DESIGN AND ANALYSIS OF AIRCRAFT HUB BY CONSIDERING MATRIX ALLOYS

P.SRIDHAR (M.Tech cad/cam), Roll No: 13TA1DO407, Dhruva Inst. of Engg & Tech, Batasingaram (V), Abdullapurmet (M), R.R Dist-501512, T.S.

BHARAT SHAHAPUR, M.Tech, HOD, Mechanical Dept., Dhruva Inst.of Engg. & Tech., Batasingaram (V), Abdullapurmet (M), R.R Dist-501512, T.S.

ABSTRACT

A wheel hub is the central portion of a wheel through which the axle passes. The wheel hub is the main part of wheel and consists of bearings and axle. The axle is connected the wheel tug.

In this paper, we design wheel hub and analyzing, meanwhile the task is to reduce the stress concentration in the Wheel Hub due to body weight of aircraft and other loads on the wheel hub.

For design of the wheel hub CATIA V5 is used. It is analyzed by using the ANSYS. While analyzing the stress, stress concentration is found in the fixed regions (location of its bolts). The stress concentration is checked in three ways and selecting the one which gives the optimum stress concentration.

The three ways are Design modification, Material changing without design modifications, material changing with design modification. From these three ways we will find out optimum changes in stress concentration.

Keywords: *Ansys, Soliworks, Aircraft Hub, Alloy Materials.*

INTRODUCTION

A wheel hub is the central portion of a wheel through which the axle passes. The wheel hub is the main part of wheel and consists of bearings and axle. The axle is connected the wheel tug.

Figure: The Wheel Hub

BASIC DEFINITIONS OF WHEEL HUB

(a) Axle: An axle is a central shaft for a rotating wheel or gear

(b) Bearings: A bearing is any of various machine elements that constrain the relative motion between two or more parts to only the desired type of motion.

(c) Wheel tug: Wheel tug is a fully integrated ground propulsion system for aircraft which puts a high torque electric motor into the hub of the nose wheel to allow for backwards movement without the use of pushback tugs and to allow for forward movement without using the aircraft's engines.

PROBLEM DEFINITION

Wheel hub is the important component of wheel, it has to with withstand to the load acting over on it. When the wheel rotating the wheel hub has to bear the load acting on the wheel hub and the torque developed by wheel while takeoff or take.

By this the high stress will be developed in the wheel hub, this caucusing to breakages of wheel hub. Due to this the wheel hub is to be with stand to load acting on it.

The Position of Wheel Hub is shown in below figure.

Figure: The Position of Wheel Hub.

Coming to the problem, wheel hub is made up of AZ91D material, the load acting on the wheel hub, is load of the aircraft and the torque and factors which are influencing wheel hub Specification and Loads acting on wheel hub are shown below:

Wheel hub outer diameter (max) = 134.049mm

Wheel hub outer diameter of base (min) = 70.775mm

Wheel hub inner diameter (max) = 68.750mm

Wheel hub inner diameter (min) = 39.5m

Wheel diameter = 762mm

Input Power = 1634 kW

Speed = 18797 rpm

Torque = 2498N-m

Load Force = 17407 N

With respect to above values, applying all factors which are influencing on wheel hub on the Existing AZ91D material component the analysis is generated Maximum amount of Stress is 113.86Mpa. This is analysis shows Maximum amount of Stress 113.86Mpa is Concentrated at one particular curvature. The Presence of stress concentration cannot be totally eliminated but it may be reduced to some extent. So we are considering three methods to reduce stress, by design changing, material changing and both of them changing.

METHODS OF REDUCING STRESS CONCENTRATION

To reduce the stresses which are developed in the wheel hub cover three methods had considered. They are:

- Design modification in existing model
- Material change in existing model
- Design modification and material change in existing model.

Under design modification, where ever Maximum Stress is concentrated at that particular cross-section only design modification had considered. In Existing component.

Maximum Stress is concentrated at one particular at key hole and upper part, by modifying its thickness of hub at upper part and changing diameter of key hole gradually stress also reduce.

Under material change within the place of existing material new material had considered. This new material mechanical properties are very less compared to existing material mechanical properties. With this weight and cost of the component is reduced.

In third method above two methods had considered simultaneously. By analyzing above three methods one method had considered which produce very less stress.

OBJECTIVE OF THE WORK:

The objective of the project work is design and analysis of a aircraft wheel hub to with stand the loads acting on it.

- Model was created in CATIA V5 R20.

- Meshing and Analysis by using ANSYS WORKBENCH.
- Evaluation of stress, deformation, factor of safety under static analysis for Existing Wheel hub.

LITERATURE REVIEW

Among the general effort to reduce CO2 pollution of the atmosphere there is also the question of whether it is possible to tax a passenger aircraft without emissions and self-sufficient on the taxiing field. A positive side effect is the reduction of the ground-based transport vehicles traffic, especially the tractors, and thus a reduction in the risk of accidents. The German Centre for Aerospace has dealt with the question and the Institute of Vehicle Concepts has designed and built a prototype of an electric nose wheel drive for a commercial aircraft Airbus A320. The project was part of the overall research project for the integration of fuel cells in aircraft.

The paper describes the boundary conditions, the requirements, and the design of the electrical machine, the gear and the test result. Design work related to the implementation of new aerospace elements requires the use of 3D-CAD modeling techniques and rapid prototyping, which makes it possible to significantly accelerate the deployment of new solutions. The article presents the possibility of using some methods of rapid prototyping to produce the research model of a wheel hub forming part of the landing gear. Incremental rapid prototyping methods - JS (Jetting System), SLA (Stereo lithography), FDM (Fused Depositing Modelling) – have been characterized with regard to the technology of executing a hub model, the parameters of the manufacturing apparatus, and comparing basic technical data of materials used in the analyzed processes. One of the elements of the process of prototyping was to process data in other equipment for rapid prototyping.

This process consisted of the following steps: defining the parameters of model building, determining the appropriate model settings in the workspace of manufacturing equipment, editing the supporting structure, verifying the subsequent layers through the simulation of model building process, generating output numerical procedures for manufacturing equipment. Manufactured prototypes have been evaluated of dimensional accuracy with the use coordinate measuring machine. Also measured the surface roughness.

Conducted studies were the basis to determine the applicability of various methods of rapid prototyping in the process of research and manufacturing aircraft wheel hub M. GOVINDARAJU, M. JAYALAKSHMI1, K. PRASAD RAO, Scientific paper UDAY CHAKKINGAL, K. BALASUBRAMANIAN Studies on the corrosion of welded areas of friction welded magnesium and its alloy AZ91D are done in view of their applications as structural materials in automotive and aerospace engineering. Friction welding of magnesium-magnesium, magnesium-AZ91D alloy and AZ91D alloy-AZ91D alloy is carried out at the welding speed of 600 RPM which produced a burn length of 2 mm. Corrosion of this welded area is studied using E-log I polarization in 0.1 m ammonium carbonate solution.

It is found that the welded areas are more corrosion resistant than their respective parent samples under similar experimental conditions. Scanning electron microscopic images confirm the grain refinement of welded zones. For better understanding, E-log I polarization studies of pure magnesium and its alloy are also done. Prototyping or model making is one of the important steps to finalize a product design. It helps in conceptualization of a design. Before the start of full production a prototype is usually fabricated and tested.

Manual prototyping by a skilled craftsman has been an age- old practice for many centuries. Second phase of prototyping started around mid-1970s, when a soft prototype modeled by 3D curves and surfaces could be stressed in virtual environment, simulated and tested with exact material and other properties. Third and the latest trend of prototyping, i.e., Rapid Prototyping (RP) by layer-by-layer material deposition, started during early 1980s with the enormous growth in Computer Aided Design and Manufacturing (CAD/CAM) technologies when almost unambiguous solid models with knitted information of edges and surfaces could define a product and also manufacture it by CNC machining.

The historical development of RP and related technologies is presented in table 1. RP process belong to the generative (or additive) production processes unlike subtractive or forming processes such as lathing, milling, grinding or coining etc. in which form is shaped by material removal or plastic deformation. In all commercial RP processes, the part is fabricated by deposition of layers contoured in a (x-y) plane two dimensionally. The third dimension (z) results from single layers being stacked up on top of each other, but not as a continuous z-coordinate.

Therefore, the prototypes are very exact on the x-y plane but have stair-stepping effect in z-direction. If model is deposited with very fine layers, i.e., smaller z-stepping, model looks like original. RP can be classified into two fundamental process steps namely generation of mathematical layer information and 2 generation of physical layer model Finite element method (FEM) is a numerical method for solving a differential or integral equation. It has been applied to a number of physical problems, where the governing differential equations are available.

Introduction to CATIA

CATIA is a suite of programs that are used in the design, analysis, and manufacturing of a virtually unlimited range of product.

CATIA is a parametric, feature-based solid modeling system, "Feature based" means that you can create part and assembly by defining feature like pad, rib, slots, holes, rounds, and so on, instead of specifying low-level geometry like lines, arcs, and circle& features are specifying by setting values and attributes of element such as reference planes or surfaces direction of creation, pattern parameters, shape, dimensions and others.

"Parametric" means that the physical shape of the part or assembly is driven by the values assigned to the attributes (primarily dimensions) of its features. Parametric may define or modify a feature's dimensions or other attributes at any time.

For example, if your design intent is such that a hole is centered on a block, you can relate the dimensional location of the hole to the block dimensions using a numerical formula; if the block dimensions change, the centered whole position will be recomputed automatically.

"Solid Modeling" means that the computer model to create it able to contain all the information that a real solid object would have. The most useful thing about the solid modeling is that it is impossible to create a computer model that is ambiguous or physically non-realizable.

There are six core CATIA concepts. Those are:

- Solid Modeling.
- Feature Based.
- Parametric.
- Parent / Child Relationships.
- Associative.
- Model Centric.

Assembly in CATIA:

(a) Bottom-Up Design (Modeling)

(b) Top-Down Design (Modeling)

(c) Degress of Freedom

(d) Assembly Constraints

(a) Bottom-Up Design (Modeling):

The components (parts) are created first and then added to the assembly file. This technique is particularly useful when parts already exist from previous designs and are being re-used.

(b) Top-Down Design (Modeling):

The assembly file is created first and then the components are created in the assembly file. The parts are build relative to other components. Useful in new designs.

In practice, the combination of Top-Down and Bottom-Up approaches is used. As you often use existing parts and create new parts in order to meet your design needs.

(c)Degrees of Freedom:

An object in space has six degrees of freedom.

- Translation – movement along X, Y, and Z axis (three degrees of freedom).
- Rotation – rotate about X, Y, and Z axis (three degrees of freedom).

(d) Assembly Constraints:

In order to completely define the position of one part relative to another, we must constrain all of the degrees of freedom

(1) CONTACT :

Two selected surfaces become co-planar and face in opposite directions. This constrains 3 degrees of freedom (two rotations and one translation).

(2) OFFSET :

Two surfaces are made parallel with a specified offset distance.

(3) COINCIDENT:

Two selected surfaces become co-planar and face in the same direction. Can also be applied to revolved surfaces. This constrains 3 degrees of freedom (two rotations and one translation). When Align is used on revolved surfaces, they become coaxial (axes through the centers align).

CATIA Modules:

1. Sketcher (2D).
2. Part (3D).
3. Assembly.
4. Drawing and Drafting.
5. Sheet Metal.

6. Rendering.

3D MODEL IS DEVELOPED USING CATIA:

The Wheel Hub developed as shown in figure.

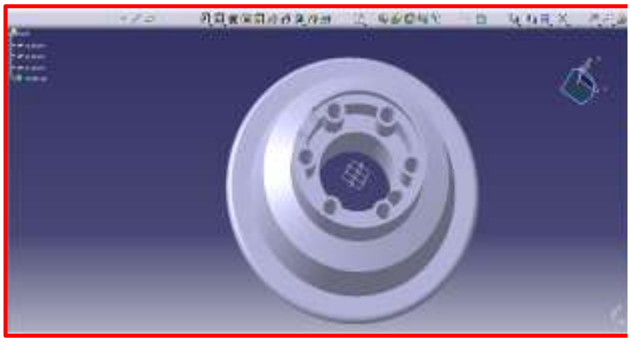

Figure: The Wheel hub developed in Catia.

2D Model Existing Wheel Hub Data:

The existing wheel hub data .the top view is shown in the figure.

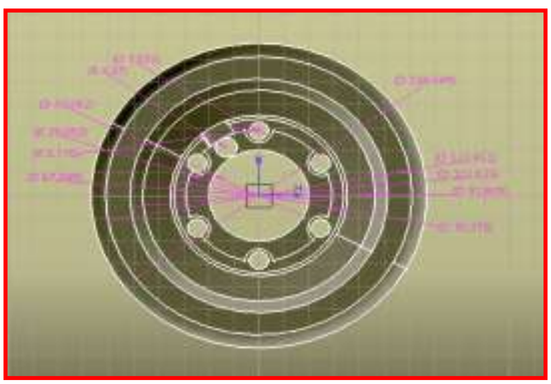

Figure: 2D Source Data for Existing Wheel Hub in Top view.

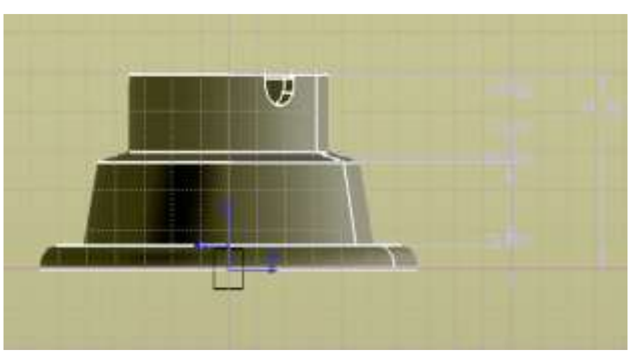

Figure: 2D Source Data for Existing Wheel Hub In Side View.

Creating 2-D Model of Existing Wheel Hub:

With the help of Catia software 2D representation of Existing component will takes place. In Catia, sketcher is the main tool used to represent 2D models. A sketcher is a 2D section of the feature being created. It is a basic 2D shape, and is created on a planar reference. Almost all the models designed in Catia, consist of Datum's, Sketched features, and placed features. However, in this case need a sketch to create any sketched feature, such as Extrude.

The Converting 2D to 3D Drawing of Wheel Hub.

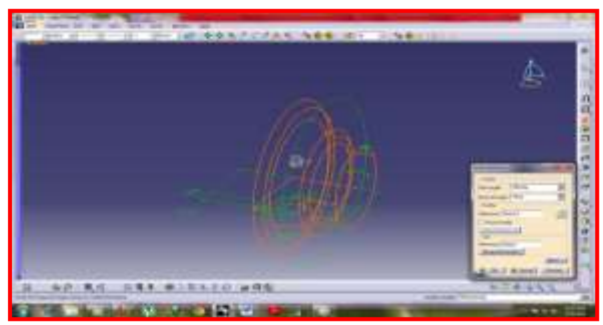

Figure : Converting 2D to 3D Drawing of wheel hub.

INTRODUCTION TO ANALYSIS:

ANSYS is an Engineering Simulation Software (computer aided Engineering). Its tools cover Thermal, Static, Dynamic, and Fatigue finite element analysis along with other tools all designed to help with the development of the product.

The company was founded in 1970 by Dr. John A. Swanson as Swanson Analysis Systems, Inc. SASI.

Its primary purpose was to develop and market finite element analysis software for structural physics that could simulate static (stationary), dynamic (moving) and heat transfer (thermal) problems. SASI developed its business in parallel with the growth in computer technology and engineering needs. The company grew by 10 percent to 20 percent each year, and in 1994 it was sold. The new owners took SASI's leading software, called ANSYS, as their flagship product and designated ANSYS, Inc. as the new company name.

BASIC STEPS IN ANSYS TO USE FROM MESH (Finite Element Software) as shown in figure.

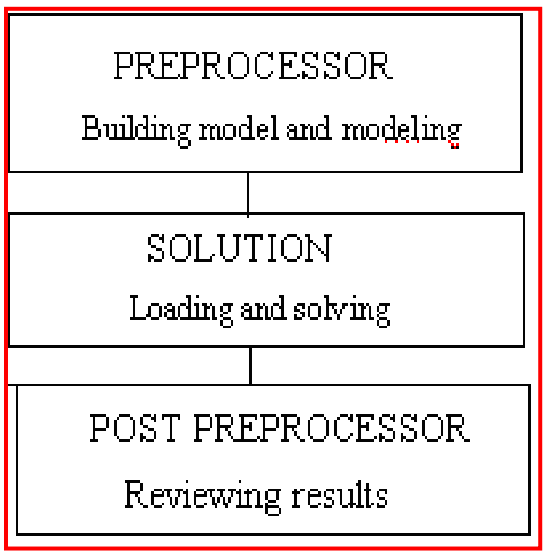

Figure: Basic Steps In Ansys To Use From Mesh (Finite Element Software).

Pre-Processing (Defining the Problem): The major steps in pre-processing are given below:

- ➢ Define key points/lines/ areas/volumes.
- ➢ Define element type and material/geometric properties
- ➢ Mesh lines/ areas/volumes as required.

The amount of detail required will depend on the dimensionality of the analysis (i.e., 1D, 2D, axi-symmetric, 3D).

Solution (Assigning Loads, Constraints, And Solving): Here the loads (point or pressure), constraints (translational and rotational) are specified and finally solve the resulting set of equations.

Post Processing: In this stage, further processing and viewing of the results can be done such as:

- ❖ Lists of nodal displacements.
- ❖ Element forces and moments.
- ❖ Deflection plots.
- ❖ Stress contour diagrams.

Steps involved from Post Processing as shown in figure

RESULTS FOR (41.062mm, 7mm):

(1) AZ91-D DEFORMATION:

The AZ91-D Deformation as shown in the figure.

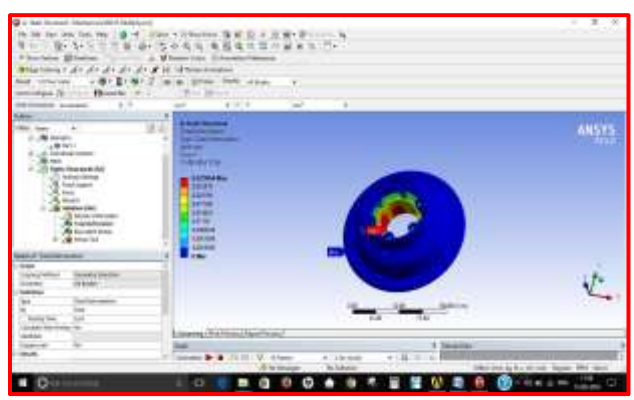

Figure: The AZ91-D Deformation.

SAFETY FACTOR:

The Safety Factor as shown in the figure.

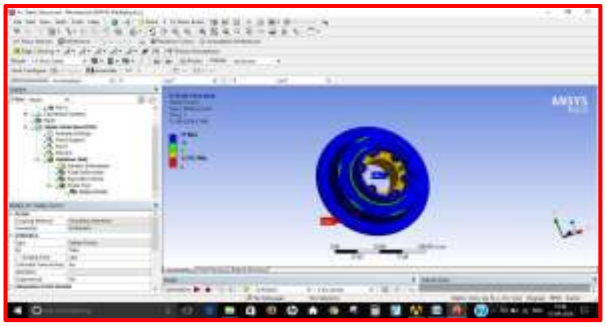

Figure : The Safety Factor.

Results Existing Model (39.75mm) AZ91D Mode1:

(1) MODE1:
The Existing model (39.75mm) AZ91D MODE1 as shown in the figure.

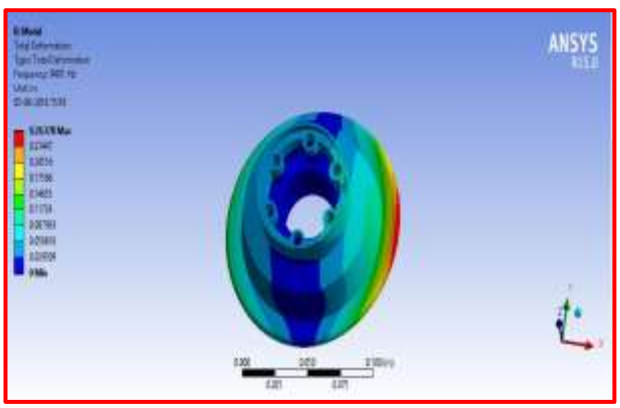

Figure: The Existing model (39.75mm) AZ91D MODE1.

The figure shows that results of natural frequency of AZ91D material and here we have mode1 frequency 9497Hz.

(2) MODE2:
The Mode2 as shown in the figure.

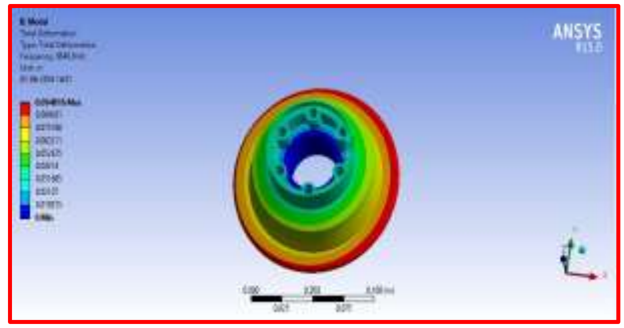

Figure: The MODE2.

The figure shows that results of natural frequency of AZ91D material and here we have mode2 frequency 9548.8Hz.

(3) MODE3:
The MODE3 as shown in the given figure.

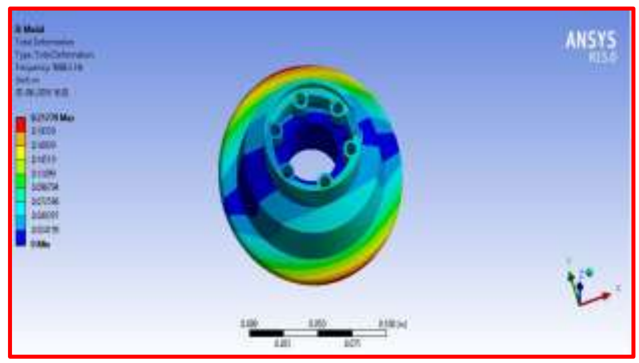

Figure: The MODE3.

(4) MODE4:
The MODE4 as shown in the figure.

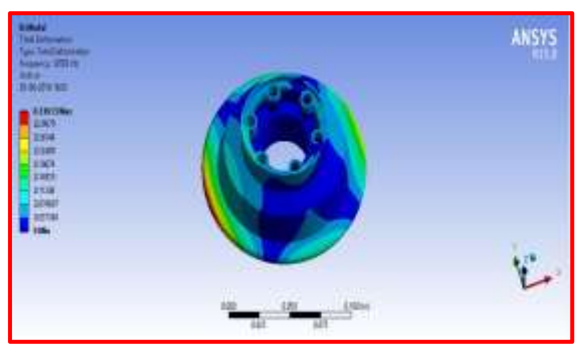

Figure: The MODE4.

(5) MODE5:

The MODE5 as shown in the figure 6.6.

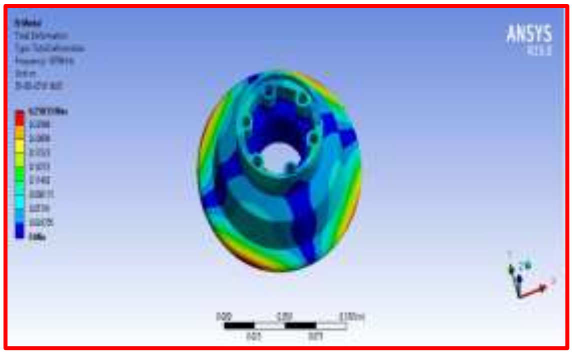

Figure: The MODE5.

MODIFIED DESIGN 2 (41.062mm) AZ91D:

(1) MODE1:

The Modified Design2 (41.062mm) AZ91D MODE1 as shown in figure.

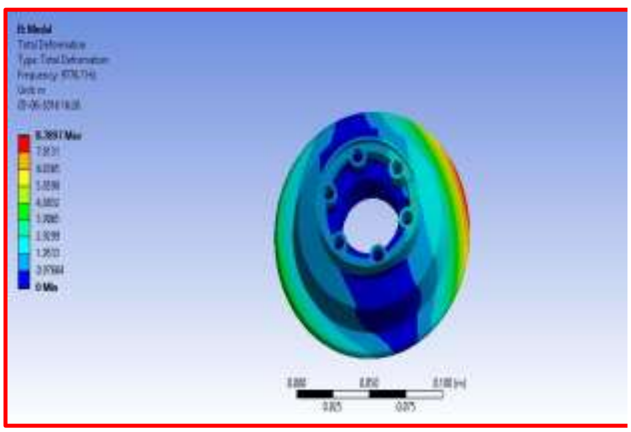

Figure: The MODE1.

(2) MODE2:

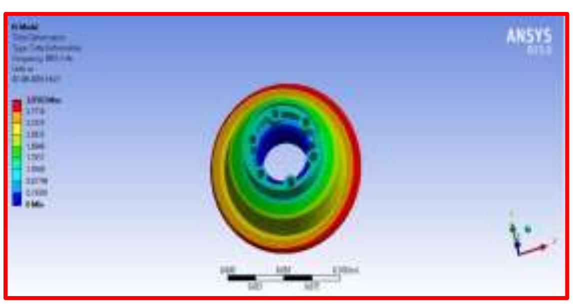

Figure: The MODE2.

(3) MODE3:

The MODE3 as shown in the figure.

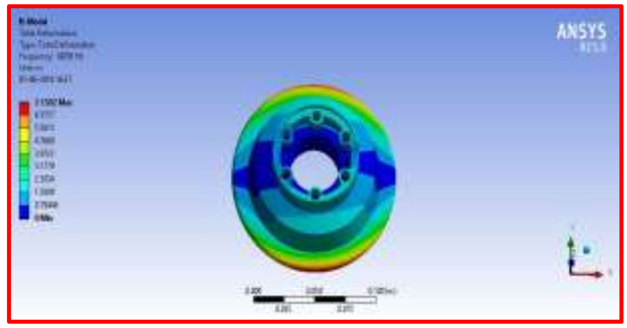

Figure: The MODE3.

(4) MODE4:

The MODE4 as shown in the figure.

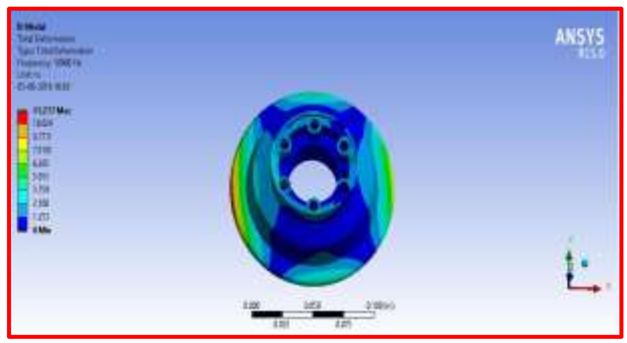

Figure: The MODE4.

(5) MODE5:

The MODE5 as shown in the figure .

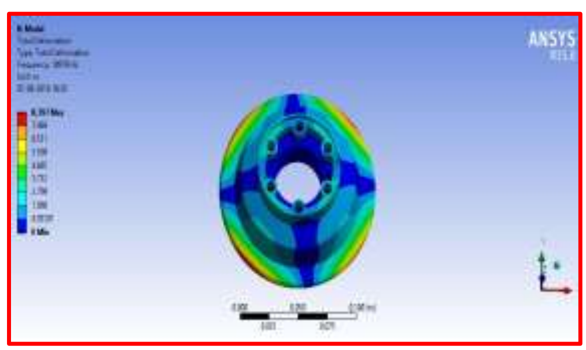

Figure: The MODE5.

AZ92-T6:

(1) MODE1:

The AZ92-T6 MODE1 as shown in the figure.

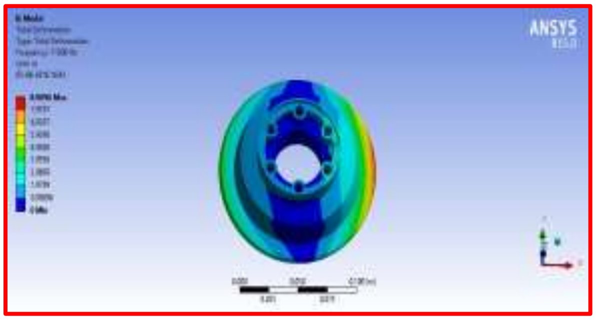

Figure: The AZ92-T6 MODE1.

(2) MODE2:

The MODE2 as shown in the figure.

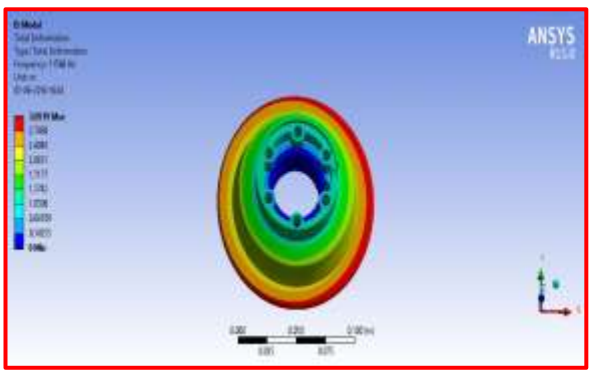

Figure: The MODE2.

(3) MODE3:

The MODE3 as shown in the figure.

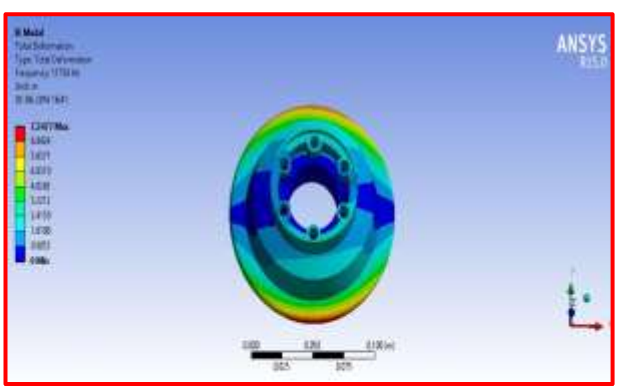

Figure: The MODE3.

(4) MODE4:

The MODE4 as shown in the figure.

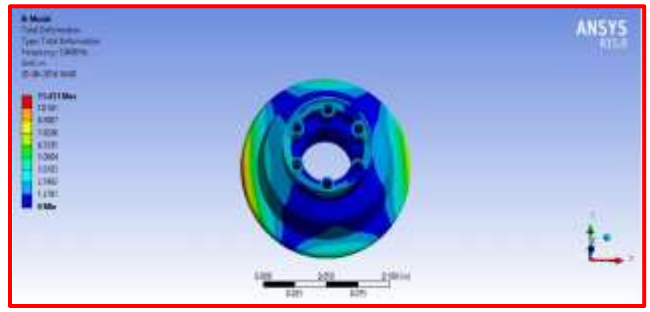

Figure: The MODE4.

(5) MODE5:

The MODE5 as shown in the figure.

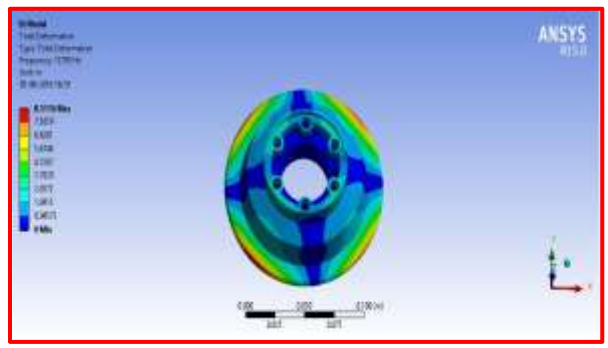

Figure: The MODE5.

CONCLUSION

The main constrain of this project is minimizes of stress. To reduce this stress three methods had considered they are:

- Design modification in existing model.
- Material change in existing model.
- Design modification and material change in existing model.

Among these three methods third method had given best results. In this method out of number of design modified model is D41.062 &D7mm model with material AL-7075-T6 will had produced less stress 109.52Mpa and FOS 4.5927 compared to other materials. This material satisfies two conditions they are less weight and less cost.

The final conclusion of this project is D41.062 &D7mm model with AL-7075-T6 material will give very less stress, and have good safety factor.

In generally we cannot say one object is good or one material is having good strength to weight ratio by only single analysis so here we also checking all these models with dynamic loading conditions and calculating natural frequency values.

From the results al-7075 has more natural frequency values than az91d existing material so that by all these static and dynamic results we can say al-7075 material can be used for aircraft wheel hub.

REFERENCE

1. Ambroziak, M., Technologie szybkiego prototypowania i wytwarzania narzdzi, Pozna2005.
2. Bubicz, M., Prototypowanie – wyzwanie XXI wieku. Cyfrowe czy jednak fizyczne, Projektowaniei konstrukcje inynierskie, Nr 4(7), 2008.
3. Bullinger, H. J., Warschat, J., Fischer, D., Rapid product development — an overview, Computers in Industry 42 (2002), pp. 99-108, Elsevier 2002.
4. Chlebus, E., Techniki komputerowe CAx w inzynierii produkcji, Wydawnictwa Naukowo-Techniczne WNT, Warszawa 2000.
5. Gebhardt, A., Rapid Prototyping, Carl Hanser Verlag, Munich 2003.
6. Haraburda, M., Techniki szybkiego prototypowania w zastosowaniach przemysowych, KIP,Warszawa 2005.
7. Horvath, I., Yang, D., Rapid technologies: solutions for today and tomorrow, Computer Aided Design, Elsevier, 34, pp. 679-682, 2002.
8. Lee, S., Kim, H., Hur, S., Yang, D. STL file generation from measured point data by segmentation and Delaunay triangulation, Computer Aided Design 34 (2002), pp. 691-704,Elsevier 2002.
9. Liu, W.F., Rapid Prototyping and engineering applications – a toolbox for prototypedevelopment, Taylor & Francis Group, Boca Raton, USA 2008.
10. Stroud, I., Xirouchakis, P., STL and extensions, Advances in Engineering Software 31, Elsevier, pp. 83-95, 2000.

BIBLIOGRAPHY

1. Ambroziak, M., Technologie szybkiego prototypowania i wytwarzania narzędzi, Poznań 2005.
2. Bernaczek, J., Śliwa, R.E., Computational methods in the SLA and FDM techniques in the process of production of an aircraft wheel hub prototype, Computer Methods in Materials Science Journal – Informaty.
3. Bubicz, M., Prototypowanie – wyzwanie XXI wieku. Cyfrowe czy jednak fizyczne, Projektowanie i konstrukcje inżynierskie, Nr4(7), 2008.