International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 06 | June-2018 www.irjet.net p-ISSN: 2395-0072

FINITE ELEMENT ANALYSIS OF BICYCLE WHEEL

M.KUMAR[1], K.RAJAN[2]

[1,2]Lecturer Senior Grade, Dept. of Mechanical Engineering, Murugappa Polytechnic College, Chennai, Tamil Nadu, India

Abstract - *In the traditional realm of Finite Elements, there are very few practical applications where the finite code can be compared to direct measurements. At best, Engineers can usually only compare their computer analysis with the results predicted by established mechanical formula. At worst, Engineers must rely on their experience and intuition to guide them towards a workable" right" answers.*

This is a familiar structure whose geometrical design has remained virtually unchanged for the past several decades. However, although the design is familiar, and the design effective, it is still not very well understood from a" textbook analysis" standpoint. The optimal design was converged upon by FEM analysis

In this project, I analyses the stress, strain & shear force in the wheel structure. The purpose of this analysis is to formulate a finite element model of the bicycle wheel using analyzing software ANSYS.

Key Words: Bicycle Wheel, FEA, Stress and Strain

Chapter- I

1.0 Introduction to Bicycle Wheel

This is a familiar structure whose geometrical design has remained virtually unchanged for the past several decades. However, although the design is familiar, and the design effective, it is still not very well understood from a" textbook analysis" standpoint. The optimal design was converged upon by FEM analysis

Spoke bicycle wheels are efficient, highly evolved, structural systems. A useful analogy for a bi-cycle wheel supporting vertical loads is that of a circular beam on a pre-stress elastic foundation, fixed at the centre and loaded radial at the circumference. To apply this analogy, the system of interlacing spokes can be modelled as a disk of uniform stiffness per length of circumference. Spokes of varying lengths may be laced into wheels of fixed dimensions, by modifying the interlacing geometry of the spokes. The connection of the spoke to the hub is accomplished via a cold-worked right angle elbow in the spoke and a flanging of the spoke material. Most spoke failures occur at this fatigue critical de-tail. Upon the failure of one spoke, ensuing unbalanced lateral forces on the rim result in large lateral deformations of the rim, which may precipitate lateral buckling of the wheel, or failure of other spokes. A loose spoke can interfere with the smooth operation of the chain, which may result in a lost race, a collision, or an injury.

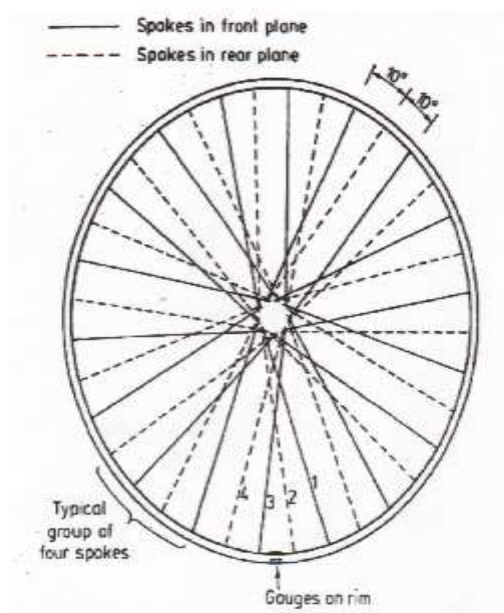

Figure 1: Bicycle Wheel Geometry

Variations in radial stiffness due to the spoke pattern, calculated using the theory of circular beams on elastic foundations, agree with those using a three-dimensional elastic frame analysis. While the spoke lacing pattern influences the over-all stiffness of the wheel, the strains in spokes at the loading point on the rim are not as sensitive to the spoke pattern. Static tests on three identical rear wheels, with different spoke lengths, also show that spoke strains are not strongly affected by the spoke pattern. Strain time-histories collected during road tests also show only small differences between the wheel types. The fatigue life of the spokes of a rear wheel supporting radial loads is therefore not significantly influenced by the spoke pattern. Spoke strain time histories, collected under actual road conditions, are used to evaluate the fatigue reliability of the three wheels. This

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 06 | June-2018 www.irjet.net p-ISSN: 2395-0072

study shows that spokes bicycle wheels have high reliability against fatigue failure.

CHAPTER- II

2.0 FATIGUE IN BICYCLE SPOKES

A bicycle spoke can be exposed to many elements and could fail due to a variety of factors. A fault tree analysis was completed to outline some of these factors. Figure 1shows the fault tree analysis that was completed for bicycle spoke failure. In the figure, the fatigue section is highlighted because this failure mode was the focus of the project. Under the fatigue section the focus was specifically on cyclic stress only, because of time constraints and complexity of the project. Fatigue failures before and after damage will be explored in this report.

Fatigue is a process of gradual fracture due to cyclic loading. Cracks form near the surface of the spoke, in an internal defect, at a stress concentration, at deep scratches, orate dislocations. The crack usually begins in a region of concentrated plastic flow as an intense slip band. The most common form of crack propagation is blunting, which occurs on each loading cycle. Usually the fine cracks start at the surface and propagate through the part or material until a failure occurs.

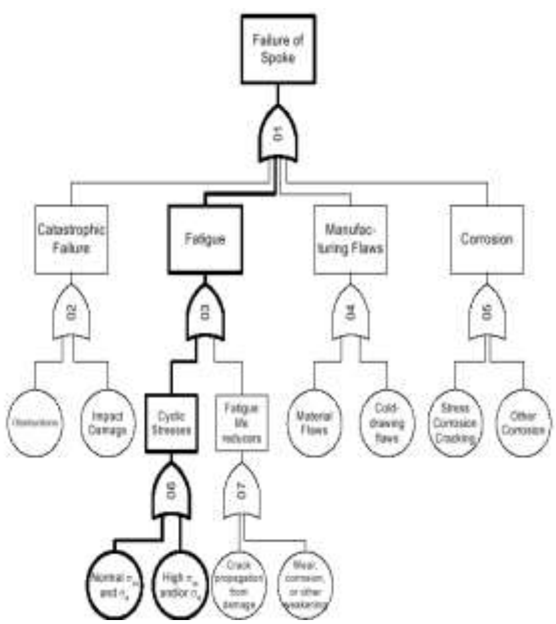

Fig 9: Fault Tree Analysis of bicycle spoke failures

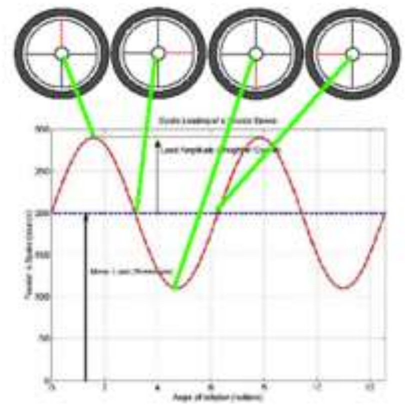

Figure 3: Variation in spoke tension throughout wheel rotation

CHAPTER – III

3.0 ANALYSIS OF A SPOKE WHEEL

But first, some semantics. It is traditional when debating the question of hanging or standing to define your terms to mean something other than what your opponent means. For example, define 'hang' as meaning 'have some tension force' and you can safely say that the hub hangs, because all the spokes have tension. The trouble with this definition is that it results in the conclusion that the hub hangs from something below it, which is not really compatible with the normal usage of the word.

3.1 Engineering - Structural Mechanics

To make sense of what's coming up, you probably need to know why I've done some things (and not done others). To understand that you need to understand some structural mechanics. This is probably first year degree course material, but the basic concepts are not too hard, and hopefully I can explain them well enough to make sense...

3.2 Linear Superposition

The first issue to address is that of linear superposition. What this means is that I'm not analyzing all the stresses in the wheel. I know that all the spokes start out with a uniform tension, and I don't really care about it. For the purposes of the analysis I simply ignore it, and it goes away!

Therefore, when the analysis shows a force in a spoke, the real force in the spoke is whatever the preload (the initial tension) was, plus the force calculated. If the force was tension, we end up with a more highly stressed spoke. If the calculated load was compression we end up with a less tensile spoke. That is, a reference to a 'compressive' spoke could be read as a 'less tensile' spoke. To get the true state in the wheel you need to superimpose (if, add) the results of this analysis on the initial state.

None of this affects the analysis. If we ignore buckling (or restrain against it), a spoke under tension which is subject to (say) 100N less tension contracts just as much as an unstressed spoke subject to 100N compression (or for that matter a compressive spoke subject to 100N more compression). In the analysis I simply don't tell the computer to let anything buckle, so I can analyses a spoke with compression without worrying that it's really a spoke that started out with a tension and now has less tension,

3.3 Statically Indeterminacy

A wheel like this is statically indeterminate. For a statically indeterminate structure, the relative stiffness of the parts of the structure influences the loads in the parts. If you change the stiffness of one part, it changes the load in that part, and since the load has to come from (or go to) somewhere, consequently alters the loads in other parts of the structure. This means simple analogies like "imagine replacing all the spokes with elastic bands..." tend to fall down and reach the wrong conclusion, unless you also imagine replacing the rim with cooked spaghetti, and then you conclude that the wheel won't work at all!

3.5 Finite Element Analysis

When you get a statically indeterminate analysis to do, normally you turn to a computer. You can categories analysis problems by the concept of degrees of freedom. These are (roughly) the number of different variables you need to solve. The flagpoles are a three degree of freedom problem, and I can solve any such problem with a pencil and paper (it becomes a set of simultaneous equations, or the inversion of a matrix). For much bigger problems, it's best to let the computer invert a matrix than do it by hand.

Finite element (FE) analysis is where you split a large complex problem up into small simple bits (the finite elements) that interact in simple ways. For example, it's difficult to write an equation that completely describes the behavior of a wheel, but it's easy to do one that describes the behavior of a spoke - a spoke has no bending and no shear, just axial load. Pull it and it stretches. Pull it twice as hard and it stretches twice as much.

Having split the problem into lots of finite elements you describe how the elements join together, then let the computer work out how they interact and come up with a solution. Easy in principal, sometimes a little more tricky in practice, because you can never model exactly what happens in the real world (the real world is too complicated), and you have to determine whether the simplifications you've made change the answers or not.

CHAPTER – IV

INTRODUCTION TO FEM

4.1 Finite Element Method Concept

Finite element structural analysis is a method of predicting the behavior of a real structure under specified load and displacement conditions. The finite element modeling is generalization of the displacement or matrix method of structural analysis to two and three-dimensional problems and three-dimensional problems. The basic concept of FEM that structure to be analyzed is considered to be an assemblage of discrete pieces called "elements" that are connected together at a finite number of points or nodes. The finite element is a geometrically simplified representation of a small part of the physical structure.

Discrediting the structure requires experience and complete understanding of the behavior of the structure can behave like a beam, truss, plate, and shell.

The finite element method is defined as discrimination whole region (model) in to small finite number of elements. These small elements connected to each other at node points. Finite element analysis grew out of matrix methods for the analysis of structure when the widespread availability of the digital computer made it possible to solve systems of hundreds of simultaneous equations using FEA software like Nastran, Ansys etc

4.2 Finite Element Analysis- General Procedure

Step 1

The continuum is a physical body, structure, or solid being analyzed. Discrimination may be simply described as the process by which the given body is subdivided into an equivalent system of finite elements. The finite elements may be triangles, group of triangles or quadrilaterals for a two dimensional continuum. The collection of the elements is called finite element mesh. The elements are connected to each other at points called nodes. The choice of element type, number of elements and density of elements depend on the geometry of the domain, the problem to be analyzed.

Step 2

The selection of the displacement models representing approximately the actual distribution of the displacement. The three interrelated factors, which influence the selection of a displacement models are

(i) The type and degree of displacement model

(ii) Displacement magnitudes and

(iii) The requirements to be satisfied which ensuring correct solution.

Step 3

The derivation of the stiffness matrix which consists of the coefficients of the equilibrium equation derived from the material and geometric properties of an element. The stiffness relates the displacement at nodal points to the applied forces at nodal points.

[K] {q} = {F}

[K]	-	Stiffness matrix
{F}	-	Force vector
{q}	-	Nodal displacement vector

Step 4

Assembly of the algebraic equation for the overall discredited continuum includes the assembly of the overall stiffness matrix for the entire body from individual element stiffness matrices, and the overall global load vector from the elemental load vectors.

The most commonly used technique was direct stiffness method.

The overall equilibrium relations between the total stiffness matrix [K], the total force vector {R}, and the node displacement vector for the entire body {r} can be expressed as

[K]{r} = {R}(2.2)

Step 5

The algebraic equations assembled in step 4 are solved for unknown displacements. In linear equilibrium problems, this is a relatively straightforward application of matrix algebra techniques.

Step 6

In this step, the element strains and stress are computed from the nodal displacements.

4.3 Degree Of Freedom

The deformations of the structure are represented by the displacements in the nodes.

These displacements are referred to as degrees of freedom.

This can be either transnational or rotational. Each element type will have a predetermined set of degrees or freedom assigned to it.

4.4 General Structure of a FEA- Procedure

The analysis of a structure during its design process is accomplished by the solution of the partial differential equations, which describe the given model. This involves the following three steps.

4.4.1 Pre-Processing

- Type of analysis
- Element type
- Material properties
- Real constants
- Build the model
- Element size
- Meshing
- Loads and boundary conditions (constraints).

4.4.2 Solution

To obtain solution

(Displacement, stress etc)

4.4.3 Post-Processing

Review the results

Graphical display of Displacements and stresses. For quick and easy Interpretation, sorting and Printing of results.

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 06 | June-2018 www.irjet.net p-ISSN: 2395-0072

CHAPTER – V

MODEL GEOMETRY & INPUT DATA

Specification:

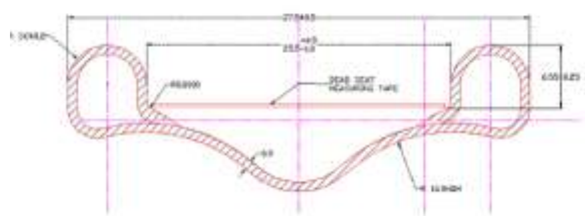

Figure 5: Typical Profile of Bicycle Rim

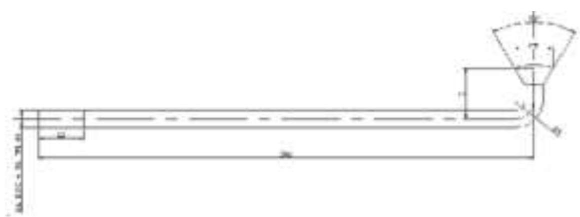

Figure 6: Typical Profile of a Spoke

S.NO	NO OF RIM NODES	NO OF SPOKES	DEGREE BETEWEEN RIM NODES
1	40	40	9

Table 1: Model Dimension

No of spokes	40
Material	Steel wire
Diameter	2.64mm
Length	306mm

Table 2: Spokes Dimension

Width	38mm
Outer Diameter	635mm
Thickness	0.9mm
Material	Cold rolled low carbon steel

Table 3: RIM Dimension

Length	50mm
Flanged dia	50 mm
Flange thickness	2mm
Hub dia	38mm
Pitch circle dia	44mm

Table 4: HUB dimension

Pressure	0.9 N/sq.mm
Poisson's Ratio	0.3
Stiffness property (E)	210E3 N/ sq.mm

Table 5: Input Data

CHAPTER- VI

1D- ANALYSIS RESULT

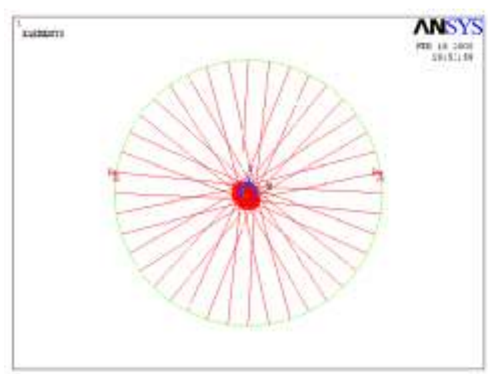

Figure 7: Constrained Model of Wheel

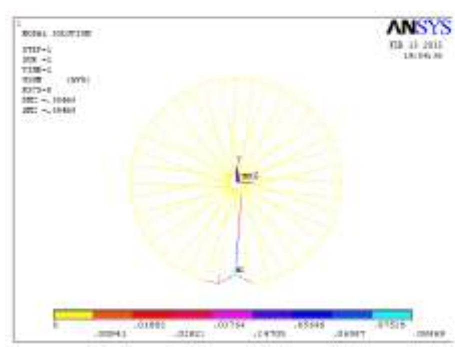

Figure 8: Strained Model

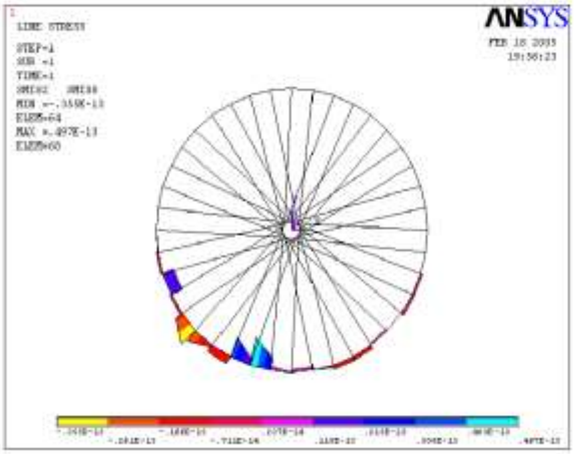

Figure 9: Shear Force

3D- Analysis Results

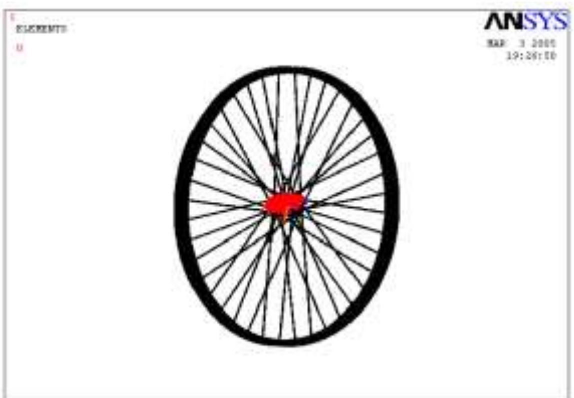

Figure 10: Meshed Model

Maximum Node Numbers	59639
Number of Defined Nodes	59362
Number of Selected Nodes	59362
Maximum DOF per Node	3
Maximum Elements Numbers	217839
Number of Defined Elements	217839
Number of Selected Elements	217839

Table 6: Mesh

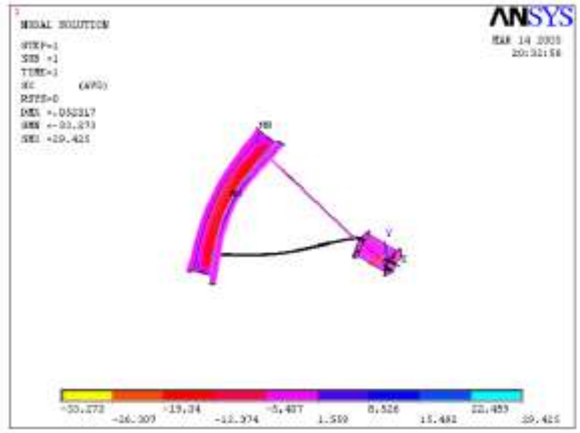

Figure 11: Stress in X direction

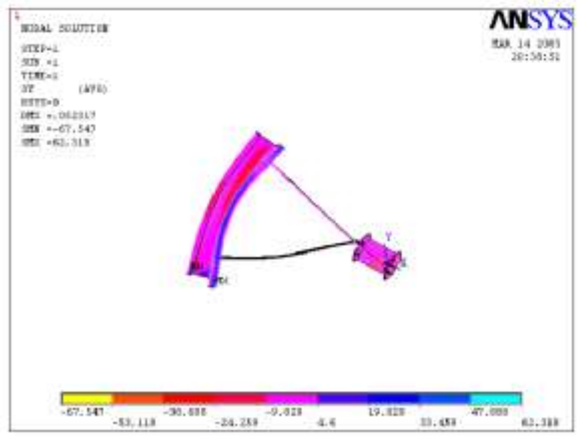

Figure 12: Stress in Y- Direction

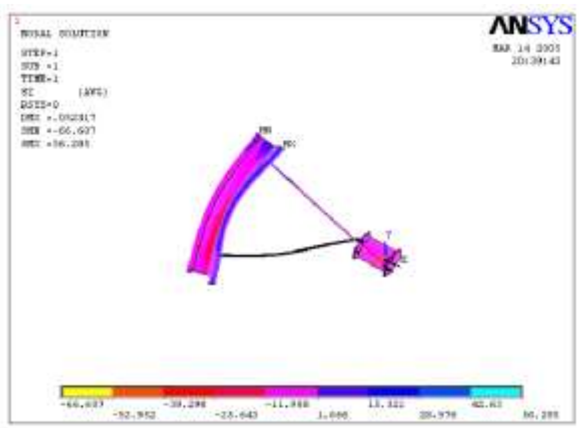

Figure 13: Stress in Z- Direction

Figure 14: Shear Stress in XY Plane

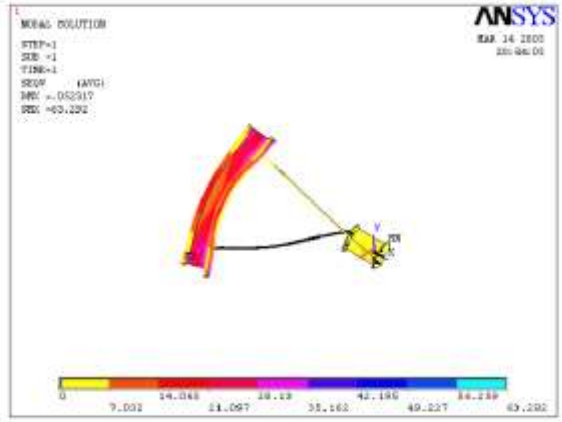

Figure 17: VON- MISSES Stress

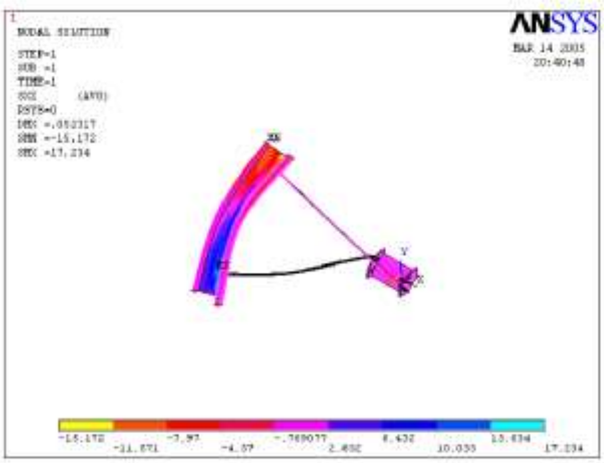

Figure15: Shear Stress in XZ Plane

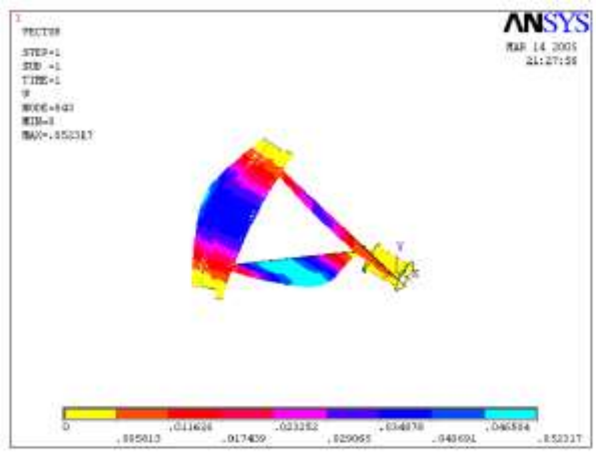

Figure 18: Deformation Vector Plot

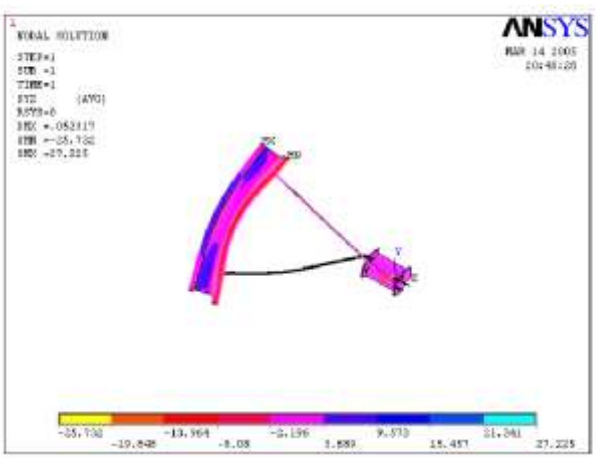

Figure 16: Shear Stress in YZ Plane

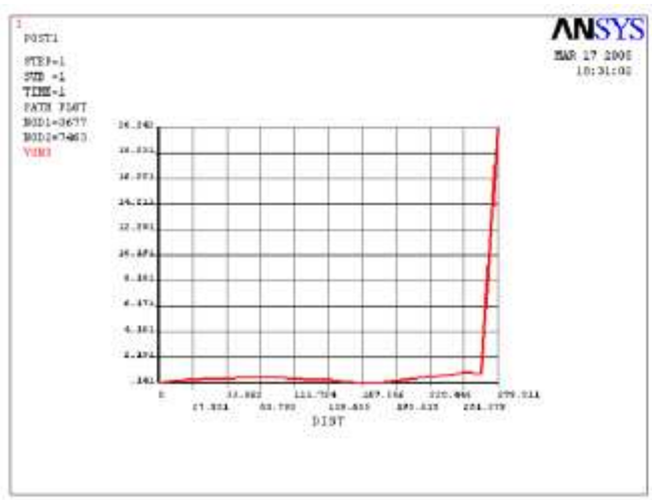

Figure 19: Von Misses Stress

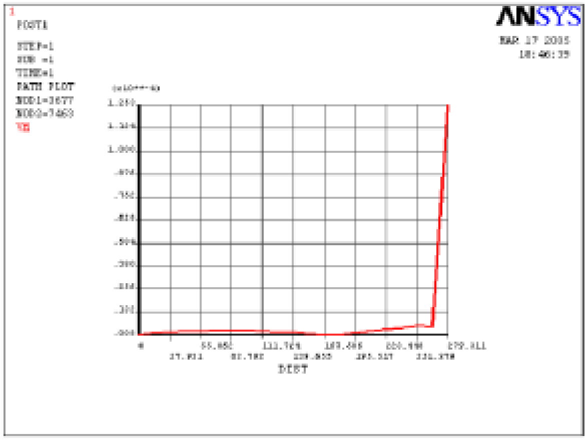

Figure 20: Von- Misses Strain

Sl.No	DESCRIPTION	N/mm²	
		Max	min
1	Stress in X direction	33.273	-29.425
2	Stress in y direction	67.547	-62.318
3	Stress in Z direction	66.607	-56.285
4	Shear stress in XY plane	17.954	-17.209
5	Shear stress in XZ plane	17.234	-15.172
6	Shear stress in YZ plane	27.225	-25.732
7	Von- Misses Stress	63.292	
8	Maximum Deformation	0.052317mm	

Table 7: Analysis Results

CHAPTER – VII

CONCLUSION

It has been concluded that in the earlier stage technocrats used design by formula method for analyzing spoke bicycle wheel, it is a time consuming process and gives less accurate results. In this project I have created the model in Pro/E and analyzed the bicycle wheel model using ANSYS software. Analysis has been done for both one dimensional and three dimensional models more accurately than design by formula .Design by formula cannot predict the exact result due to complex orientation of spokes.

From the analysis result it has been concluded that stress will be more at the point of contact between the spokes and rim than at spokes and hub contact point, this causes failure of spokes near the rim

The formulated ANSYS program is best suited for analyzing different bicycle wheel just by changing the geometrical values in the program. This method of analysis will help to analysis any type of spokes wheel, like automobile two wheeler wheels.

CHAPTER – VIII

REFERENCES

[1] Indian standard (third revision -1997) Bicycles Rim Specification

[2] Bureau of Indian standards, new delhi-110002

[3] Indian standard (third revision-1997) cycle and rickshaw pneumatics tyres-specification

[4] Bureau of Indian Standards, New Delhi-110002 Spokes – Specification

[5] Burgoyne, C.J. and R. Dilmaghanian. "Bicycle Wheel as Prestressed Structure." Journal of Engineering Mechanics.

[6] Division Of Engineering , Brown University ENB: Introduction to engineering and static's November 2004

[7] Eric moody & Scott Henson Brown University "ANALYSIS OF WHEEL", September 2004

[8] Gavin, H.P. (1986) "Spoked Bicycle Wheels: A Comparative Experimental and Finite Element Analysis of Static and Dynamic Characteristics of Bicycle Wheels with Variable Spoking Patterns." Undergraduate Thesis, Dept. of Civil Eng., Princeton University, Princeton, N.J.

[9] Price, D. and Akers, A. (1985) "Stiffness Characteristics of Bicycle Wheels." Bike Tech. Emaus, PA: Rodale Press. Vol. 4, No. 3, June 1985.

[10] Pippard, A.J.S., and Francis, W.E. "The stresses in a radially-spoked wire wheel under loads applied to the rim." Philosophical Magazine,

[11] Fuchs, H.O. and Stephens, R.I. (1980) Metal Fatigue in Structures. New York: John Wiley and Sons.