Structural and Modal Analysis of Kaplan Turbine

Mekala Prashanth[1], Mr. K.Sainath[2]

[1]M.Tech. Student, Department of Mechanical Engineering, Sreyas Institute of Engineering & Technology, Hyderabad

[2]Assistant Professor, Department of CAD/CAM, Sreyas Institute of Engineering & Technology, Hyderabad.
---***---

Abstract - *The Kaplan turbine was an evolution of the Francis turbine. Its invention allowed efficient power production. The head ranges from 10 to 70 meters and the output from 5 to 200 MW. Runner diameters are between 2 and 11 meters. 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, an 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 a cad software solid works. By performing static structural analysis total deformation, stress and strain valve, product life and failures etc., can be calculated. Ansys software helps to find the accurate or approximate solutions.*

1.INTRODUCTION - The kaplan turbine is a propeller-type water turbine which has adjustable blades. It changed into developed in 1913 through czech professor viktor Kaplan.Its development approved proficient power generation. the top stages from 10 to 70 meters and the yield from five to 200 MW. Runner diameters are inside the place of 2 and eleven meters. Kaplan mills at the moment are substantially used sooner or later of global in high float, low head electricity manufacturing.In this kaplan sprinter the turns at the bottom of the pointy side are visible; those license the point of view of the cutting edges to be altered at the same time as taking walks. The center comprises of stress driven barrels for editing the factor of view.

The working head of water is low so huge coast costs are permitted inside the kaplan turbine. The water enters the turbine through the guide vanes which can be balanced, for instance, to offer the floats a fitting acknowledgment of spin picked as per the rotor of the turbine. The axial drift of water with a aspect of swirl applies force at the blades of the rotor and loses its momentum, both linear and angular, generating torque and rotation (their product is energy) in the shaft.

1.1 INTRODUCTION OF SOLIDWORKS - Solidworks is a 3d solid modeling bundle which permits users to broaden complete solid fashions in a simulated surroundings for each layout and analysis. In solidworks; your personification musings and explore different avenues regarding particular plans to make 3-d models. Outlining in a demonstrating bundle bargain comprehensive of solid works is valuable since it spares time, exertion, and trade that would out some other case be spent prototyping the format.

Solid Works Components-

Part-

The first and most primary detail of a strong works model is a part.

Elements might be the constructing blocks for all of the fashions that you may create

Assembly-

Assemblies are collections of parts which are assembled in a particular fashion the usage of buddies (constraints).

Any complicated version will typically include one or many assemblies.

Drawing-

The third and final factor in strong works is the drawing.

Drawings are critical due to the fact they offer a preferred manner of sharing your layout.

User Interface-

The solid works application comprises of individual interface devices and capacities that will enable you to make and alter models proficiently, comprehensive of: Home windows abilities: the solid works programming incorporates familiar. Home windows capacities along with dragging and resizing home windows. A number of the equal icons, including print, open, keep, reduce, and paste also are part of the strong works application.

Modeling: Modeling Of Kaplan Turbine - Go to solid works software and select file on top then it will ask the file type or module type part, assembly and drawing. Then select part module then click ok. On part module go to sketch. And select sketch, then create sketch. Later select on exit sketch. After sketch. Go to features and select revolved boss/base. In revolved boss, sketch above profile as a object to revolve and specify axis of revolutions. Then click ok or click on right mark. On part module go to sketch. And select sketch, then create sketch, After sketch. Go to features and select extrude cut boss/base .In extrude cut boss, sketch above profile as a object to depth and specify direction, Then click ok or click on right mark. After extrude cut, go to plane in reference and select front plane as a reference and specify offset distance 0.45metre. On part module go to sketch. And select sketch, then create sketch. After sketch. Go to features and select extrude cut boss/base .In extrude cut boss, sketch above profile as a object to depth and specify direction. After extrude go to fillet in the feature and select two end of wings as shown in figure and specify radius of fillet is 0.3 meter. In feature module and select circular pattern and select the above wings and fillets are the object to pattern. Then select axis as a reference. On part module go to sketch. And select sketch, then create sketch as shown in figure in below. Later select on exit sketcher. After sketch. Go to features and select extrude cut boss/base. In extrude cut boss, sketch above profile as a object to depth and specify direction.

1.2 INTRODUCTION OF ANSYS- Ansys provides simulation solution that enables designers to simulate design performance directly on the desktop. Thusly, it gives quick effective and cost proficient item advancement from outline idea stage to execution approval phase of item improvement cycle. It serves to speeding up and streamlines the item improvement process by helping architects to determine issues connection to basic warm liquid stream electromagnetic impact a blend of these wonders acting together and soon. In ansys, the rudiments of fea ideas, demonstrating and the examining of designing issue utilizing ansys workbench. Likewise, portray of significance apparatuses and ideas given at whatever point required this following simulation streams of ansys.
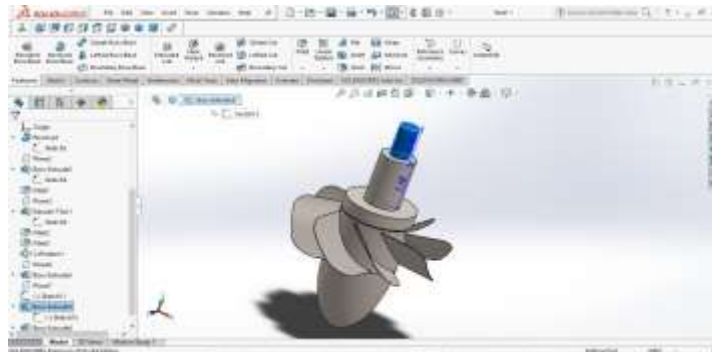
Structural analysis A. Static structural analysis B. Modal analysis C. Transient structural analysis

Thermal analysis A. Steady state thermal analysis B. Transient thermal analysis.

Introduction Of Fea - The concept of fea can be explained through a basic measurement of dimensions In fea reenactment the stacking state of plan and assurance the outline reactions in those conditions. It can be utilized as a part of new item configuration .and also in existing item refinement. A model is separated into a limited number of locales/divisions call components, these components can be of predefinition shapes, for example, triangular, quadrilateral, hexahedron, tetrahedron and soon. This predefined shapes (elements) finds the whole of reactions of all components in display gives the aggregate reactions of finish show.

Degree Of Freedom -

The degree of freedom is defined as the least number of independent coordinate required to define the configuration of the system in space. In the previous example, we are only concerned with displacement and forces. By making one endpoint fixed, you will restrict all degrees of freedom for that particular node. Which means that, there will be no translational or rotational degrees of freedom for that node.

Boundary Conditions –

The boundary conditions are used to elimination the unknowns in the system. At conditions that are feasible is unimportant without the information. Along these lines, the relocation could have been indicated set up of powers as limit conditions and the scientific model could have been illuminated for the powers. As such, the limit condition can help to lessens or take out the obscure in the framework.

Advantage -

It diminishes the measure of model testing, in this way sparing the cost and time. The limited component displaying and investigation are performed in the preprocessor and arrangement stages which if done physically would expend a considerable measure of time and at times, may be difficult to perform.

Limitations -

It does not provide exact solutions. Fea package are costly.An inexperienced user can deliver incorrect answer, upon which expensive decision will be based.

Types of Engineering Analysis –
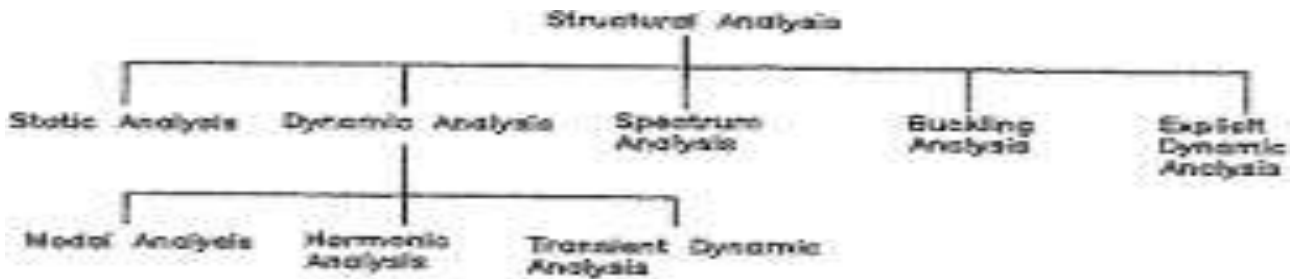
1.2 STATIC STRUCTURAL ANALYSIS –

Create the static basic examination framework, Apply various sorts of materials. Applying of limit conditions, Apply an alternate kind of limitations, Apply various burdens. Generate the outcomes according to required.

Pre-Processing – The pre-preparing of an examination framework includes indicating the material, producing a work, and characterizing limit conditions.

Solution - In an investigation, after pre-preparing (coinciding, determining fabric, and indicating restriction situation) is carried out, the subsequent degree is to illuminate the exam. In ansys workbench, you'll utilize the settle instrument from the same old toolbar to run the solver. The solver keeps jogging out of sight of a product and secures outcomes of an investigation, in mild of the predetermined limit situations.   **Post-Processing -** After the investigation is finished, you have to create the report in the mechanical window. To do as such, pick the report see tab from the base of the designs screen; the ansys report age in advance message is shown on the screen. After at some point, this message vanishes and the report is produced.
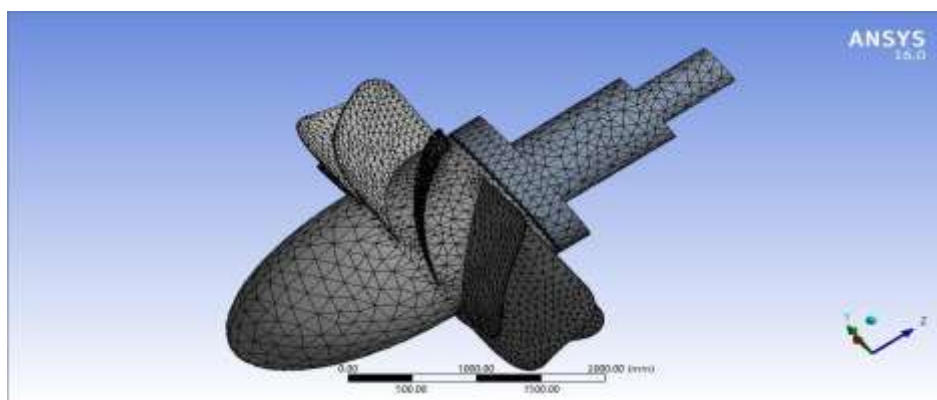
Mesh Generated With Default Mesh Controls
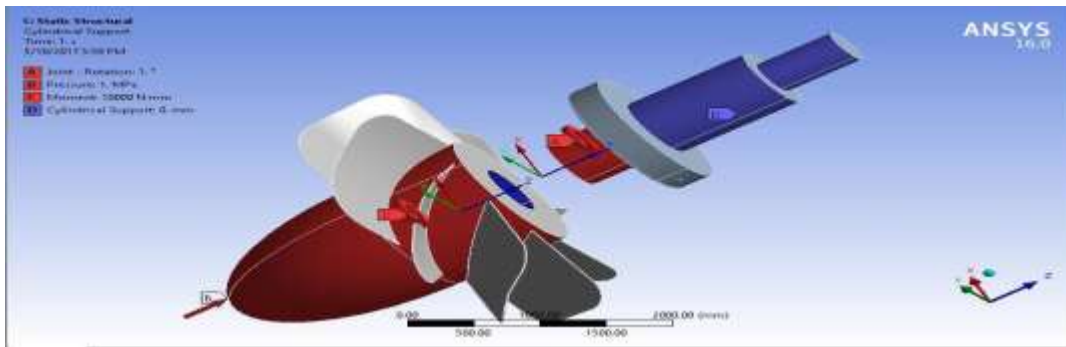
The Violet Face of the Model Displaying the Joint Connections

Solving The Fe Model And Analyzing The Results - After the limit and burden conditions are specified for the model, you have to unravel the investigation. Subsequent to explaining, you will get the aggregate and directional distortions because of the given condition. Likewise, you will get proportionate pressure, most extreme essential, and least foremost anxieties.
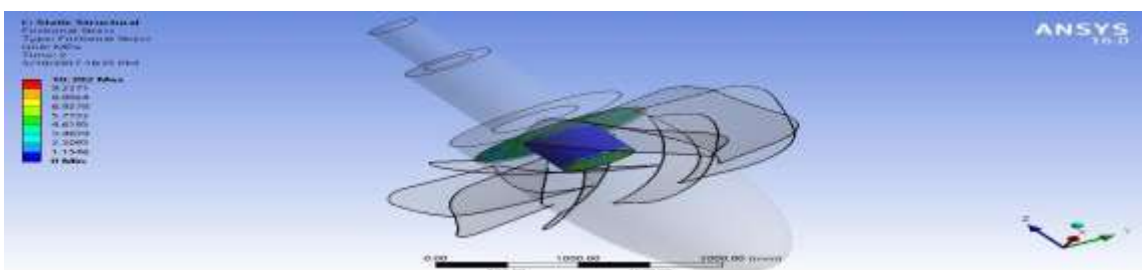
For Stainless Steel -
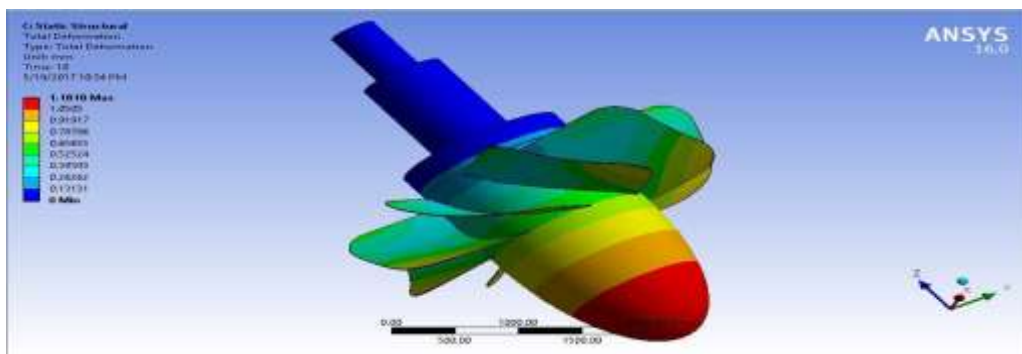
The Maxi And Mini Values Of Total Deformation.
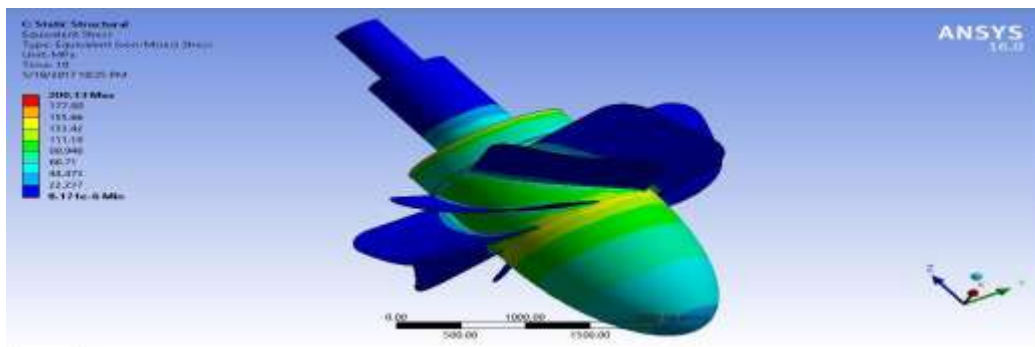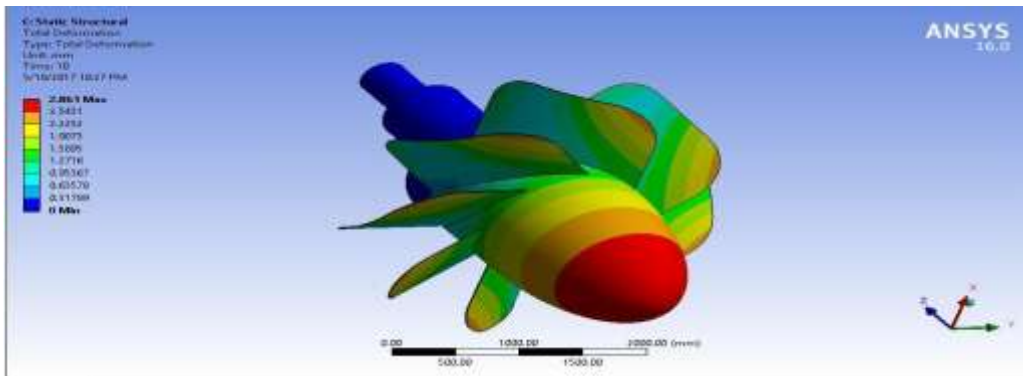

The Maxi And Mini Values Of Equivalent Stress.

The Values Of Frictional Stress At Contact Region.

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	1.1818	mm	10.
Equivalent Stress	8.171e-006	200.13	MPa	10.
Status	1.	3.	Units Unavailable	10.
Frictional Stress	0.	10.392	MPa	2.

Probe: Reactions	X Magnitude	Y Magnitude	Z Magnitude	Total	Units	Time (s)
Joint Probe	3.1537e+005	-1.814e+005	-2.0652e+007	2.0655e+007	N	10.

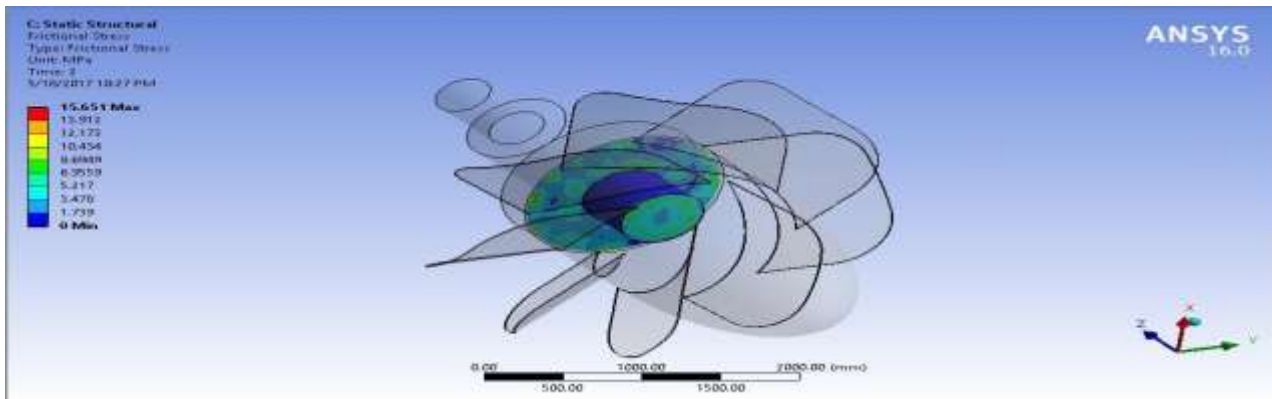**Summary From Workbench**

For Aluminum Alloy –

The Maxi And Mini Values Of Total Deformation
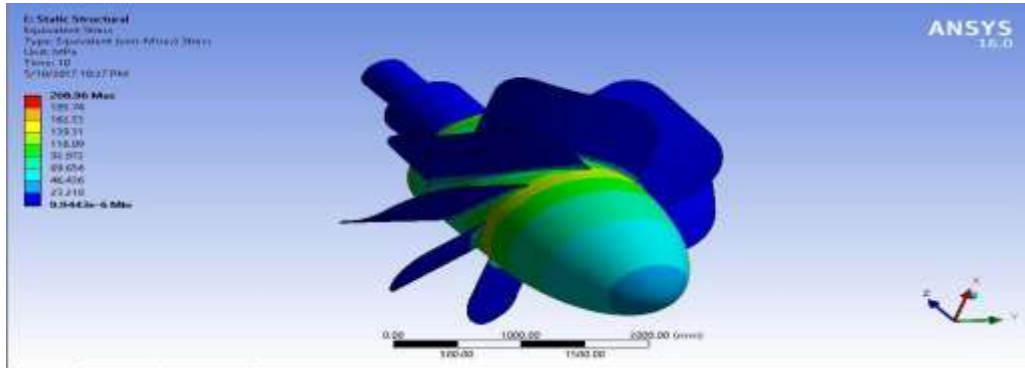
The Maxi And Mini Values Of Equivalent Stress.

The Values OfFrictional Stress At Contact Region.

Results	Minimum	Maximum	Units	Time (s)
Total Deformation	0.	2.861	mm	10.
Equivalent Stress	9.9443e-006	208.96	MPa	10.
Status	1.	3.	Units Unavailable	10.
Frictional Stress	0.	15.651	MPa	2.

Probe: Reactions	X Magnitude	Y Magnitude	Z Magnitude	Total	Units	Time (s)
Joint Probe	4.7388e+005	-1.0268e+005	-2.9501e+007	2.9505e+007	N	10.

Summary From Workbench

**1.3 INTRODUCTION TO MODAL ANALYSIS**

Set the analysis preference, Create or import the geometry into ansys workbench, Define component characteristics (component types, genuine constants, and material properties).Define meshing attributes. Generate a mesh for the model. Specify the examination type, investigation alternatives, and apply loads. Obtain the solution. Review the results. Most of these steps have already been discussed in previous chapters. Adding Modal Analysis System To Ansys Workbench.
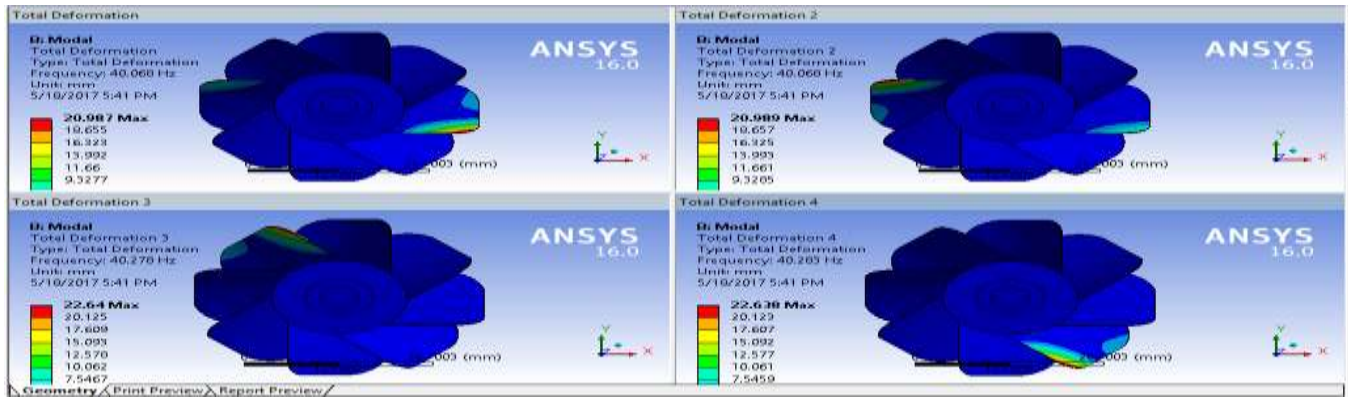
Generating The Mesh for The Model –
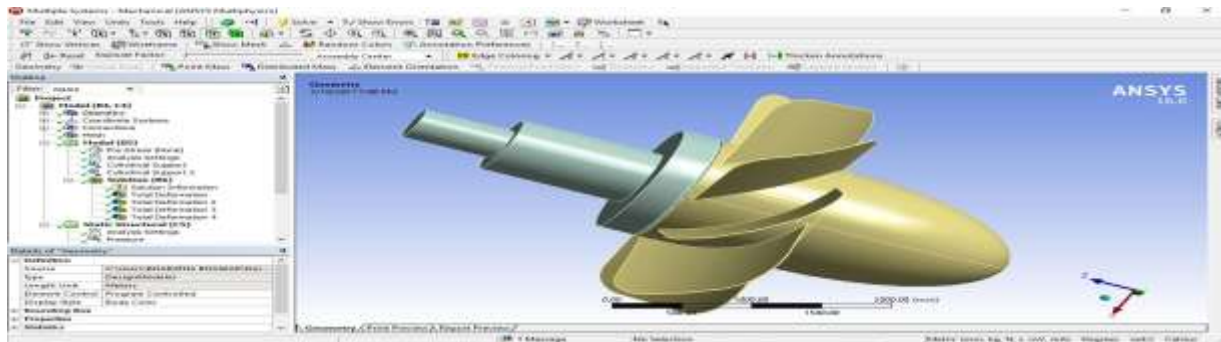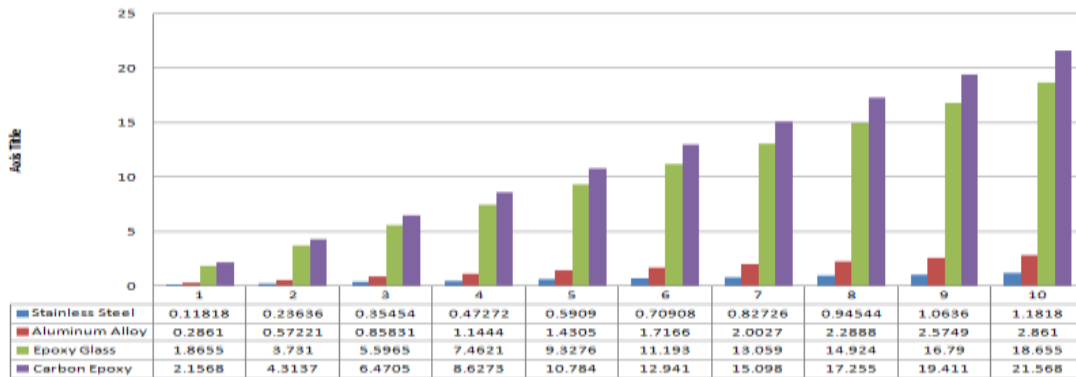

Mechanical Window With The Modal
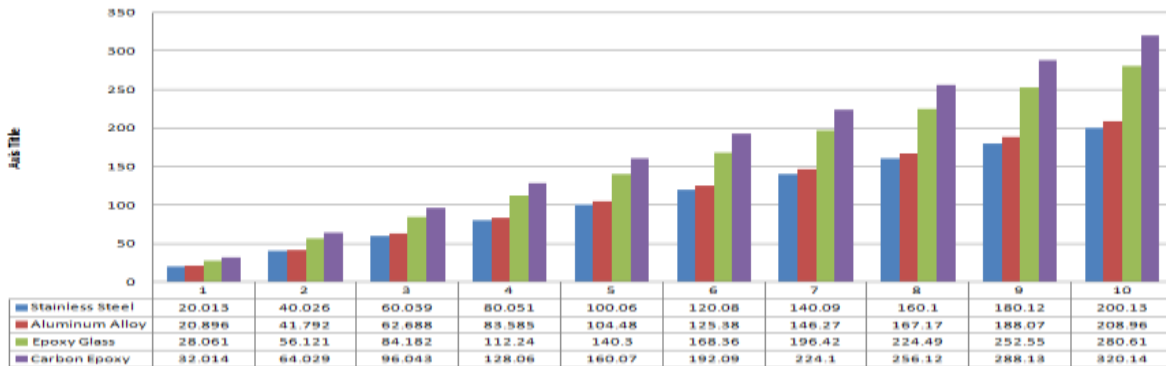
For Stainless Steel
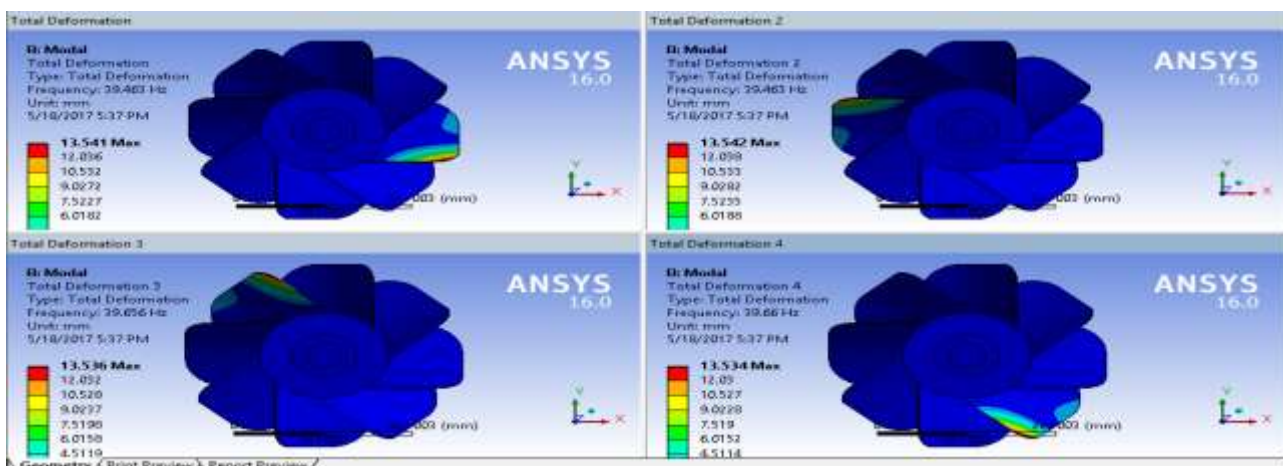
Total Deformation

For Aluminum Alloy

RESULTS -

Graphs and Charts

TOTAL DEFORMATION (MM) WITH CHANGE WITH MATERIAL

	1	2	3	4	5	6	7	8	9	10
Stainless Steel	0.11818	0.23636	0.35454	0.47272	0.5909	0.70908	0.82726	0.94544	1.0636	1.1818
Aluminum Alloy	0.2861	0.57221	0.85831	1.1444	1.4305	1.7166	2.0027	2.2888	2.5749	2.861
Epoxy Glass	1.8655	3.731	5.5965	7.4621	9.3276	11.193	13.059	14.924	16.79	18.655
Carbon Epoxy	2.1568	4.3137	6.4705	8.6273	10.784	12.941	15.098	17.255	19.411	21.568

EQUIVALENT STRESS WITH CHANGE OF MATERIALS

	1	2	3	4	5	6	7	8	9	10
Stainless Steel	20.013	40.026	60.059	80.051	100.06	120.08	140.09	160.1	180.12	200.13
Aluminum Alloy	20.896	41.792	62.688	83.585	104.48	125.38	146.27	167.17	188.07	208.96
Epoxy Glass	28.061	56.121	84.182	112.24	140.3	168.36	196.42	224.49	252.55	280.61
Carbon Epoxy	32.014	64.029	96.043	128.06	160.07	192.09	224.1	256.12	288.13	320.14

REFERENCES

1. Paish, o., small hydro power: technology, Renewable and sustainable energy reviews, 2002.

2. Adam2, p.s.a.i.k., design, simulation and experimental of the very low head turbine. Ijmme-ijens,

3. Ajazbashirjanjua*, m.s.k., and muhammadsaeed, blade profile optimization of kaplan turbine using cfd analysis Volume 32, no. 4: p. 559-574.

4. Parker, g.j., faulkner, s. A., &giddens, e. P., a low head turbine for micro hydropower. 15(2): p. 137-146.

5. Sam williamson, b.h.s., j.d.booker. Low head pico hydro off-grid networks. 2011. Uk.