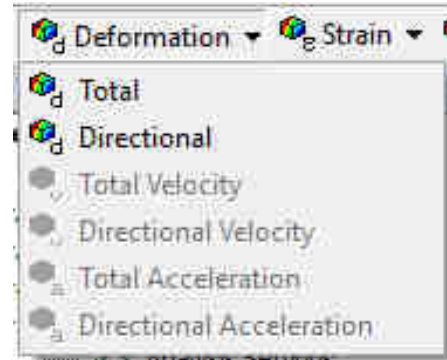


Figure 40 Choosing The Total Tool From The Deformation Drop Down

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 1" his 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.
7. Choose the solve tool from the standard toolbar; the parameters are evaluated
8. In the tree outline, select total deformation to visualize the results; the deformed model is shown in the graphics screen
9. In the details of "total deformation" window, expand the results node, if it is not already expanded. Note that the maximum and minimum deformations displayed are respectively.

H. Results for Aluminum Alloy

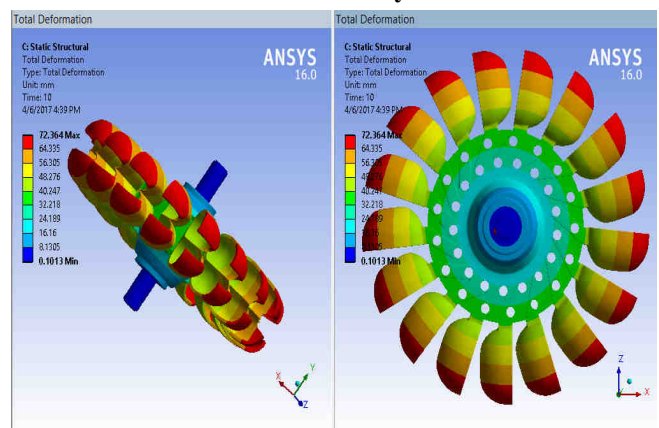


Figure 41 The Maximum And Minimum Values Of Total Deformation AT ALL BODIES Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

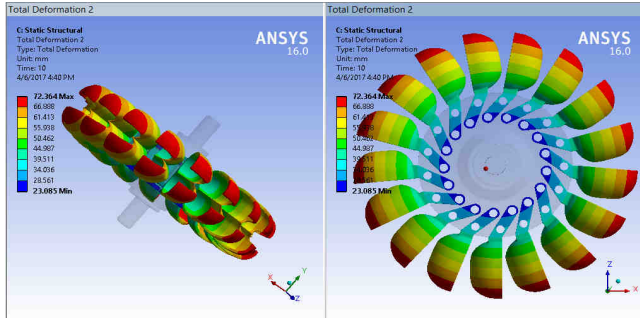


Figure 42 The Maximum And Minimum Values Of Total Deformation AT CAPS Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

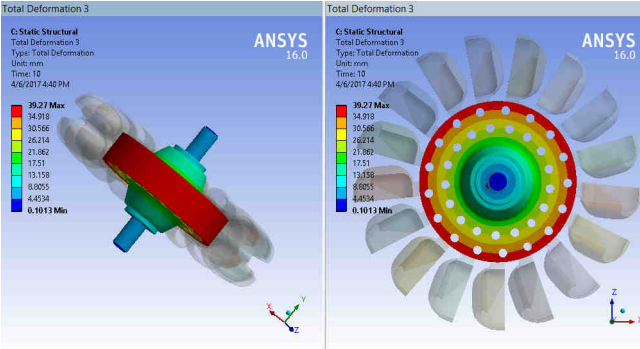


Figure 43 The Maximum And Minimum Values Of Total Deformation AT WHEEL Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

10. 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.

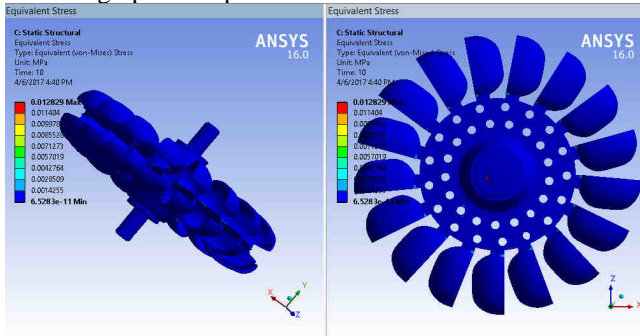


Figure 44 The Maximum And Minimum Values Of Total Deformation AT ALL BODIES Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

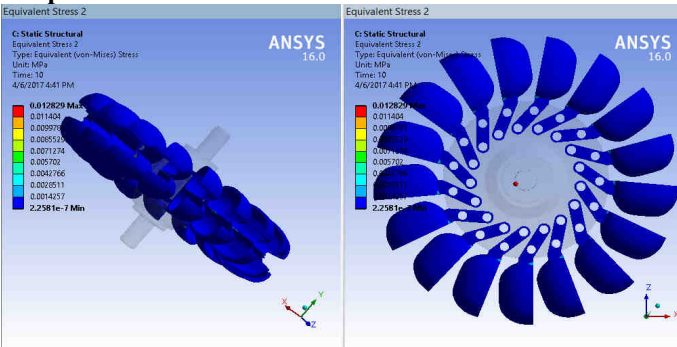


Figure 45 The Maximum And Minimum Values Of Total Deformation AT CAPS Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

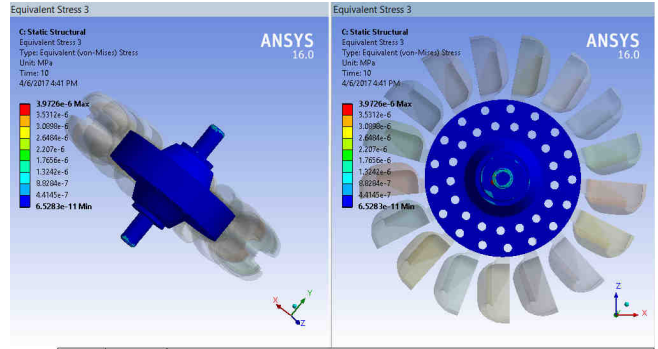


Figure 46 The Maximum And Minimum Values Of Total Deformation AT WHEEL Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen.

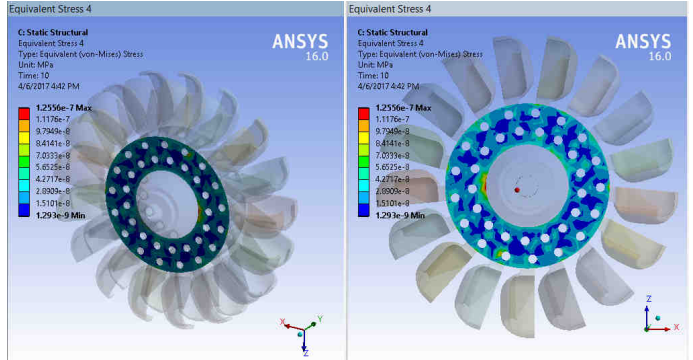


Figure 47 The Maximum And Minimum Values Of Equivalent Or Von-Mises Stress AT CONTACT REGION Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen

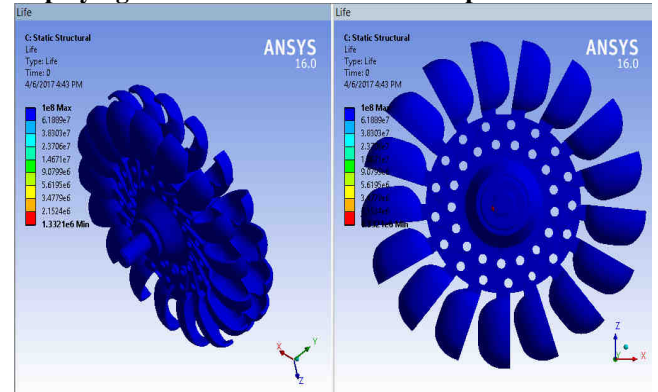


Figure 48 The Maximum And Minimum Values Of Life Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen

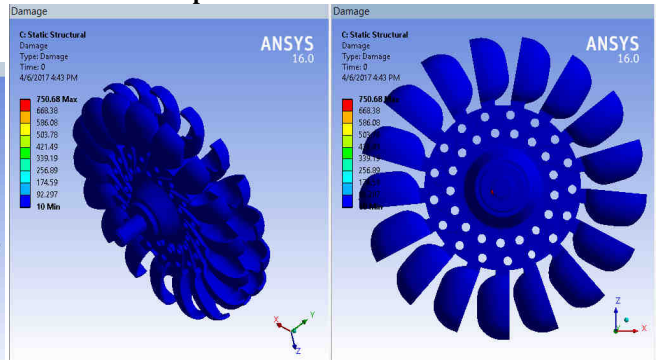


Figure 49 The Maximum And Minimum Values Of Damage Can Also Be Obtained From The Legend Displaying The Color Bands In The Graphics Screen

11. 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.

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

I. Result From The Workbench

7) For Aluminum Alloy

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.1013	72.364	mm	10.
Equivalent Stress	6.5283e-011	1.2829e-002	MPa	10.
Total Deformation 2	23.085	72.364	mm	10.
Total Deformation 3	0.1013	39.27	mm	10.
Equivalent Stress 2	2.2581e-007	1.2829e-002	MPa	10.
Equivalent Stress 3	6.5283e-011	3.9726e-006	MPa	10.
Equivalent Stress 4	1.293e-009	1.2556e-007	MPa	10.
Life	1.3321e+006	1.e+008	Units Unavailable	0.
Damage	10.	750.68	Units Unavailable	0.

Table 3 Resultant Summary From Ansys Workbench

CHAPTER-5

V. STEADY STATE THERMAL ANALYSIS

A. 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.

B. 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

1. Open an existing project and save it with a different name
2. Change the unit and generate the mesh.
3. Apply thermal boundary conditions
4. Analyze the results.
5. Save the project and exit Ansys workbench.

C. 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.
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 are 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.

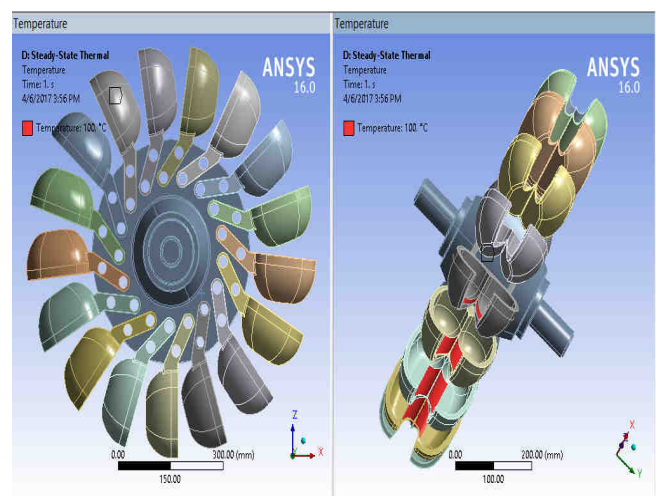


Figure 50 Windows Displayed On Selecting Temperature From The Steady-State Thermal Node In The Tree Outline

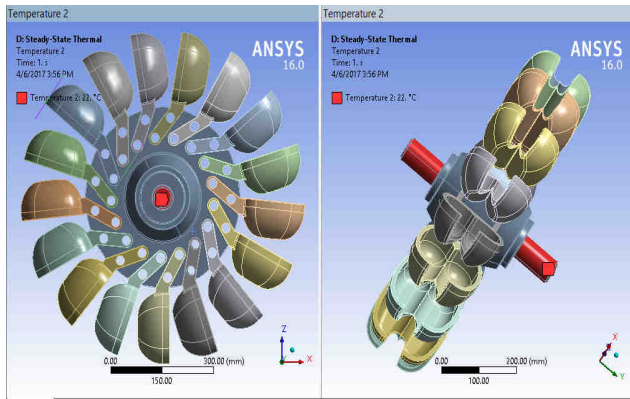


Figure 51 Window Displayed On Selecting Temperature From The Steady-State Thermal Node In The Tree Outline

D. Analyzing the Results

8) For Aluminum Alloy

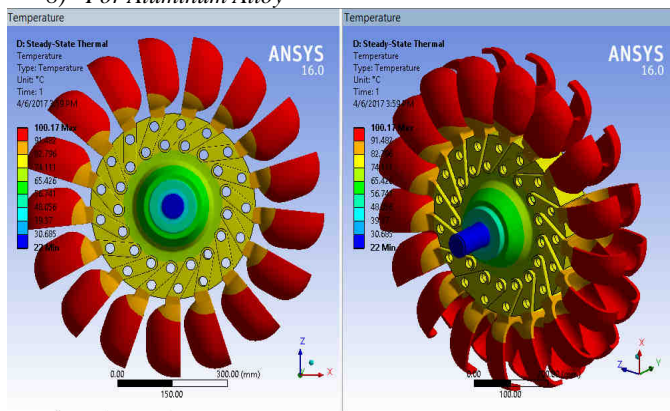


Figure 52 Total Temperature With A Yellow Thunderbolt Is Added Under Die Solution Node

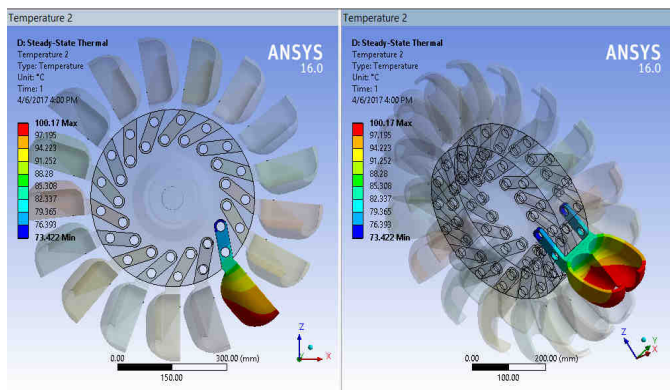


Figure 53 Total Temperature With A Yellow Thunderbolt Is Added Under Die Solution Node

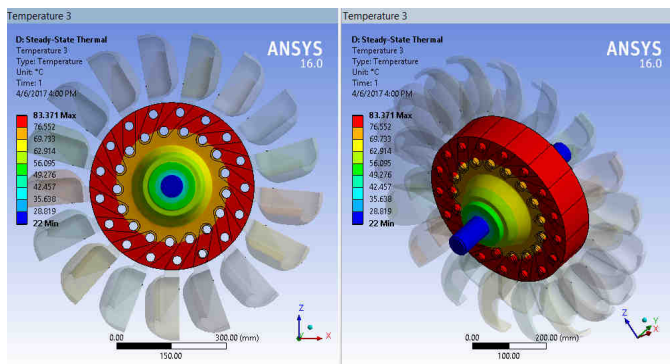


Figure 54 Total Temperature With A Yellow Thunderbolt Is Added Under Die Solution Node

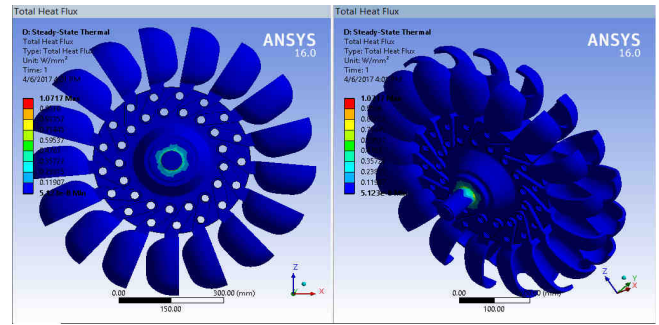


Figure 55 Total Heat Flux With A Yellow Thunderbolt Is Added Under Die Solution

1. In the details of “total heat flux” window, select the geometry selection box to display the apply and cancel buttons.
2. Choose the body tool from the select toolbar and then select the model,
3. 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.
4. 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.
5. Exit the mechanical window to display the workbench window.

CHAPTER-6

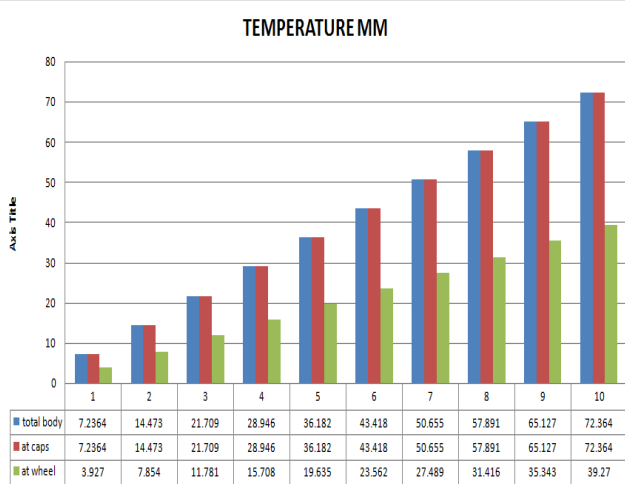
VI. RESULTS

1. From Modal Analysis

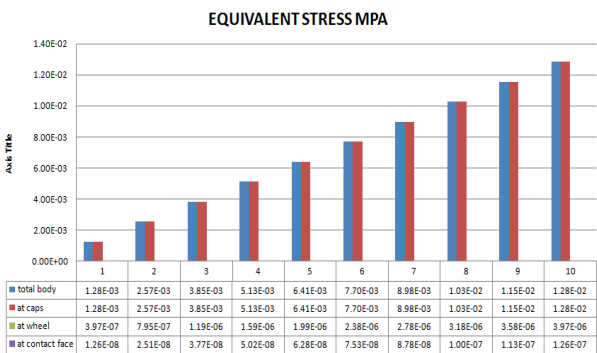
S. No	Modes	Total Deformation Mm	Frequency Hz
1	1	2.2036	71.422
2	2	3.7335	78.581
3	3	1.8934	124.89
4	4	4.646	154.23
5	5	1.9918	160.91
6	6	5.1832	186.76

2. from Static Structural Analysis

S.No	Load / Angle	Temperature Mm			Equivalent Stress Mpa			
		Total	At	At	Total	At	At	At
		Body	Caps	Wheel	Body	Caps	Wheel	Contact Face
1	1	7.2364	7.236	3.927	1.28E-03	1.28E-03	3.97E-07	1.26E-08
2	2	14.473	14.47	7.854	2.57E-03	2.57E-03	7.95E-07	2.51E-08
3	3	21.709	21.71	11.781	3.85E-03	3.85E-03	1.19E-06	3.77E-08
4	4	28.946	28.95	15.708	5.13E-03	5.13E-03	1.59E-06	5.02E-08
5	5	36.182	36.18	19.635	6.41E-03	6.41E-03	1.99E-06	6.28E-08
6	6	43.418	43.42	23.562	7.70E-03	7.70E-03	2.38E-06	7.53E-08
7	7	50.655	50.66	27.489	8.98E-03	8.98E-03	2.78E-06	8.78E-08
8	8	57.891	57.89	31.416	1.03E-02	1.03E-02	3.18E-06	1.00E-07
9	9	65.127	65.13	35.343	1.15E-02	1.15E-02	3.58E-06	1.13E-07
10	10	72.364	72.36	39.27	1.28E-02	1.28E-02	3.97E-06	1.26E-07



Graph 1 for Total Deformation



Graph 2 For Equivalent Stress

3. Form Steady State Thermal Analysis

At applied thermal temperature is 100 C

S. No	Type	Minimum	Maximum
1	Total temperature C	22.	100.17
2	Total temperature C at pelton caps	73.422	100.17
3	Total temperature C at wheel	22.	83.371
4	Heat flux w/mm^2	5.123e-008	1.0717

CHAPTER – 7

VII. CONCLUSION

From the literature review it is seen that, very less work has been done in the field of the dynamic behavior of pelton wheel turbine and their effects in design and operation. Forced vibration is that mode of vibrations in which the system vibrates. Having this mode of vibrations, this may cause a serious damage of the machine.

REFERENCES

1. AmanRajak, et al., dynamic analysis of pelton turbine and assembly, proceedings of IOE graduate conference, Nepal, 2014 pp. 103-109.
2. Lixiangzhang et al. , analysis of dynamic characteristics of the main shaft system in a hydro-turbine based on ANSYS, procedia engineering 31 (2012) pp. 654 – 658
3. Uzma nawaz1, muhammadnaeemrabab, gulrukh, international journal of electrical and computer sciences, 14 (06) , 2014, pp. 1-6
4. Chong-won lee, saving campbell diagram for dynamic analysis of complex rotor systems, icsv20, bangkok, thailand, july 2013, pp. 1-14.

5. Jiaqiliang, p umped storage hydro-plant models for system transient and long-term dynamic studies, institute of electrical and electronics engineers, 2007 pp 1-8.
6. Prakash k. Dhakan and abdulbasheerpombrachalil, design and construction of main casing for four jet vertical pelton turbine, engineering mechanics, vol. 20(2) 2013, pp. 77–88
7. Ze li and lixiangzhang, research on numerical method of flow-induced vibration on spiral casing structure of large-scale hydropower station, international conference on advances in computational modeling and simulation, China, 2012, pp 689-695
8. J.C. Chavez et al., Failure analysis of a Pelton impeller, Elsevier Ltd. Vol. 48, (2015), pp. 297–307
9. Amodpanthee, et al., CFD analysis of pelton runner, international journal of scientific and research publications, vol. 4(8), 2014 ISSN 2250-3153 pp 1-8.
10. Bilal Abdullah Nasir, design of high efficiency pelton turbine for micro hydro power plant, international journal of electrical engineering and technology , vol. 4(1), (2013), pp. 171-183
11. D jost, et al., numerical prediction of pelton turbine efficiency, iop conf. Series: earth and environmental science, in UK, 12 (2010) pp 1-8.