International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

IJIEMR Transactions, online available on 15th FEB 2019. Link :
http://www.ijiemr.org/main/index.php?vol=Volume-08&issue=ISSUE-02

Title: **SIMULATION STUDIES ON LOCO WHEEL USING ANSYS**

Volume 08, Issue 02, Pages: 32–38.

Paper Authors

GUDIVADA ARUNDHATHI LAXMANA REKHA,P. CHANDRA SEKHAR

AIMS College of engineering, Mummidivaram

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

SIMULATION STUDIES ON LOCO WHEEL USING ANSYS

GUDIVADA ARUNDHATHI LAXMANA REKHA[1] ,P. CHANDRA SEKHAR[2]
[1] PG scholar, Dept. of M.E., AIMS College of engineering, Mummidivaram
[2] Professor& Head, Dept. of M.E., AIMS College of engineering, Mummidivaram

ABSTRACT:
The wheels are inherently subjected to damage caused by the rolling contact between the wheel and the rail. The locomotive wheels are subjected to high rail contact loads. There is a static load due to the rail pressure and there is a large amount of thermal contact load. The wheel and rail contacts also encompass high thermal stresses. In this work, an attempt has been made to investigate the variation of temperature against cycle time, the maximum residual stress by varying the profile, parametric variations of dimensions of the wheel and contact pressure, the materials of the wheel under transient heat transfer analysis. A tool for aforementioned attempts has been developed for the FE analysis carried out. The collection of methods and tools, the use of which is governed by a process superimposed on the whole is undertaken. Generally, a method, which is an organized, single purpose discipline or practice, evolves as a distillation of the best-practices experienced in a particular domain of cognitive or physical activity. This is referred to as the methodology. A tool refers to a software system, such as a Finite Element System, designed to support the method. For aforementioned tasks and a tool has been chosen appropriately a method and a tool has been developed. As the wheel in continuous contact with the rail for a long time, there is a chance of failure occurring in the structure due to the wear and tear. So, structural analysis is performed, where the mechanical loadings are not axisymmetric, unlike in the case of the thermal analysis, and therefore half of the wheel needs to be considered.The wheel for a particular profile is analyzed for the varying parameters and the associated results that are obtained are included in this work. Further, the materials are also varied and the comparison between the various parameters for numerous material considerations are assessed and also presented in the form of graphs etc., This definitely serves as a ready reckoner for those who wish to continue a extremely detail oriented analysis on the suitability of materials to fabricate the loco wheel The results obtained are reviewed and the work is provided with some conclusion and the future scope is also included.

1.0 INTRODUCTION:

The rail wheel assembly is an integral part of many systems that are intended to move in a guided path, for example, Trains, Cross-Trolleys for Overhead Cranes etc. The entire traveling mechanism depends upon the rail wheel assembly, without which the movement would have been very difficult.

The design and analysis of the rail wheel assembly becomes a critical part, as its failure can result in a great damaged and loss. I intend to consider one such design for the wheel and conduct a transient analysis on the loco wheel. I also intend to suggest some design modifications in the existing

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

design and studyThe effects of this change on the original design. I am considering a standard design for a
Loco wheel.The loco wheel has been used in practice for over years now. We see a lot of different types of wheels used for specific purposes. There are different standard sizes and types of wheels available according to the type of their use and the type of load applied to it. The very important criterion for considering the design is maximize the strength and minimize the weight which can be accomplished by minimizing the material used, as it is directly proportional to the cost. Many modifications have been done to the loco wheel, before the latest one is attained. After an iterative approach and constant change in the design,The wheels are manufactured from steel made by electric, basic oxygen or any other process recognized as equivalent buys the purchaser. The steel shall be of killed quality.

1.1CHEMICAL COMPOSITION

LADLE ANALYSIS

Ladle analysis of steel when carried out by the method of specified in the relevant parts of IS: 228 or any other established instrumental / chemical method.

ELEMENT(symbol)	PERCENTAGE(%)
Carbon (C)	0.57-0.67
Manganese (Mn)	0.60-0.85
Silicon (Si)	0.15
Phosphorus (P)	0.03
Sulphur (S)	0.03
Chromium (Cr)	0.25
Nickel (Ni)	0.25
Molybdenum (Mo)	0.06

1.2 MANUFACTURING PROCESS

The wheels are manufactured by forging and rolling process from cropped ingots or from blooms produced by continuous casting process in such a manner that the central axis of the ingot or the bloom coincides with the axis of the wheel. Each ingot or bloom shall be of suitable section and length to produce two or more wheel blanks after top and bottom discards have been made to eliminate completely the defective portion.In case of continuous casting process is employed, the steel shall be refined in the ladle furnace and vacuum de-gassed suitable shrouding arrangements from the ladle to tundish and from tundish to mould shall be made.

1.3 DESIGN OPTIMIZATION

Design optimization is a technique that seeks to determine an optimum design. By "optimum design," we mean one that meets all specified requirements but with a minimum expense of certain factors such as weight, surface area, volume, stress, cost, etc. In other words, the optimum design is usually one that is as effective as possible.

Virtually any aspect of your design can be optimized: dimensions (such as thickness), shape (such as fillet radii), placement of supports, cost of fabrication, natural frequency, material property, and so on. Actually, any ANSYS item that can be expressed in terms of parameters can be subjected to design optimization. The ANSYS program offers two optimization methods to accommodate a wide range of optimization problems. The *sub problem approximation* method is an advanced zero-order method that can be efficiently applied to most engineering problems. The *first*

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

order method is based on design sensitivities and is more suitable for problems that require high accuracy.For both the sub problem approximation and first order methods, the program performs a series of analysis-evaluation-modification cycles. That is, an analysis of the initial design is performed, the results are evaluated against specified design criteria, and the design is modified as necessary. This process is repeated until all specified criteria are met.In addition to the two optimization techniques available, the ANSYS program offers a set of strategic tools that can be used to enhance the efficiency of the design process. For example, a number of random design iterations can be performed.

FINITE ELEMENT METHOD:

Finite Element Modeling (FEM) and Finite Element Analysis (FEA) are two of the most popular mechanical engineering applications offered by the existing CAD/CAM systems. This is attributed to the fact that the Finite Element Method is perhaps the most popular numerical technique for solving engineering problems. The method is generally enough to handle any complex shapes or geometry, any material properties, any boundary conditions, and any loading conditions. The generality of the Finite Element Method fixes the analysis requirements of today's complex engineering systems where closed dash form solutions of governing equilibrium equations are usually not available. In addition to this, it is an efficient design by which designers can perform parametric design studies by considering various design cases, analyzing them, and choosing the optimum design.

Finite Element Method is a numerical analysis technique for obtaining approximate solutions to a wide variety of engineering problems. The method originated in the aerospace industry as a tool to study the stress in complex air framed structures. It grew out of what was called the matrix analysis method, used in aircraft design. The method has gained increased popularity among both researchers and practitioners. This is a solution for obtaining solutions to many of the problems encountered in engineering analysis. In the finite element method, the solution region is considered as built of many small inter connected sub regions called finite elements. As an example consider the milling machine structure, it is very difficult to find the exact response (like stress and displacements) of the machine under any specified running conditions. This structure is approximated as composed of several pieces in the finite element method. In each piece or element, a convenient approximate solution is assumed and the conditions of overall equilibrium of the structure are derived. The satisfaction of these conditions will yield an approximate solution for the displacements and the stresses. The finite element may be divided into two phases. The first phase consists of study of the individual element. The second phase is the study of the assemblage of elements representing the entire body.

1.4 GENERAL DESCRIPTION OF FINITE ELEMENT METHOD

The step-by-step procedure for the static structural problem can be stated as follows.

Step 1: Discretization of structure

Step 2: Selection of a proper interpolation model

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

Step 3: Derivation of element stiffness matrices (characteristic matrices) and load vectors
Step 4: Assemblage of element equation to obtain the overall equilibrium equations
Step 5: Solution of system equations to find nodal values of the displacement

DISADVANTAGES

1. Specific numerical result is obtained for a specific problem
2. Experience and judgment are required in order to construct a good finite element model.
3. A computer with more memory capacity and a reliable computer program (software) are essential.

Input and output data are tedious to prepare and interpret.

2.0 FINITE ELEMENT ANALYSIS

2.1 Evolution of ANSYS program

ANSYS has evolved into multi-purpose design analysis software program, recognized around the world for many of its capabilities. Today, the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable, and faster. In this way, ANSYS Inc. helps engineers meet pressures and demands of the modern product development environment.

2.2 OVERVIEW OF THE PROGRAM

The ANSYS program is flexible, robust design analysis and optimization package. The software operates an major computers and operating systems, from PC's to work-stations to supercomputers. It features file compatibility throughout the family of products and across all platforms. Its design data access enables user to import computer-aided design models into ANSYS, eliminating repeat work. This ensures enterprise-wide, flexible engineering solution for all ANSYS users.

2.3 PROCEDURE FOR ANSYS ANALYSIS

2.3.1 BUILDING THE MODEL

The maximum amount of time, in Finite Element Analysis, is spent on generating elements and nodes and elements automatically and at the same time allowing the user to control over size and number of elements.Using Pre-processor, which is an interactive model builder, generates the Finite Element Model. This allows easy transfer of models from one system to the other. The Finite Element Model can be generated in two different methods.

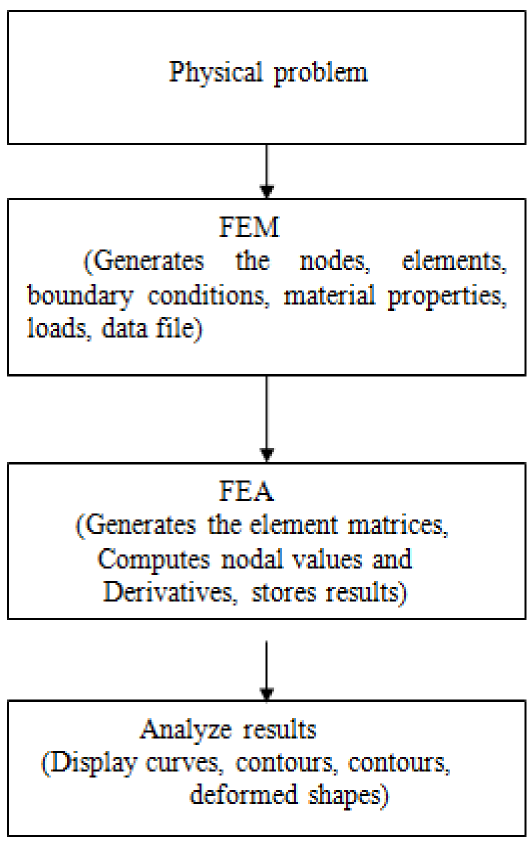

Fig2.1: Practicing FEM and FEA.

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

2.3.2 OBTAINING THE SOLUTION

In this phase of the analysis, the computer solves the simultaneous equations that the finite element method generates. The results of the solution are:

a) Nodal degree – of – freedom values, which form the primary solution, and

b) Derived values, which form the element solution.

The ANSYS program writes the results to the database as well as to the result file. Several methods ate available for solving the simultaneous equations are available in ANSYS:

- Frontal Solution
- Sparse direct solution
- Jacobin Conjugate Gradient (JCC) solution
- Incomplete Cholesky Conjugate Gradient (ICCG) solution
- Preconditioned Conjugate Gradient (PCG) solution and
- Automatic iterative solver Option (ITER).

The frontal solver is the default, but we can select a different solver.

3.0 THERMAL ANALYSIS

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or a component. Typical thermal quantities are:

1. The temperature distributions
2. The amount of heat lost or gained
3. Thermal gradients
4. Thermal fluxes.

The various types of the thermal analysis that can be performed are as follows:

1. Steady state thermal analysis
2. Transient thermal analysis

3.1 ELEMENT CONSIDERED FOR THERMAL ANALYSIS

According to the given specifications, the element type chosen is PLANE 77 (Dimensions: 2D; Shape or Characteristic: Quadrilateral, eight nodes; Degrees of freedom: Temperature at each node; Usage Notes: Useful for modeling curved boundaries).

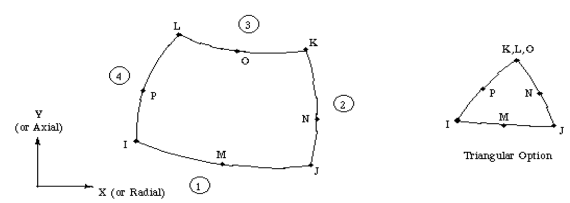

Fig: 3.1 Schematic diagram of 8 node thermal solid element.

3.2 MATERIAL PROPERTIES

Material properties are entered as follows:

Thermal co-efficient of expansion (K_{xx})	= 1.7039e^{-5} /°C
Thermal conductivity (K)	= 0.473445 W/m-°C
Specific heat	= 453.6245 J/kg-°C
Convection film coefficient (h)	= 2.27 E-5 W/mm^2
Applied temperature (t)	= 25°C
Ambient temperature (t_∞)	= 23.8°C
Heat fluxes on nodes (q)	= 746 W/mm^2

THERMAL ANALYSIS

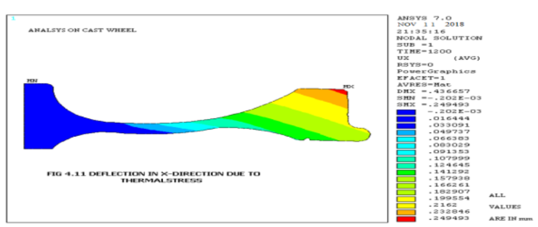

FIG 4.11 DEFLECTION IN X-DIRECTION DUE TO THERMALSTRESS

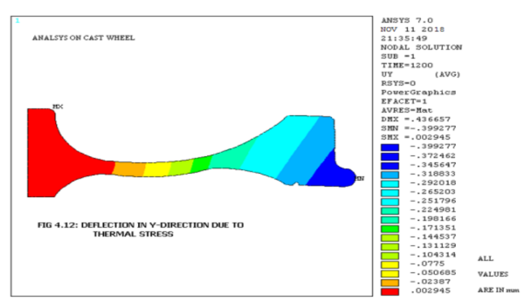

FIG 4.12: DEFLECTION IN Y-DIRECTION DUE TO THERMAL STRESS

International Journal for Innovative Engineering and Management Research
A Peer Revieved Open Access International Journal
www.ijiemr.org

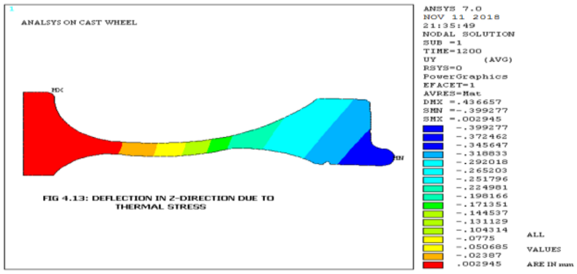

FIG 4.13: DEFLECTION IN Z-DIRECTION DUE TO THERMAL STRESS

4.0 STRUCTURAL ANALYSIS

Structural analysis is the most common application of the finite element method. The term *structural (or structure)* implies not only civil engineering structures such as bridges and buildings, but also naval, aeronautical, and mechanical components such as pistons, machine parts, and tools.

4.1 TYPES OF STRUCTURAL ANALYSIS:

There are seven types of structural analysis available in ANSYS. One can perform the following types of structural analysis. Each of these analysis types are discussed in detail as follows.

1. Static Analysis,
2. Modal Analysis,
3. Harmonic Analysis,
4. Transient Dynamic Analysis,
5. Spectrum analysis,
6. Buckling Analysis, and
7. Explicit Dynamic Analysis.

4.2 STUCTURAL STATIC ANALYSIS:

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include *steady inertia loads* (such as gravity and rotational velocity), and time-varying loads that can be approximated as *static equivalent loads* (such as the static equivalent wind and seismic loads commonly defined in many building codes). A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads such as gravity and rotational velocity, and time-varying loads that can be approximated as static equivalent loads such as the static equivalent wind and seismic loads commonly defined I many building codes.

RESULTS OF STRUCTURAL ANALYSIS

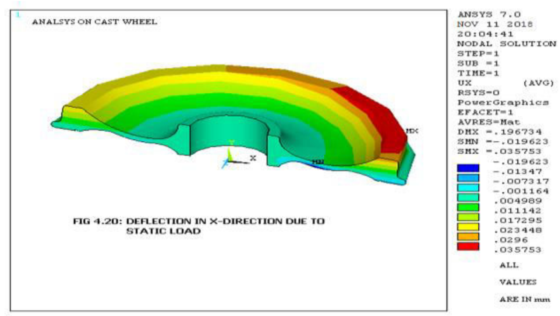

FIG 4.20: DEFLECTION IN X-DIRECTION DUE TO STATIC LOAD

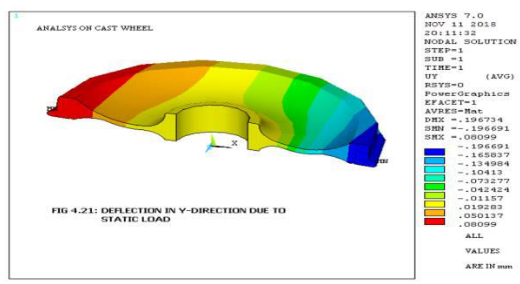

FIG 4.21: DEFLECTION IN Y-DIRECTION DUE TO STATIC LOAD

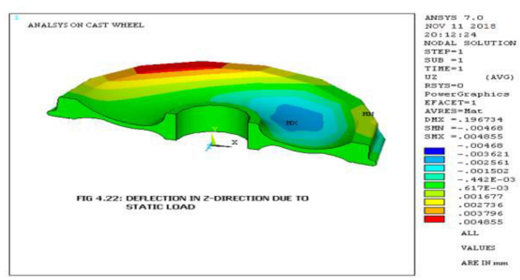

FIG 4.22: DEFLECTION IN Z-DIRECTION DUE TO STATIC LOAD

CONCLUSION

A methodology has been presented to predict the temperature distribution at the end of braking time for a loco wheel of the standard size. The geometry of the wheel can easily be changed. The model can be generated from measured wheel profiles. The method and design provided can be applied to provide critical design information for engineers responsible for the durability and reliability of the loco wheels.The thermal analysis of Axi-symmetric model has been performed due to symmetric boundary conditions and loading conditions in thermal analysis. The results of loco wheel have been obtained with general purpose Finite Element Package ANSYS 5.4 for the specified thermal loads and boundary conditions. In accordance with the thermal analysis, the results obtained are within the permissible limits. The convergence studies have confirmed accuracy and convergence of the results. Hence the design is safe, based on rigidity and strength.The regime for a rail and wheel contact is frequently characterized as either stress related or wear related. This work can be further extended for the various other profiles and sizes. Also, for various other materials, it can be checked for the validity of the rigidity and distribution of the stresses.

REFERENCES

[1] Chandrapatla, R.Tirupathi, Ashok D Belugundu, 'Introductio to Finite Elements in
Engineering', Prentice Hall of India, 3rd Edition.

[2] Reddy, P.R, 'Procedure for Analytic evaluation of locomotive and freight car design,
Association of Rail Roads Standards s-660-83, ASE, pp.83-87, 1999.

[3] ANSYS User Manuals, Swan Sea, USA, 1970.

[4] ANSYS Reference Manuals, Swan Sea, USA, 1970.

[5] Ravinder Reddy P, Computer Aided Design and Analysis, Proceedings of Winter School sponsored by ISTE, 12th Aug., 1999.

[6] Rao S.S., 'Introduction to FEM', Wiley Estern Int., 2nd edition, 1985.