International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

DESIGN AND ANALYSIS OF WHEEL DISC USING FEA

N FRANCIS[1], A VAMSI KRISHNA[2], S KAMALESH[3]
franciskurnool@gmail.com, adonivamshiksishna301@gmail.com, kamaljntu2012@gmail.com

Abstract

The wheel is an integral part of any automobile. Design of wheel under various loads is to be studied for practical application to ensure safety and smooth running. Here, wheel is subjected to brake loads and mechanical loads due to weight of the vehicle. In cars, Disc brakes and Drum brakes are generally used on front wheels and rear wheels respectively. Due to application of brakes large amount of heat is generated which is distributed across the wheel section.
The objective of the present work is to analyze the stresses induced in the given wheel design due to braking loads and structural loads .For this work, Linear thermal analysis with temperature dependent material properties and Linear static analysis has been performed using finite element software such as ANSYS.
The heat generated by the braking is applied as loading and convective heat transfer has been applied as loading and convective heat transfer has been given as external boundary condition. The temperature plots at various regions are reported. The stress analysis is carried out by switching thermal analysis to structural analysis and Von mises stresses are plotted.
Further, the analysis is also extended to static loading on the wheel. In this, 3D model is generated and this full model is meshed with a 3D solid element, the deflections and stresses are plotted. In static analysis we are considering the loads due to weight of the vehicle, side thrust and pressure due to application of brakes. Finally the stability of the wheel is discussed with the plotted results.

I. INTRODUCTION

The wheel is an assembly of hub, disc (body), rim and spokes (spokes are not necessary in many types).Wheel is the most common commodity in the market that will surely not go out of stock. Its purpose is not just to serve as contact point between the car and the road but also as a means to show off the car's performance. Besides supporting the vehicle's weight and protecting it from road shocks they also transmit power and are steered. The wheels are subjected to braking stresses and side thrusts during their motion. They are, therefore, required to be strong enough. Their construction has undergone drastic changes over the last few decades. From wooden spokes to metallic spokes, pressed steel to light alloys, welded rim to detachable rim, solid wheels to hollow hub type wheels are the novelties of such changes.
Presently the trend is towards use of light-alloy wheels made of aluminium and magnesium alloys. These are light weight, almost corrosion free, casted wheels. They are aesthetically very good looking and are suitable for racing car applications too.

II. BACK GROUNG INFORMATION

Wheel Materials

Now-a-days two types of wheel materials are most common, they are Steel and Aluminium, the other type is

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

Alloy wheels and their description as follows:

Steel Wheels
These wheels can be easily produced in large numbers and are relatively inexpensive. They are usually quiet strong. This costs less than alloy wheels due to ease of manufacturing and lower material costs. These wheels are typically available in black or silver finish depending on the application. Basic styling can often be updated with wheel covers. Steel has a high fatigue limit (the structure can theoretically withstand an infinite number of cyclical loadings at this stress).The biggest disadvantage of this wheel is its weight, lack of performance.

Aluminium Wheels
These wheels are of light weight, good at heat dissipating, it can be easily fabricated in many ways. They are more resistant to corrosion. Pure aluminium has a low tensile strength, but when combined with thermo-mechanical processing, aluminium alloys display a marked improvement in mechanical properties, especially when tempered. Aluminium alloys form vital components of aircraft and rockets as a result of their high strength-to-weight ratio. Aluminium readily forms alloys with many elements such as copper, zinc, magnesium, manganese and silicon. But they are very costly. One important structural limitation of aluminium is its fatigue properties. Aluminium's fatigue limit is near zero, meaning that it will eventually fail under even very small cyclic loadings, but for small stresses this can take an exceedingly long time.

III. INTRODUCTION TO ANSYS

The rapid advances made in computer hardware and software led to significant developments in finite element analysis software. A number of general purpose finite element analysis software packages with a processor capability and facility for the user to have wide choice of several types of elements, analysis of different types of problems like static, dynamic, material and geometric non-linear, coupled situations, heat transfer, interaction problems etc, and pre and post processing features have been developed .the names of some of the popular packages are: ABAQUS, ADINA, ANSYS, ASKA, COSMOS. NISA, PAFEC, SAP, SESAM-80 etc

Steps involved in Ansys
The ANSYS program was introduced by Dr. John Swanson, Swanson analysis systems Incorporated.(SASI) in 1970.The ANSYS software has many finite element analysis capabilities ranging from a simple linear static analysis to complex nonlinear transient dynamic analysis .The 3 basic stages in ANSYS are
1. Pre-processing
2. Solution
3. Post-processing

Pre-Processing
Pre-processor has been developed so that the same program is available on micro, mini, super-mini and mainframe computer system. This allows easy transfer of models from system to other. Pre-Processor is an interactive model builder to prepare the FE(finite element) model and input data. The solution phase utilizes the input data developed by the pre processor, and prepares the solution according to the problem definition. It creates input files to the temperature etc., on the screen in the form of contours. It consists of model generation and discretization into finite elements. Definitions of properties of model are Element Type, Material and various constants such as Young's Modulus, Poissons Ratio etc., dimension of

each elements i.e. Thickness, Moment of Inertia, Area etc.

Geometrical definition

There are four different geometric entities in pre-processor namely key points, lines, areas and volumes. These entities can be used to obtain the geometric representation of the structure. All the entities are independent of other and have unique identification labels.

Material definitions

All elements are defined by nodes, which have only their location defined. In the cases of plate and shell elements there is no indication of thickness. This thickness can be given as element property. Property tables for a particular property set 1D have to be input. Different types of elements have different properties, for e.g.

Beams : Cross sectional area, moment of inertia etc
Shells : Thickness
Springs : Stiffness
Solids : None

The user also needs to define material properties of the elements. For linear static analysis, modulus of elasticity and Poisson's ratio need to be provided. For heat transfer, coefficient of thermal expansion, densities etc are required. They can be given to the elements by the material property set to 1D.

It is used to determine the displacements, stresses, strains and forces that occur in a structure or component as a applied loads. The governing equation for static analysis is:

$$[K]q = F$$

Where, [K] = Structural stiffness matrix
q = Nodal Displacement vector
F = Concentric loads, thermal loads etc (Nodal Force Vector)

In the solution phase we really end up with governing equations at each node, we obtain the degrees of freedom which would give the approximate behavior of the complete model.

Finite element generation

The maximum amount of time in a finite element analysis is spent on generation of elements and nodal data. Pre-processor allows the user to generate nodes and elements automatically at the same time allowing control over size and number of elements. There are various types of elements that can be mapped or generated on various geometric entities.

The elements developed by various automatic element generation capabilities of pre- processor can be checked for element characteristics that may need to be verified before the finite element analysis for connectivity, distortion-index, etc. Generally, automatic mesh generating capabilities of pre- processor are used rather than defining the nodes individually. If required, nodes can be defined easily by defining the allocations or by translating the existing nodes. Also one can plot, delete, or search nodes.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

Thermal Analysis

Thermal analysis calculates the temperature distribution and related thermal quantities such as amount of heat lost or gained, thermal gradients, thermal fluxes etc., thermal analysis is of two types, they are as follows:

A Steady state thermal analysis determines the temperature distribution and other thermal quantities under steady state loading conditions. A steady state loading condition is a situation where heat storage effects varying over a period of time can be ignored.A Transient thermal analysis determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

Advantages of Ansys

ANSYS finite element analysis software enables engineers to perform the following tasks:

Build computer models or transfer CAD models of structures, products, components, or systems.Apply operating loads or other design performance conditions.Study physical responses, such as stress levels, temperature distributions, or electromagnetic fields.Optimize a design early in the development process to reduce production costs.Do prototype testing in environments where it otherwise would be undesirable or impossible (for example, biomedical applications).The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions and commands.

Modeling

The component wheel is modeled using PROE as shown in the Fig 4.1. Using part drawing option the wheel is generated. First the cross-section of the wheel is drawn, and then this cross-section is arranged at some distance to the axis. This cross section is rotated by 360 degrees to get the basic view of the wheel. This structure is then exported to IGES format for further work

.

Fig 4.1.A Disc Wheel Modeled in PRO-E

IV. Element Input

While starting the analysis we have to define the element type, where the meshed model comprises of those elements. Here we are using two types of elements for solving a problem in static and thermal conditions.

SOLID45

SOLID45 is used for the 3-D modeling of solid structures. The element is defined by eight nodes having three degrees of freedom at each node: translations in the nodal x, y, and z directions. Its properties are defined in the following way.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

UX, UY, UZ

Material Properties

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

Surface Loads
Pressures --

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N), face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-P

Body Loads
Temperatures --

T(I), T(J), T(K), T(L), T(M), T(N), T(O), T(P)

Special Features

Plasticity, creep, swelling, stress stiffening, large deflection, and large strain capabilities.

SOLID70

SOLID70 has a 3-D thermal conduction capability. The element has eight nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 3-D, steady-state or transient thermal analysis. If the model containing the conducting solid element is also to be analyzed structurally, the element should be replaced by an equivalent structural element (such as Solid 45). Its properties are defined in the following way.

Nodes

I, J, K, L, M, N, O, P

Degrees of Freedom

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

TEMP

Material Properties

KXX, KYY, KZZ, DENS, C, ENTH

Surface Loads
Convection or Heat Flux –

face 1 (J-I-L-K), face 2 (I-J-N-M), face 3 (J-K-O-N),face 4 (K-L-P-O), face 5 (L-I-M-P), face 6 (M-N-O-PTEMP)

Body Loads
Heat Generations --

HG(I), HG(J), HG(K), HG(L), HG(M), HG(N), HG(O), HG(P)

Special Features

Birth and death

Assigning Material Properties:

1. Aluminium

Density	:	2830Kg/m^3
Modulus of Elasticity	:	71.7Gpa
Poissons ratio	:	0.33
Ultimate tensile strength	:	552Gpa
Yield tensile strength	:	490Gpa
Thermal Conductivity	:	153w/m^{0}C
Specific heat	:	860 j /kg^{0}C

2. Steel

Density	:	7845Kg/m^3
Modulus of Elasticity	:	200Gpa
Poissons ratio	:	0.29
Ultimate tensile strength	:	485Gpa
Yield tensile strength	:	450Gpa
Thermal Conductivity	:	50.7w/m^{0}C
Specific heat	:	515 j /kg^{0}C

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

Geometry Modeling and Meshing:

Generally geometry is modeled in the Ansys with the options in Pre-Processor. But for our analysis we directly export the model into IGES format and it is imported to Ansys for our required analysis.

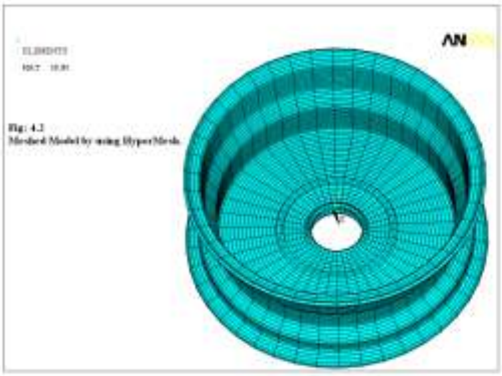


Fig: 4.2
Meshed Model by using HyperMesh.

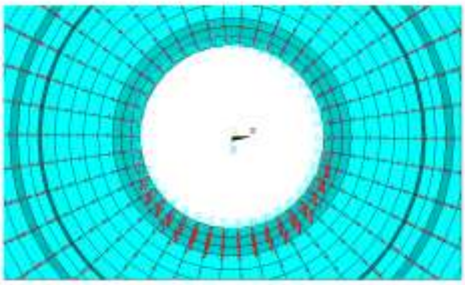


Fig: 5.2 Meshed Model with the body weight applied at half of the hub portion.

Applying Boundary Conditions & Loads:

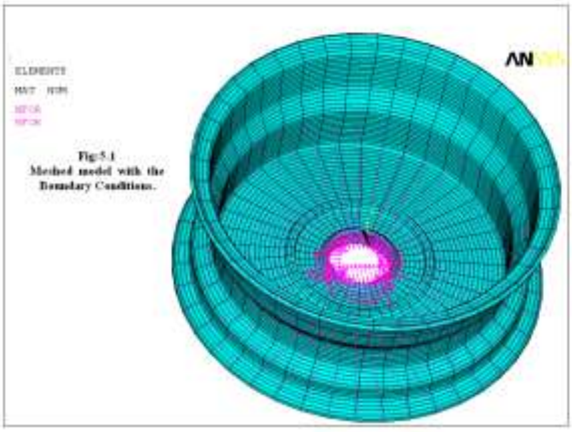


Fig:5.1
Meshed model with the Boundary Conditions.

Constraints for the wheel are given to the nodes at the half of the portion of the hub where the axle is connected to the wheel hub as shown in the Fig 5.1

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

V. MAIN RESULTS

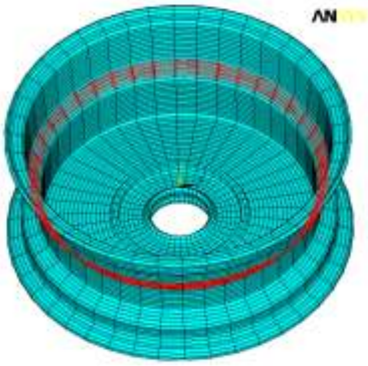


Fig: 5.4
Meshed Model with applied braking pressure around the circumference of the Disc wheel.

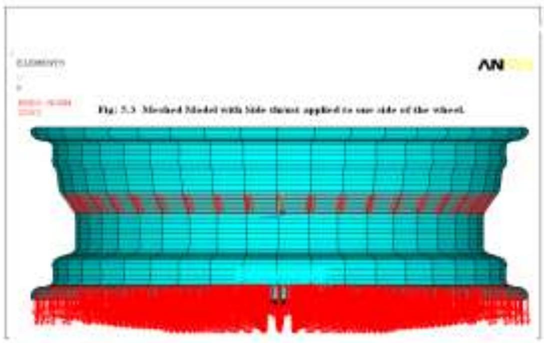


Fig: 5.3 Meshed Model with Side thrust applied to one side of the wheel.

VI. CALCULATIONS

1. For static analysis:

Mass of the car= Mass of the Body + Mass of the Engine
= 2300kg

Weight of the Car = 2300 X 9.81 = 17658 N

Force on each wheel = Weight of Car /4 (as Shown in the Fig 5.2)
= 17658/4 N
= 4414.5 N

Side Thrust = mrw^2 (as Shown in the Fig 5.3)

Where, $m = \frac{(2300+10)}{2}$ Kg [10 Kg is the weight of the Car disc wheel]

r = Radius of curvature = 2 m

w = Angular Velocity of the wheel = 10.57 rad/sec^2

Braking Pressure = 200 psi = 1378.951 Kpa (as shown in the Fig 5.4)

2. For thermal analysis:

Temperature = 200 ^{0}C [Heat generated in the disc brake during the application
Of brakes, shown in Fig 5.5]

Convective film co-efficient = 25w/m^2 ^{0}C

Enthalpy = 86.97kj/kg

Thermal diffusivity = 1.473e-05 m^2/sec

Solving the System:
After applying the loads and constraints, solve the system to obtain the results.
Solution>Solve>Current LS

Viewing the results:
The results are obtained in the Post-Processor stage.
For Deformed shape:
Post-Processor→ deformed shape
For Stresses
Post-Processor→ nodal DOF→ stresses.

VII. RESULTS

Static Analysis (Steel Alloy):

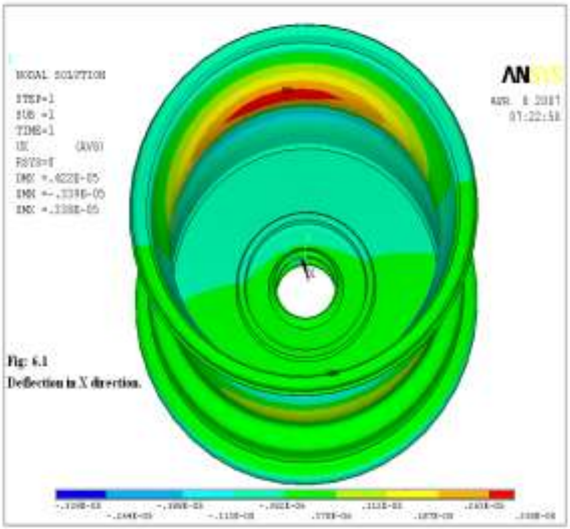


Fig: 6.1
Deflection in X direction.

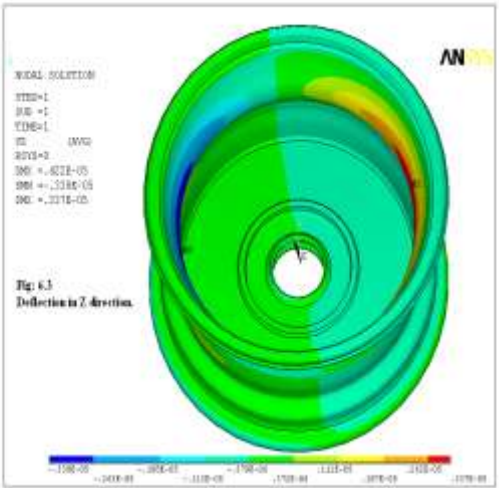


Fig: 6.3
Deflection in Z direction.

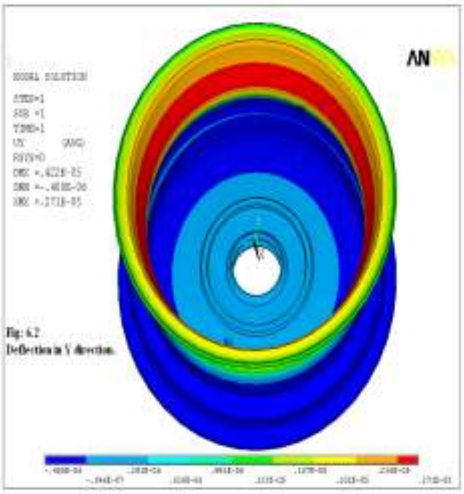


Fig: 6.2
Deflection in Y direction.

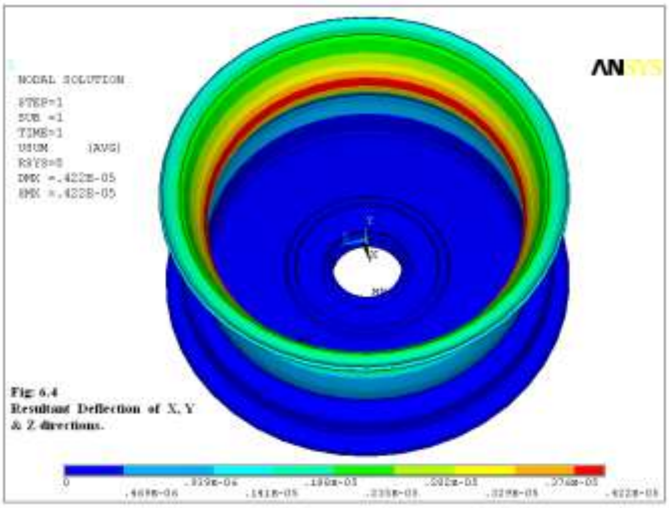


Fig: 6.4
Resultant Deflection of X, Y & Z directions.

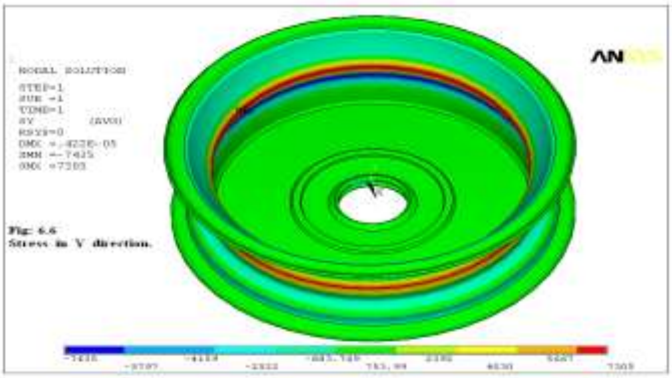


Fig: 6.6
Stress in Y direction.

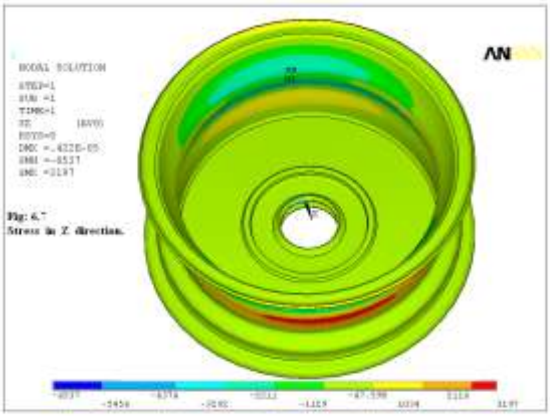


Fig: 6.7
Stress in Z direction.

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

The Min. & Max. values of deflections in X direction are -.339E-05 m, .338E-05 m.

The Min. & Max. values of deflections in Y direction are -.400E-06 m, .271E-05 m.

The Min. & Max. values of deflections in Z direction are -.338E-05 m, .337E-05 m.

The resultant deflection is .422E-05 m.

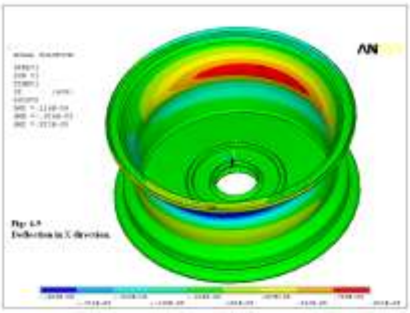
The Min. & Max. values of stresses in X direction are $-6523 N/m^2$, $3188 N/m^2$.

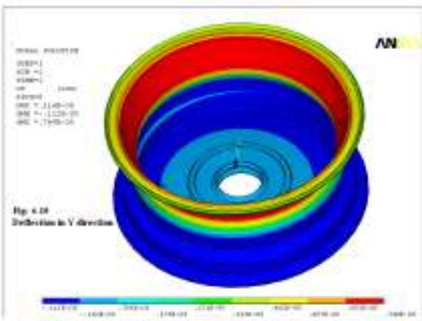
The Min. & Max. values of stresses in Y direction are $-7435 N/m^2$, $7305 N/m^2$.

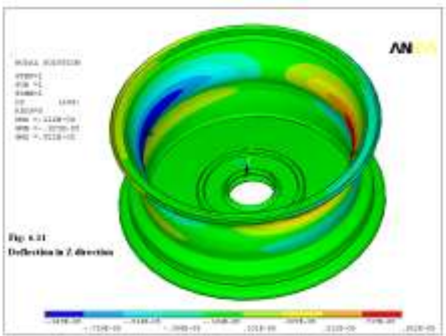
The Min. & Max. values of stresses in Z direction are $6537 N/m^2$, $3197 N/m^2$.

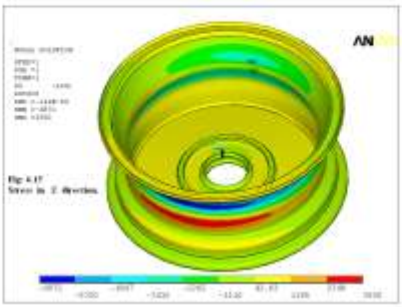
The Min. & Max. values of Von – mises stresses are $2.552 N/m^2$, $10975 N/m^2$

Static Analysis (Aluminium Alloy):









International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

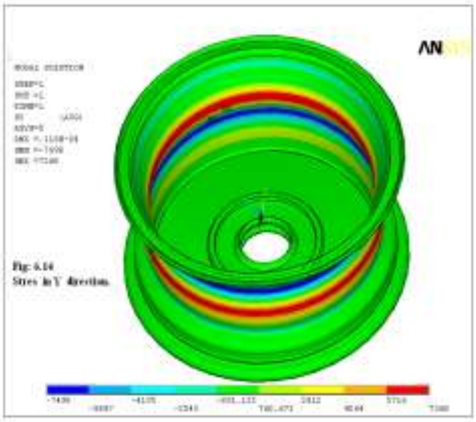


Fig: 6.14
Stres in Y direction.

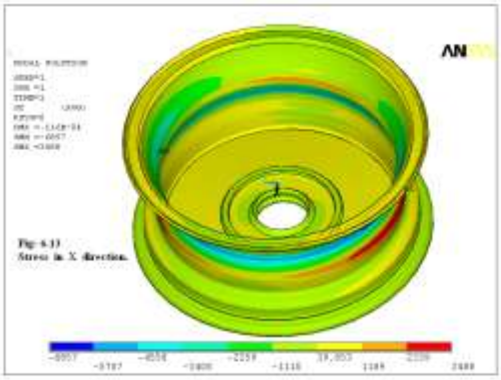


Fig: 6.13
Stress in X direction.

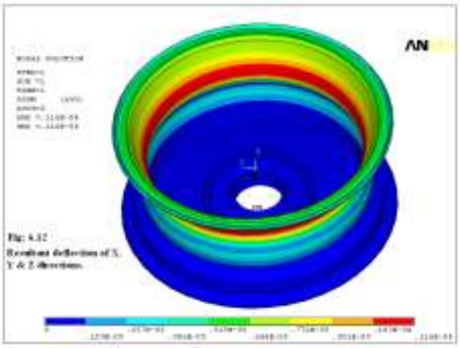


Fig: 6.12
Resultant deflection of X, Y & Z directions.

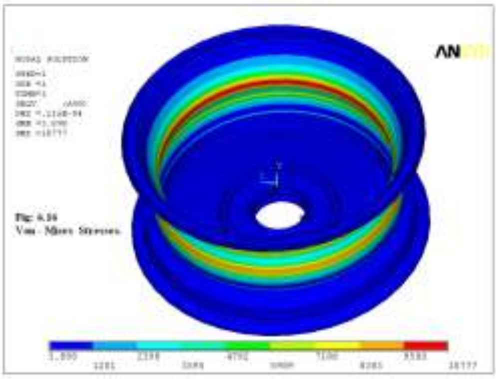


Fig: 6.16
Von - Mises Stresses.

The Min. & Max. values of deflections in X direction are - .926E-05 m, .923E-05 m.

The Min. & Max. values of deflections in Y direction are - .112E-05 m, .749E-05 m.

The Min. & Max. values of deflections in Z direction are - .925E-05 m, .922E-05 m.

The resultant deflection is .116E-04 m.

The Min. & Max. values of stresses in X direction are - 6857N/m^2, 3488N/m^2.

The Min. & Max. values of stresses in Y direction are - 7498N/m^2, 7368N/m^2

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

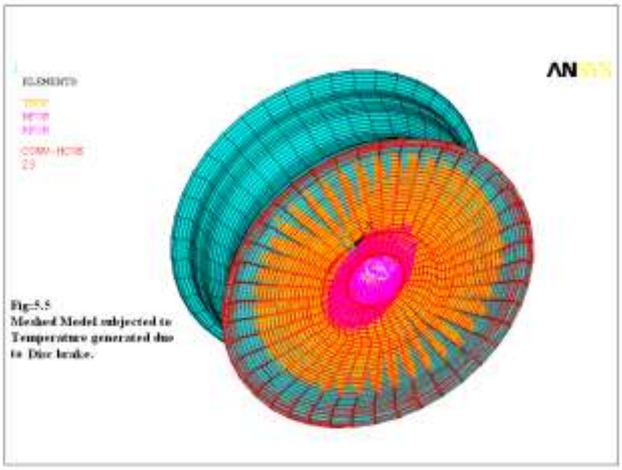
The Min. & Max. values of stresses in Z direction are $-6872 N/m^2, 3500 N/m^2$.

The Min. & Max. values of Von – mises stresses are $3.898 N/m^2, 10777 N/m^2$.

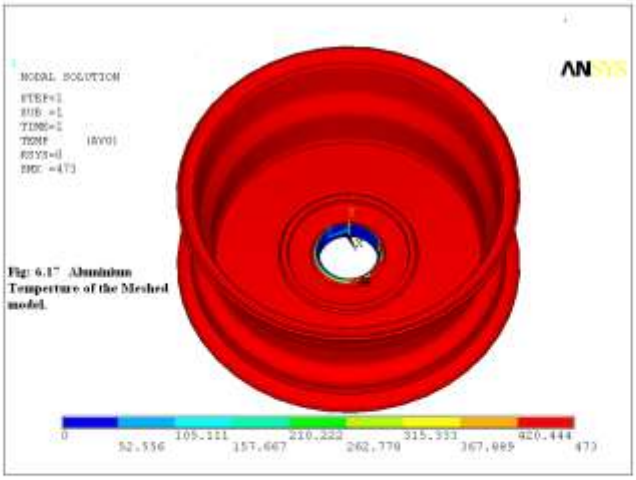
It was observed that the deflections and stresses obtained for the two materials (Steel & Aluminium) are below their values of ultimate stresses. So, we can say that our design is safe for both the materials. It is observed that half of the Steel disc wheel portion has undergone one kind of deflection and other half portion is of other kind. Due to this uneven deflections, there is a chance of crack propagating in the wheel portion when subjected to higher loads, where this is not the case observed in the Aluminium material. So, finally we suggest a wheel material as Aluminium alloy for our analysis.

Thermal Analysis:

Here, we have displayed the thermal results of both Steel and Aluminium materials. It was observed that the temperature generated due to brake drum and thermal gradient values obtained for both materials are same. A uniform temperature was obtained inside the disc wheel portion due to very high convective film coefficient of air. Stresses obtained due to heat loads are negligible when compared to the stresses obtained from static analysis, so we didn't consider the thermal stresses.

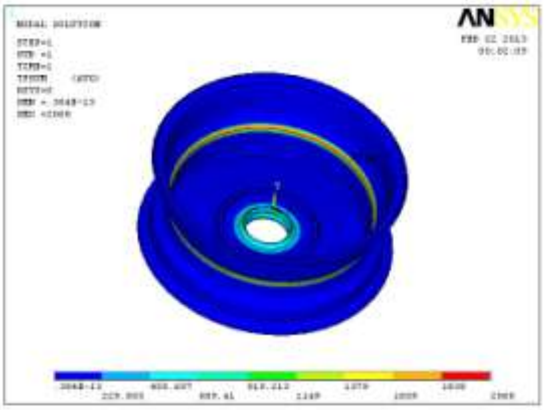


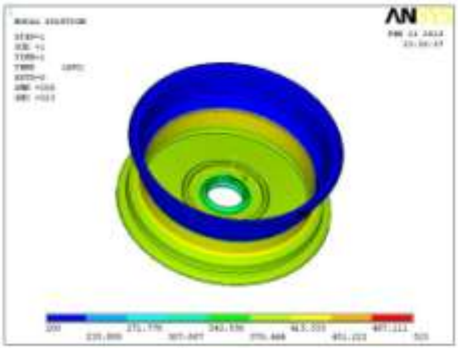
Aluminium Results



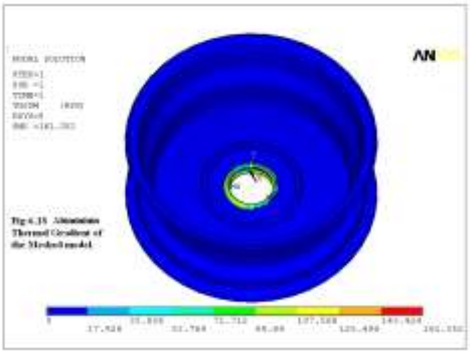
Nodal Temperature of Disc Wheel

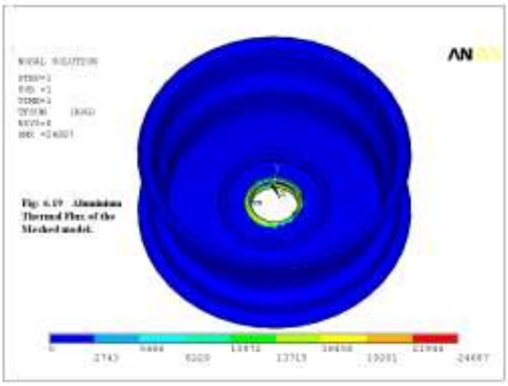
Thermal flux of Disc Wheel **steel rule**





Nodal temperature of Steel **Thermal Flux of Steel**





VIII. CONCLUSIONS

By using Aluminum and steel Materials Static and Thermal analysis was done to see stress and temperatures of Disc wheel. By using aluminum and Steel Materials analyzed couple field analysis (structural and Thermal Analysis By comparing Results of Steel and Aluminum. Finally, we can say Aluminum is better than Steel.Maximum stress induced is within safe limit.Maximum thermal stresses are setup when the temperature difference is maximum from outside to inside

International Journal of Core Engineering & Management (ISSN: 2348-9510)
Special Issue, NCETME -2017, St. Johns College of Engineering and Technology, Yemmiganur

REFERENCES

[1] Introduction to finite elements in engineering, 3rd edition by tirupathi r.chandrupatla, ashok d.belegundu.

[2] Automobile engineering - volume – i, 9th edition, by kirpal singh.

[3] Automobile engineering - volume – ii, power transmission, 1st edition, by anil chikkara.

[4] Automobile engineering, by k.m.gupta.

[5] Ansys help (tutorials), ansys version 9.0 software.

[6] Material properties, aisi data, matweb website.

[7] Strength of materials, 12th edition, by s.ramamrutham, r.narayanan.

[8] Heat transfer, 2nd edition, b.v.karlekar, r.m.desmond.

[9] Thermodynamics, by p.k.nag. engineering tool box website.