IPHV7I3009X
International Journal Of Advanced Research and Innovation -Vol.7, Issue .III
ISSN Online: 2319 – 9253
Print: 2319 – 9245

OPTIMIZATION OF LOCOMOTIVE WHEEL BY USING FINITE ELEMENT TECHNIQUE

#1Pavankumar Kampelli - M.Tech Pursuing,
#2V.Ashok Kumar - Associate Professor, HOD
Department of Mechanical Engineering,
MOTHER THERESSA COLLEGE OF ENGINEERING & TECHNOLOGY, Peddapalli, Karimnagar, TS, India.

Abstract: Damage mechanisms such as surface cracks, plastic deformation and wear can significantly reduce the service life of rolling stock. They also have a negative impact on the rolling noise as well as on the riding comfort. A proper understanding of these mechanisms requires a detailed knowledge of physical structure and specifications of wheel. New specifications are being imposed on railway wheel wear and reliability to increase the time between wheel re-profiling operations, improve safety and reduce total wheel set lifecycle costs. In parallel with these requirements, changes in railway vehicle missions are also occurring. These have led to the need to operate rolling stock on track with low as well as high radius curves, increase speeds and axle loads and contend with a decrease in track quality due to a reduction in maintenance. These changes are leading to an increase in the severity of the wheel/rail contact conditions. So there is a need to optimize the wheel through several considerations such as material properties, shape, design features etc. The optimization is done by preparing a model in PRO/E and analysis in ANSYS.

Keywords: FEM, OPTIMIZATION, FEA, LOCOMOTIVE

I.INTRODUCTION

Indian railway, the prime movers of the nation, has the distinction of being one of the largest railway systems in the world under a single management. its contribution to the nations progress is immeasurable and it has a dual role to play as a commercial organization as well as a vehicle for fulfillment of aspiration of the society at large.

During the last decades substantial progress has been made in the design of railway vehicles and running gear. Tilting trains, high speed trains, active steering wheel sets and other sophisticated solutions have been introduced. In spite of this progress, the mechanics of a railway wheel set remains unchanged and an inappropriate combination of wheel and rail profiles can easily undermine these technological advances. Besides, older equipment has a special need for appropriate combinations of wheel/rail profiles as they do not have high-tech devices that improve their performance. The design of a wheel profile is an old problem and various approaches were developed to obtain a satisfactory combination of wheel and rail profiles. Usually, it is possible to find an optimal combination when dealing with a closed railway system, i.e. when only one type of rolling stock is running on a track and no influence of other types of railway vehicles is present.

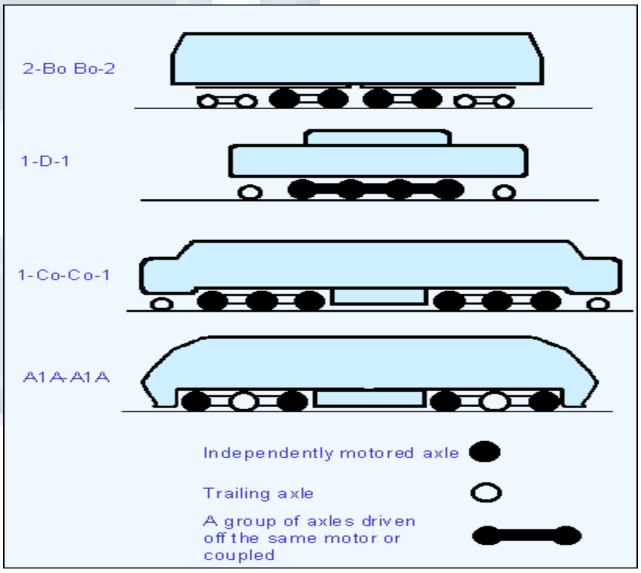
In order to deal with these demands new design methodologies are required that integrate advanced numerical tools for modeling of railway vehicle dynamics and suitable models to predict damage sustained by wheels under typical operating conditions.

WHEEL ARRANGEMENTS

Modern Diesel and Electric Locomotives

Starting with modern equipment and the usual method of describing how the driving and non-driving (carrying or trailing) wheels are distributed under a locomotive; there are two simple basic rules. First, the wheels are not individually identified, only the axles and, second, trailing wheels are allocated numbers and driving wheels are allocated letters. The letter or number refers to the number of axles in a single frame, for example:

Older Designs



The appearance of shelling, spalling and thermal mechanical shelling can be very similar, particularly if the spalling extends around the wheel tread from repeated sliding and skidding, but spalling can often be corroborated by slip marks/patches on the wheel tread. Built-up tread and slid flats For built-up tread (why made code 76), the AAR (2001) specifies that “A wheel is condemnable whenever the tread has built-up metal 1/8 inch or higher on the wheel tread.” This defect is the responsibility of the car owner. Slid flat wheels (why made code 78) are the responsibility of the handling line and are condemnable if the flat spot is “two or more inches in length” or there are “two or more adjoining spots each 1-1/2 inch or over in length.” Clearly, these defects can cause high impact loads and damage to wheels, rails, lading and the car. Out-of-round wheels to be condemnable under

AAR rules, out-of-round wheels must register at least 90,000 pounds on a wheel impact load detector and have a verified out-of-round "run out" of 0.070 inches (AAR, 2001).

TYPES OF WHEELS

	Specification (mm)
Solid wheel for passenger car	915 C - 915 D 950 -1092
Solid wheel for freight car	840B - 840C 840D - 840E
Whole processing wheel for passenger car	915D 915D (S-type) 950 (international true train)
Solid wheel for metallurgical rolling stocks	840 thick web
Rough wheel tyre for locomotive	846 - 1756
Rough wheel tyre for electric locomotive	606 - 686
High quality carbon steel rolled civil ring	outer diameter 600 -2010 (ring, max 2500) inner diameter >500 thickness 50 - 160 height 70 - 200 weight 100 -970 kg.
Alloy structure steel rolled civil ring	
High quality carbon steel hot rolled civil ring	
50 Mn gear ring for truck crane	

Designation of Indian locomotives

Locos, except for older steam ones, have classification codes that identify them. This code is of the form '[gauge][power][load][series][subtype][suffix]'

In this the first item, '[gauge]', is a single letter identifying the gauge the loco runs on:

W = Broad Gauge

Y = Meter Gauge

Z = Narrow Gauge (2' 6")

N = Narrow Gauge (2')

The second item, '[power]', is one or two letters identifying the power source:

D = Diesel

C = DC traction

A = AC traction

CA = Dual-power AC/DC traction

B = Battery electric(rare)

The third item, '[load]', is a single letter identifying the kind of load the loco is normally used for:

M = Mixed Traffic

P = Passenger

G = Goods

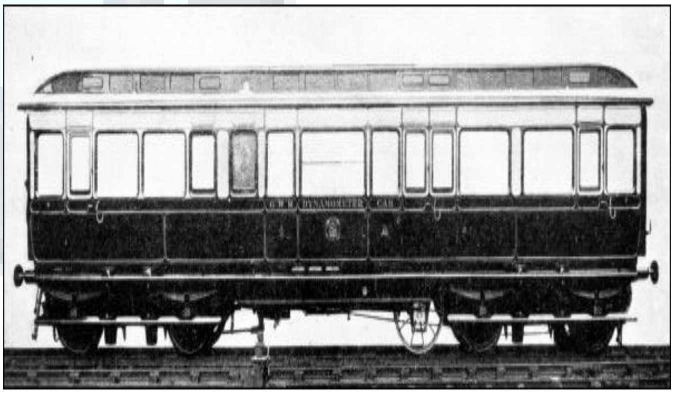
S = Shunting

L = Light Duty (Light Passenger?) (no longer in use)

U = Multiple Unit (EMU / DEMU)

R = Railcar

II. TESTING A LOCOMOTIVE



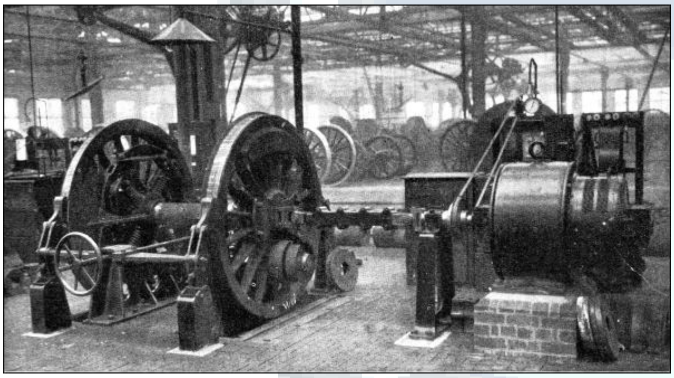
Testing of this description is also used to precede time-table changes, as it is necessary to prove, if accelerations of service are planned, that they will not lead to excessive coal consumption and inefficient working of the engines concerned.

There is another way of testing a locomotive, which reproduces all the conditions of running, in the matter of load and speed, but without requiring that the engine shall be taken for a test journey over a stretch of line with the dynamometer car.

This method is independent of weather variations, especially of the effect of side-winds, which dissipate power through increased friction, and of mist or rain, which makes the rails greasy. Every different type of locomotive can be tested in conditions which are not merely uniform, but which cannot vary. The disadvantage of this method is that it requires the erection of a costly testing-plant. Such plants are in use in the United States, Germany, and France, but the only one of its kind in Great Britain is at the Swindon Works of the Great Western Railway, and this can test only up to 500 horsepower, which fails for short of modern requirements.

One of the moist up-to-date plants is at Vitry, near Paris, to which the London & North Eastern locomotive "Cock o' the North" was sent in December, 1934, in order that exhaustive tests might be made. Along the floor of the testing-plant there are arranged a series of rollers, the position of which can be adjusted until each of the wheels of the engine under test is resting exactly on the centre of a roller. Each of the rollers beneath the driving-wheels of the engine drives a Froude hydraulic brake in its turn the shaft of the brake drives a hydraulic pump, and any increase of the engine's speed also increases the rate of water supply to the brake, so increasing the braking force, and keeping constant the rate of rotation of the engine's driving wheels.

The draw bar or coupling of the engine is then attached to a hydraulic dynamometer, which is anchored to a vertical girder set in concrete. The engine held thus securely in position, is put in steam, started, and worked up to the equivalent of a high speed, travelling, theoretically, some hundreds of miles.



WHEEL BALANCING, an important part of a locomotive's testing which is carried out during construction. This machine, at Swindon locomotive works, is used to ensure that all engine driving wheels shall be capable of spinning without undue vibration, at a speed equivalent to over sixty miles an hour on the track.

III.FEM

FINITE ELEMENT METHOD

The Finite Element Method has become a powerful tool for the numerical analysis technique for obtaining approximate solutions of a wide range of engineering problems.
Because of its diversity and flexibility as an analysis tool, it is receiving much attention in engineering schools and industries. In more and more engineering situations, we find that it is necessary to obtain approximate solutions to problems rather than exact closed from solution.
It is not possible to obtain analytical mathematical solutions for any engineering problems. An analytical solution is a mathematical expression that gives the values of the desired unknown quantity at any location in the body, as consequence it is valid for infinite number of location in the body. For problems involving complex material properties and boundary conditions, the engineering study shifts towards numerical methods that provide approximate, but acceptable solutions.

The finite element method has become a powerful tool for the numerical solution of a wide range of engineering problems. It has developed simultaneously with the increasing use of the high speed electronic digital computer and with the growing emphasis on numerical methods for engineering analysis.
The fundamental areas that have to be learn for the working capability of finite element method include:

- Matrix algebra.
- Solid mechanics.
- Variational methods
- Computer skills.

Matrix techniques are definitely most efficient and systematic way to handle algebra of finite element method. Basically matrix algebra provides a scheme by which a large number of equations can be stored and manipulated. since vast majority of literature on finite element methods treat problem in structural mechanics. It is useful to consider the finite element procedure basically as a variational approach.
The term “finite element distinguishes the techniques from the use of infinitesimal “differential elements” used in calculus, differential equations. the method is also distinguished from finite differential equations, for which although the steps in to which space is divided in to finite elements are finite in size, there is a little freedom in the shapes that the discrete steps can take. FEA is a way to deal with the structure that are more complex than dealt with the analytically using the partial differential equations. FEA deals with the complex boundaries better than finite differential equations and gives answers to the “real world “structural problems it has been substantially extended scope during the roughly forty years of its use.
FEA makes it possible to evaluate a details and complex structure, in a computer during the planning of the structure. The demonstration in the computer about the adequate strength of the structure and possibility of improving design during planning can justify the cost of these analysis works. FEA has also been known to increase the rating of the structures that work significantly over design and build many decades above.
In the absence of finite element analysis, development of structures must be based on hand calculations only. for complex structures, the simplifying assumptions are required to make any calculations possible can lead to a conservative and heavy design. Considerable factors of ignorance can remain as to whatever the structure will be adequate for all design loads. Significant changes in design involve expensive strain gauging to evaluate strength and deformation.

TERMS COMMONLY USED IN FEM

Descritization : The process of selecting only a certain number of discrete points on the body can be termed as descritization .

1. Continuum: The continuum is a physical body, structure or solid being analyzed.
2. Node: The finite elements, which are inter connected at joints, are called nodes or nodal points
3. Element: small geometrical regular figure are called elements.
4. Displays models: The simple functions, which are assumed approximate the displacement for each element. These functions are called the displacements models or displacement functions.
5. Local coordinate system: Local coordinate system is one that is define for a particular element and not necessary for the entire body or structure.
6. Global system: The coordinate system for the entire body is called global system.
7. Natural coordinate system: Natural coordinate system is a local system, which permits the specification of points

within the element by a set of dimension less numbers, whose magnitude never exceeds unity.

8. Interpolation functions: It is a function, which has unit value at one nodal point and a zero value at all other nodal points
9. Accept ratio: The accept ratio describes the shapes of the element in the assemblage for two dimensional elements these parameters is defined as the ratio of the largest dimension of the element to the smallest dimension
10. Field variable: The principle unknowns of problems are called the variables.

GENERAL DESCRIPTION OF FEM

In the finite element method, the actual continue of body matters like solid, liquids or gas is represented as an assemblage of sub divisions called finite elements. These elements are considered to be interconnected at specified points known as nodes or nodal points.

These nodes usually lie on the element boundaries where an adjacent element is considered to be connected since the actual variations of the field variables(like displacement, stress, temperature, pressure and velocity) inside the continuum are is not known, we assume that the variation of the field variable inside a finite element can be approximated by a simple function. These approximating functions (also called interpolation models) are defined in terms of the values at the nodes when the field equations (like equilibrium equations) for the hole continuum are written, the new unknown will be the nodal values of the field variable. By solving the field equations, which are generally in the form the matrix equations the nodal values of the field variables will be known. Once these are known, the approximating function defines the field variable throughout the assemblage of elements.

The solution of a general continuum by the finite element method always follows as an orderly step by step procedure for static structural problems can be stated as follow

Step 1: Descritization of structure (domain)

The first step in the finite element method is to divide the structural solution region into subdivisions or elements.

Step 2: **Selection of proper interpolation model**

Since the displacement (field variable) solution of a complex structure under any specified load conditions cannot be predicted exactly. We assume some suitable solution with in an element to approximate the solution. The assumed solution must be simple and it should satisfy certain convergence requirement.

Step 3: Derivation of element stiffness matrices and load vectors.

From the displacement model the stiffness matrix [K(e)] and the load vector P(e) of element's are to be derived by using the either equilibrium conditions are a suitable variational principle.

Step 4: Assemblage of element to obtain the equilibrium equations:

Step the structure is composed of several finite elements the individual element stiffness matrices and load vectors are to be assembled in a suitable manner the overall equilibrium equation as to be formulated

$$[K][\Phi] = [P]$$

Where K is called assembled stiffness matrix.

Φ is called the vector of nodal displacement and

P is the vector or nodal force complete structure.

Step 5: solution of system equation to find nodal values of displacement (field variable)

The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. after the incorporation of the boundary conditions, the equilibrium equations can be expressed as

$$[K][\Phi] = [P]$$

For linear problems, the vector 'Φ' can be solved very easily. But for non linear problems, the solution has to be obtained in a sequence of steps, each step involving the modification of the stiffness [K] and Φ or the load vector P.

Step 6: computation of element strains and stresses.

From the known nodal displacements if required the element strain and stresses can be computed by using the necessary equations of solid or structural mechanics (implemented the general FEM step by step procedure.

BASIC APPROACH TO FEA

Basic approach for any finite element analysis can be divided into three parts:

- Pre-processor
- Solver
- Post-processor

Pre-processor:

Pre-processor mainly contains building of model, meshing, assigning material properties etc**.**

Building of models:

Geometry is usually difficult to describe as it has to be as close as real. Since it has to take real world loads and boundary condition. It is equal to prototype simulated in a computer.

Creation of finite element model of Meshing:

After assigning material properties and structural properties to be model, meshing is done. Meshing means dividing a solid model in to a finite number of finite sized elements.

FEA uses a complex system of points called nodes, which make grid called MESH. This mesh is programmed to contain material and structural properties, which defined how the structure will react to certain loading conditions. Nodes are assigned at a certain density throughout the material depending on the anticipated stress levels of particular area. Reasons that will receive large amounts of stress usually have a higher node density than those when experience little or no stress.

Point of interest may consist of fracture point are previously tested material, fillets, corners, complex detail and high stress areas. The mesh acts like a spider web in that from each node. There extents a mesh element to each of the adjustment nodes. This web of vectors is what carries the materials properties to the object, creating many elements. Each finite element analysis programs may come with an element library or one is constructed over time. Elements are thought to fill the whole experiment hence choice of element is very critical.

While meshing it has to be decided whatever we have to go for finer mesh or coarser one. The finer the mesh, the better will be the result but longer analysis time. Therefore, a compromise between accuracy and solution speed is usually made. The mesh may be created manually or automatically. In the manually created mesh, we will notice that the elements are smaller at joist. This is known as mesh refinement. And it enables the stress to be captured at the geometric discontinuities.

Manual meshing is a tedious process for models with any degree of complications, but inbuilt mesh engine creates mesh automatically thus reducing the time. These automatic meshing has limitations us to regards mesh quality and solution accuracy.

Solver:

Solvers are geometric task oriented. These are developed for specific applications. Solvers are designed based on continuum approach where In construction of mass, momentum

ISSN Online: 2319 – 9253
Print: 2319 – 9245

and energy equations of stats, thermodynamic equation as and when required for each of the elements and solution is obtained by interpreting these solution. The solution of these equations essentially depends on two methods:

1. Implicit
2. Explicit

Choice of the method is based on the nature of the problems.

The main goal of a finite element analysis is to examine a structure or a component response to certain loading conditions. Therefore specifying the proper loading conditions is a key step in the analysis. These loading conditions may be static, dynamic or transient whose nature may be linear or non-linear.

The word loads in FEA includes the applied forcing functions externally or internally. For example, loads in structural are displacement, forces, pressures, temperatures and gravity. In thermal analysis the applied loads are temperature, heat flow rates convection and heat generation.

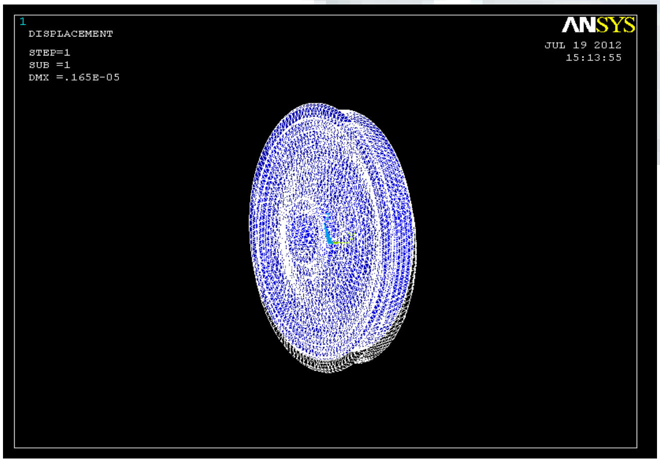
Solver reads the model created by the preprocessor and formulates the mathematical representation of the model all the parameters defined in the preprocessing stage are used to do this. If the model is connect the solver proceeds to form the element stiffness matrix for the problem. After calculating the element stiffness matrix is calculates the results like displacements, pressure, stresses, strains etc. at node within the component or continuum. All these results are sent to a result file which may be read by the postprocessor.

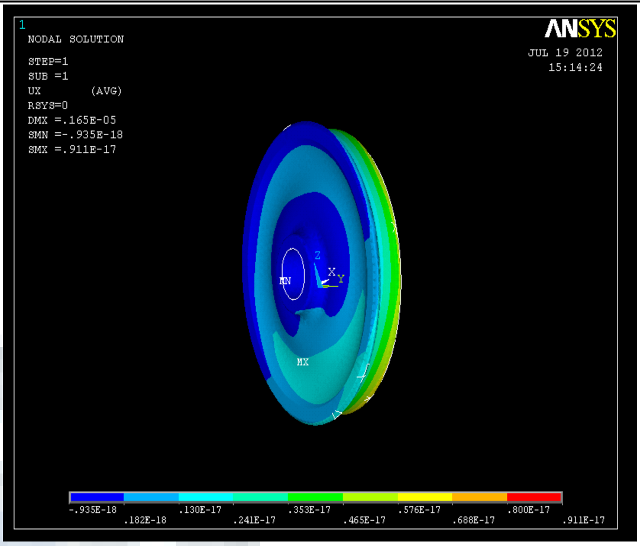
IV. EXPERMENTAL RESULTS

RESULTS for AISI 8640:

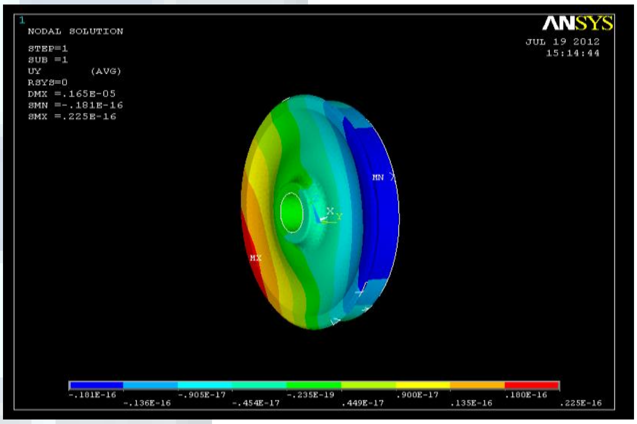
MODE 1:

Results for set 1

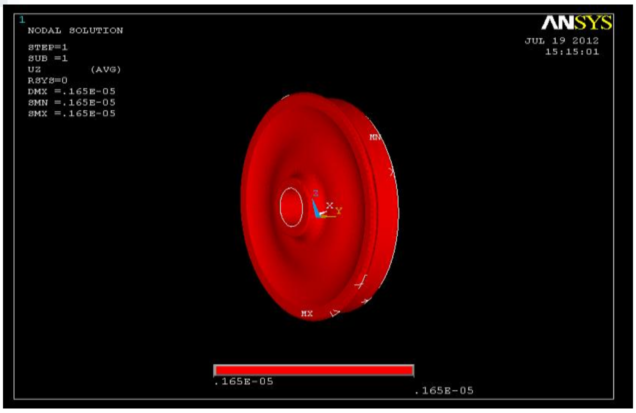




SET 1 DOF IN X



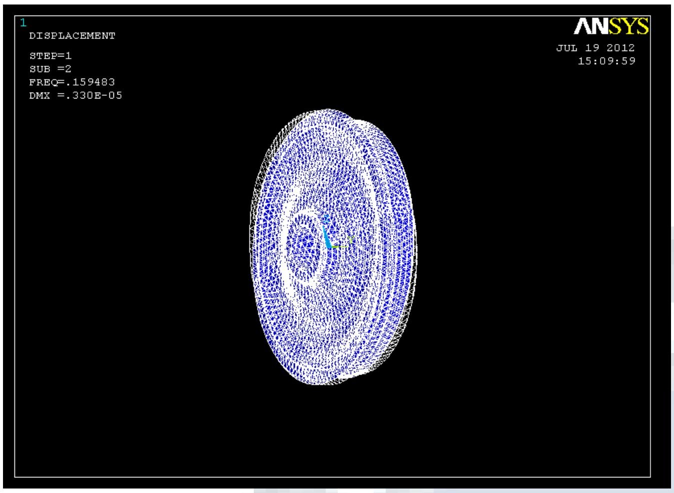
SET 1 DOF IN Y

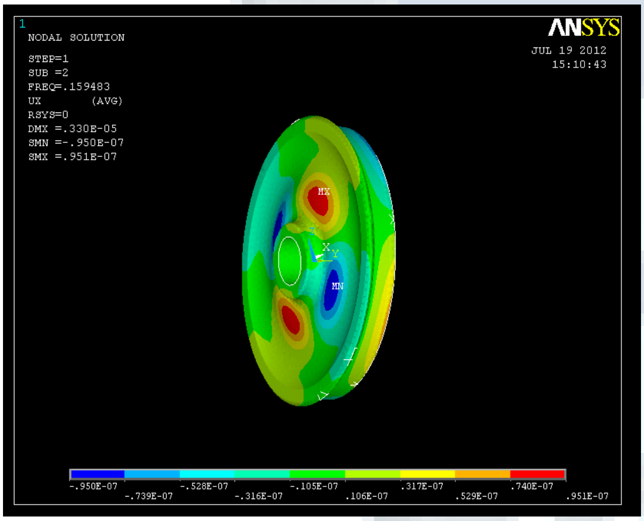


SET 1 DOF IN Z

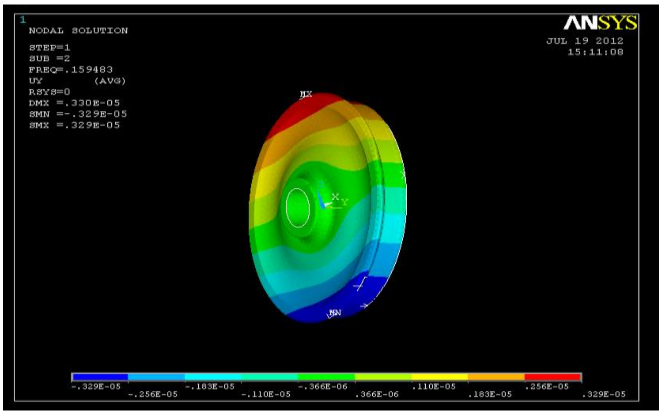
IPHV7I3009X
International Journal Of Advanced Research and Innovation -Vol.7, Issue .III
ISSN Online: 2319 – 9253
Print: 2319 – 9245

Results for set 2

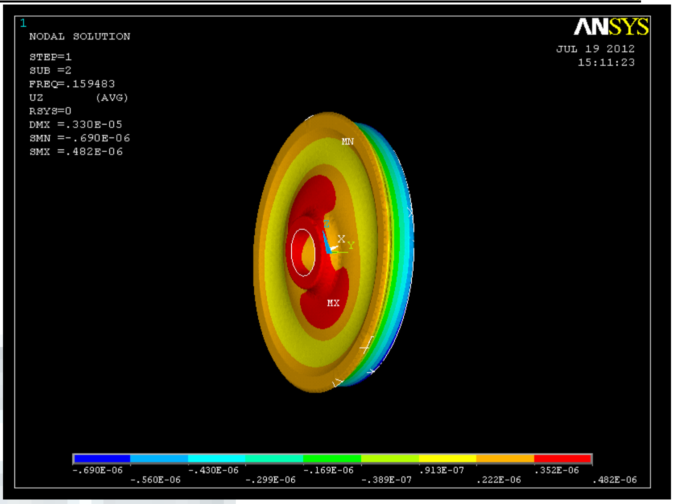




SET 2 DOF IN X



SET 2 DOF IN Y



SET 2 DOF IN Z

V.CONCLUSIONS

I had done the model analysis for three types of materials for the model, The frequencies obtained for the material AISI 8640 are less when compared to other two materials. Therefore AISI 8640 can be preferred as a material for the railway wheel. Due to its high cost and other factors, it is not being used by Indian railways. So plain carbon steel is being used by Indian railways. There is a need for replacing traditional plain carbon steel with alloy steel, which Will reduce wear rate up to some extent. Study for better and economic alloy steels is needed to improve wheel life. Also study should be conducted in wheel profile for increasing the wheel life. Research should be concentrated on wheel tread since it is the part which will be Always in contact with rail.

REFERANCES:

[1]. Machine Design an integrated approach by Robert L Norton
[2]. Machine Design by Shigley
[3]. Machine Design by R.S.Khurmi
   a. Page no: 830
[4]. Automobile Engineering by Kirpal Singh
[5]. Design Data Book
[6]. www.machinedesign.com
[7]. www.wikipedia.com
[8]. www.irfca.com.
[9]. www.rdso.com
[10]. www.irs.com.