Design and Analysis of Indian Wheel-Rail Assembly for Super-Elevation

R.Kasanna[1], K.Ajay[2], M.K.Naidu[3], S.Adinarayana[4], I.Sudhakar[5]

[1,2,3,4,5]*Mechanical Department, M.V.G.R College of Engineering (A).*

Abstract: Mechanics of the rail-wheel is one of the fundamental areas of the study in the railway engineering. Complicated geometries like rail-wheel problems are solved by using Finite element analysis software. In present years, loads on axle of railway cars increase because increased in transport of goods and faster infrastructural growth. The rail-wheels are subjected to high contact stresses of alternating magnitude. The rail-wheels are found to fail mainly due to fatigue under these loads. 3-D elastic frictional element model of the rail-wheel is helped to investigate effect of curve radius and super-elevation on contact stress. The present work is focused on the interaction between left and right wheels. The rail model includes 52 kg/m standard rail, sleeper, elastic rail clips, grooved rubber sole plate, SCGI insert, ballast bed. Fatigue analysis is done to calculate the life, damage, safety factor. Modal analysis is done to calculate the Modal shapes, natural frequencies. The results shows that curve radius and super-elevation have significantly effects on contact stress, life, damage, safety factor. And natural frequencies are obtained and the values are within the standard frequency range.
Key words: wheel-rail, superelevation, contact stress, Fatigue, Mode shape,

I. INTRODUCTION

As the train travels on curved track, the vehicle and any passengers inside are influenced by the centrifugal force which is proportional to square of the train velocity and inversely proportional to the radius of the curve. To counteract this effect superelevation is applied to the curve by raising the outside rail when compared with the inside rail. So there is significant change in height of the inside and outside rail the contact positions and stresses on these two positions also different.

Small radius curve and rail joint, turnout are the major problems in railway track structure. Approximately one third of railway track is the curved track in India. Wear, fatigue, and fracture are one of the main damage mechanisms for the rail, which affects the service life of railway track severely. Stress in rail is the key to understand and predict these damage mechanisms of rail. Therefore it is necessary to investigate stresses in rail during passing through curved track.

With the advancement in rail transport and requirement ofrapid transit of goods the failure of railway axles became anardent problem drawing the attention of designers andresearchers. The designs of components of railways should befor infinite life based on the fatigue limit of the material. Sinceaccidents due to fracture of rail-wheel affects the safety ofrailway facility directly.The contact section between the rail and railroadwheel are extremely stressed under the dead and live loads dueto loco motion of the railways.The fatigue problem of railroad wheels is often referred to asrolling contact fatigue, which is caused by recurrent contactstress during the rolling motion.At present, there are three major methods used toinvestigate the wheel/rail contact, namely Hertz method,non-Hertz method and finite element method[2]. First twomethods are limited by the elastic half-space assumption. Inorder to satisfy this assumption and then obtain accurateresult, the dimension of the contact area must be smallercompared with the relative radii of the curvature for eachbody. The elastic deformation of contact bodies is onlyconsidered. For wheel in contact with rail corner, exact stressin wheel and rail cannot be acquired. However, the finiteelement method is not restricted by the elastic half-spaceassumption in wheel/rail contact analysis.

There are othermethods for obtaining wheel-rail contact stress. For example,the distribution of wheel-rail contact stress at contact zone isinvestigated with an ultrasonic technique by Pau et al. [8].A comparison of three-dimensional finite elementmodel and Hertz contact theory was studied by Yan andFischer [7]. Telliskivi and Olofsson [3] compared Hertzmethod and the non-Hertz method with finite elementmethod. They concluded that for the contact zone in railcorner, the results from finite element method do notsignificantly agree with the Hertz method and the non-Hertzmethod.A fatigue–ratcheting damage modelwas developed to predict the lifetime of a rolling element. Asper the study it was observed that the Bainitic steel exhibits longer lives as compared to the Hadfield steel under the samecontact loading.

Vinod Kumar et al.[1] investigated on occurrence of stresses in rail-wheel assembly due to high contact loads at the assembly points. This study involves the determination of equivalent stress (von-Mises), strain, safety factor using finite element method on

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor:6.887
Volume 5 Issue VII, July 2017- Available at www.ijraset.com

ANSYS® 2016. The rail wheel assembly is assumed to operate under the designed load. The wheel and rail are having a rolling contact.Mr. Prabhakar Purushothamanet al. [2] focused on the hertz contact theory validation using finite element Analysis**.** The comparison of manual analytical result, result obtained by Hertz stress calculator software and finite element analysis results shows the finite element analysis results are acceptable.Mehmet Ali Arslan et al. [4] focused on the fundamental way of handling Rail–Wheel contact problems from the FEA standpoint, and highlighted required steps for more realistic 3D solutions to these types of problems. Resulted that most of the plastic deformation occurs at the Rail, on the contrary almost very small plastic deformation occurs at the Wheel.Yongming Liu and Liming Liu et al. [11]focused on subsurface propagation and analysis CAD Model and tools validation for rail-wheel rolling contact fatigue problem.The vertical loading, crack length, wheel diameter, crack depth and crack orientation have relatively significant effects on the subsurface stress intensity factor ranges.At present, there are few papers which study wheel set incontact with double rails at the same time.Variations of the contact stress in the rail with various curve radiuses and superelevationsare investigated to offer information for finding the methodwhich can predict and solve fatigue and fracture of rail.

II.   FINITE ELEMENT ANALYSIS PROCEDURE.

The 3-D CAD model is generated in Creo parametric-2.0 and elastic-plastic material properties are applied on finite elemental model in ANSYS-16.2 workbench to identify the suitability of finite element analysis for the rail-wheel contact problems.

Fig1 Procedure for static structural and fatigue analysis[1]

A.   Geometry of an Assembly

The rail-wheel model includes 52 kg/m standard rail, sleepers, Elastic rail clips, grooved rubber sole plate, liner, SGCI insert, and ballast bed all are designed as per the Indian standards.

Fig 2 Dimensions of wheel, rail & axle [1].

The dimensions ofthe rail-wheel model are as per the standards obtained fromrailway coach factory, Punjab, India [1].
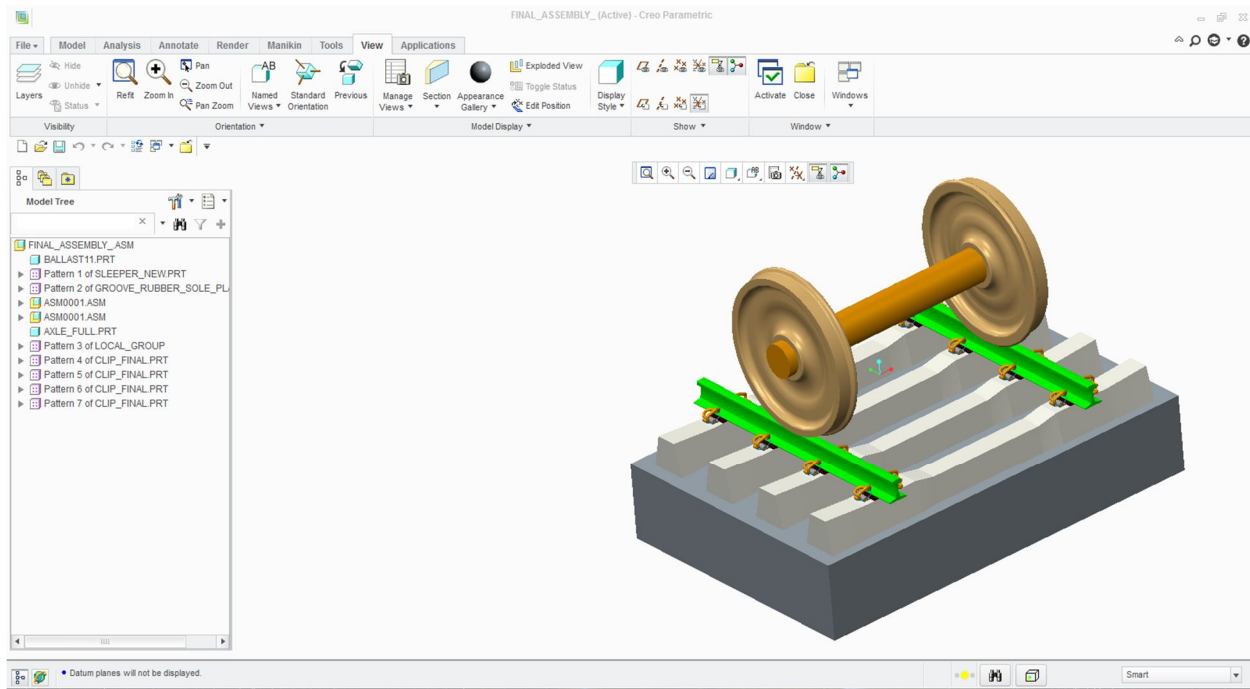

Fig 3 Rail-Wheel assembly without Super-elevation.

With the 26 tonnes axle load at a train speed of 100km/h, theload cases are computed for various curve radiuses (835m,975m, 1150m, 1380m, 1720m) and superelevations(80mm, 100mm, 120mm, 140mm,165mm) in order toinvestigate the influence of curve radius and superelevationon contact stress in rail.

B. Structural Analysis

Structural analysis (nonlinear static analysis) is used todetermine the stress and strain distribution between wheel andrail contact. In this analysis, rail-wheel loading, boundary condition and frictional contact are analyzed.

1) Assigning Properties to the Assembly: Wheel, rail and axel material properties with Bi-linear kinematic hardening elastic–plastic material model are adapted from Arslan and Kayabaşi [4]. The elastic material properties are presented in Table 1.Concrete material properties are assigned to the sleeper,ballast.

Part name	Modulus of elasticity (GPa)	Density (g/cc)	Poisson's ratio(-)
Wheel	210	7.850	0.3
Rail	200	7.850	0.3
Axle	210	7.850	0.3

Table1. Properties of rail-wheel and axle.[4]

2) Meshing of a Rail-Wheel Assembly: The wheel is in the contact with the rail over the sleeper in calculated model. Because it is considered that other sleepers exert an influence on the stress in the contact zone of the wheel/rail, four sleepers are considered in the rail model. The contact between rail and wheel is happened at only two places those are left side and right side. The total assembly is discretized into 5,54,264 nodes are used.To capture high stress and strain gradientsnear the rolling contact surface(rail-wheel contact area), a fine mesh (higher mesh density) is used.
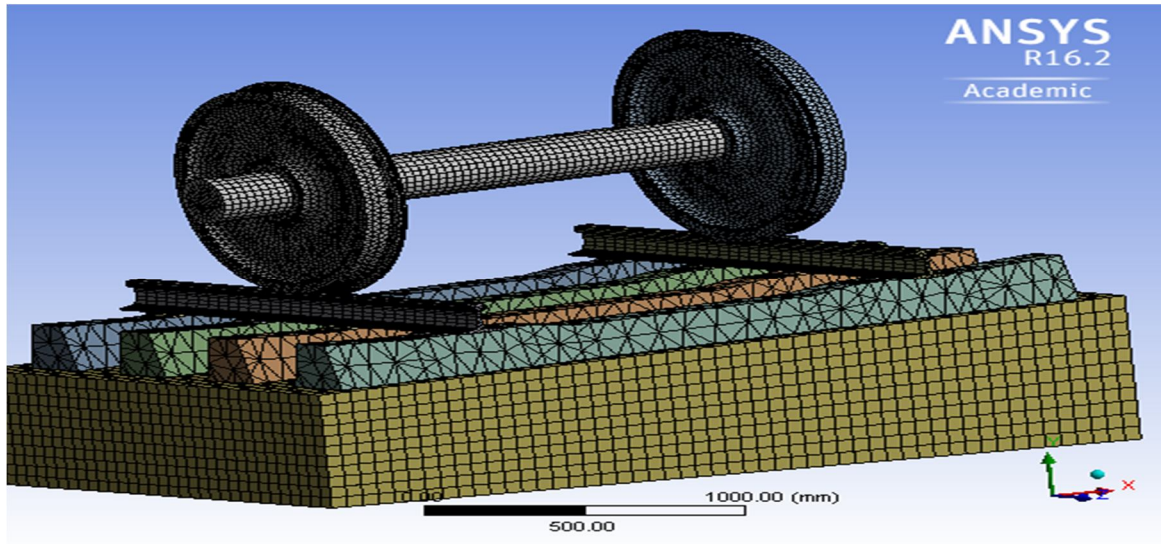
Fig 4.Meshing of a super-elevated Rail-Wheel assembly.

3) *Applying Loads on Rail-Wheel Assembly:* The applied loads acting on the wheel set through the electrical-locomotive are mainly composed of the 26tonnes axle load which applies to wheel set centre at each end of the wheel set, centrifugal force which acts on wheel set centre at the right end (away from the centre of curvature) of the wheel set and the driving torque induced by the traction motor, which can be transformed into a circle of tangential forces uniformly spaced at the node points on the axle surface. Friction coefficient is 0.3 between the wheel and rail. The ballast bed is completely fixed at the bottom. At the same time, the linear displacement of the wheel set centre at each end of the wheel set along rail is fixed in order to avoid rigid body displacement of the wheel set. The load is distributed from elements of axle to elements of rail through nodes of wheel. Then contact is formed between wheel and rail, on which the stress values are calculated.
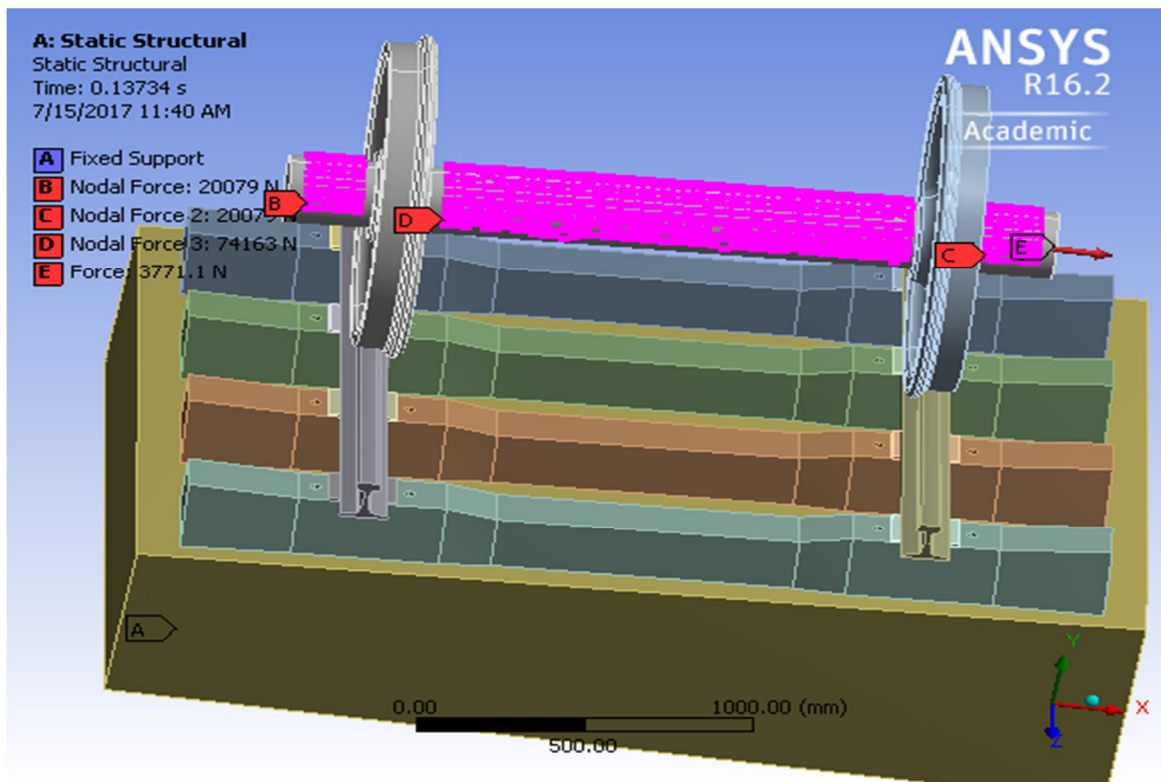
Fig 5. Loads on a super-elevated Rail-Wheel assembly.

4) *Solving the Model and Check for Convergence:* The model is solved and its convergence is checked. If model is converged then the stress histories are calculated and later the fatigue life, damage, safety factor of Wheel-rail is calculated. If the model is not convergence then the step 4 to 5 are repeated until the model convergence is obtained.

C. Fatigue Analysis

In the fatigue analysis, the equivalent von Mises stress and equivalent elastic strain history obtained from the structural analysis is taken as input for the fatigue analysis. Equivalent von Mises stress and equivalent elastic strain are then calculated at each time increment. Based on this technique, the damage and fatigue life, damage, safety factor are calculated using fatigue analysis module on ANSYS Workbench 16.2.The parameters are chosen for fatigue analysis based on the option for Goodman method.

D. Modal Analysis

For modal analysis the material properties, meshing conditions are same as the static structural analysis. Loading conditions are varies. Modal analysis requires only one boundary condition that is fixed support. Ballast bed is fixed and the mode shapes and natural frequencies are calculated.

III. RESULT AND DISCUSSION
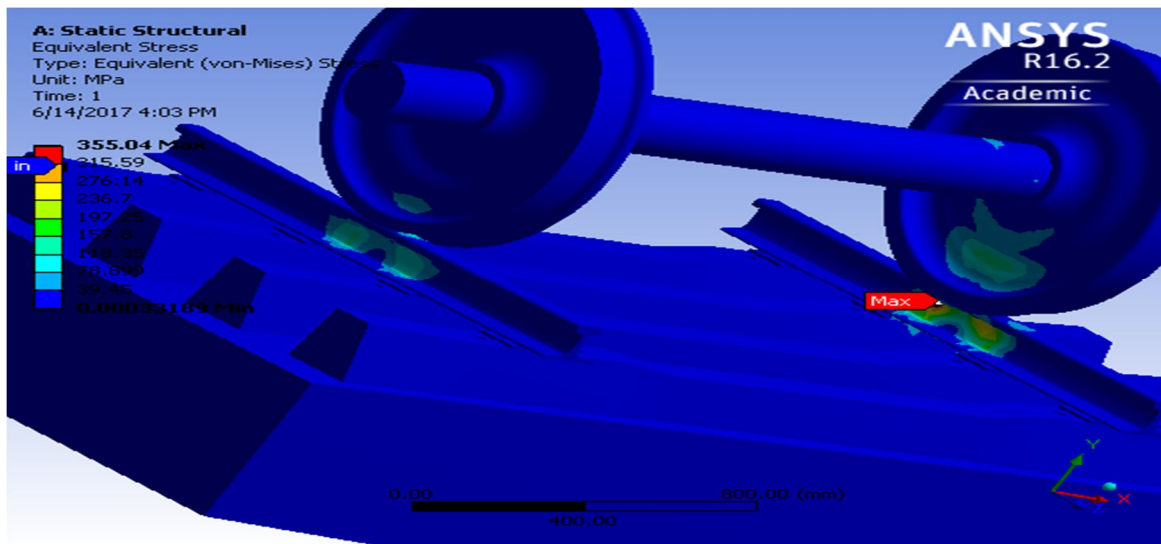
Fig 6. Maximum Equivalent Von-Mises stress in Rail-Wheel assembly for 835m curvature
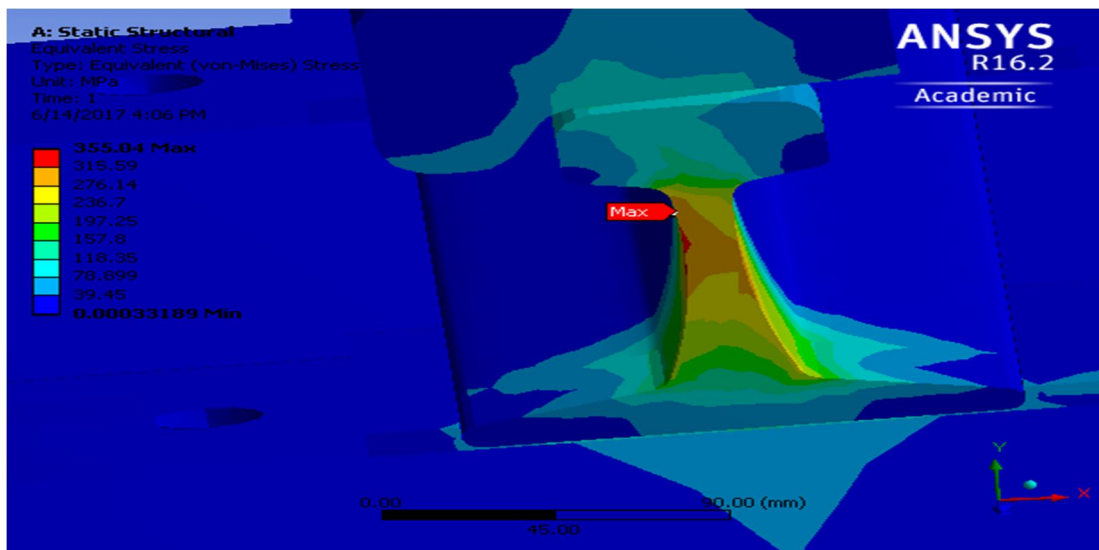
Fig 7. Sectional view of Maximum Equivalent Von-Mises stress in Right railof 835m curvature.
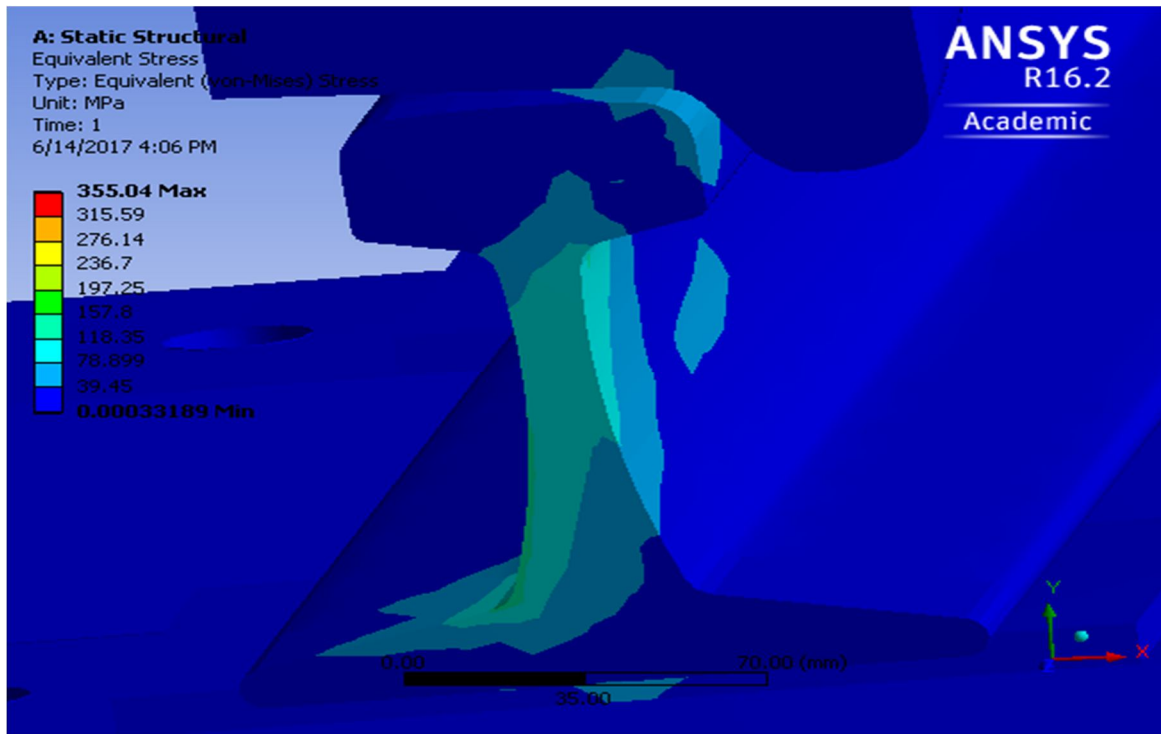
Fig8. Sectional view of Maximum Equivalent Von-Mises stress in left rail with 835m curvature.

Fig 5 shows that equivalent Von-Mises stress in the right and left rail with 800m curve radius and the units of stresses are in MPa. The maximum Von Mises equivalent stress in the left and right rail is 198 MPa and 355 MPa respectively.

The maximum Von Mises equivalent stress in the left rail occurs at a depth approximate 3-4 mm below the rail head, and that of the right rail takes place at the surface. The sectional views of right and left rail are shown in fig 6, fig 7. It indicates that the failure chances of right rail are more when compared with the left rail while it passing through sharp radius curve.

The distributions of the Von Mises equivalent stress in the left and right rail vary little with curve radius, The effect of curve radius on the maximum Von Mises equivalent stress in the left and right rail is plotted in fig 8.
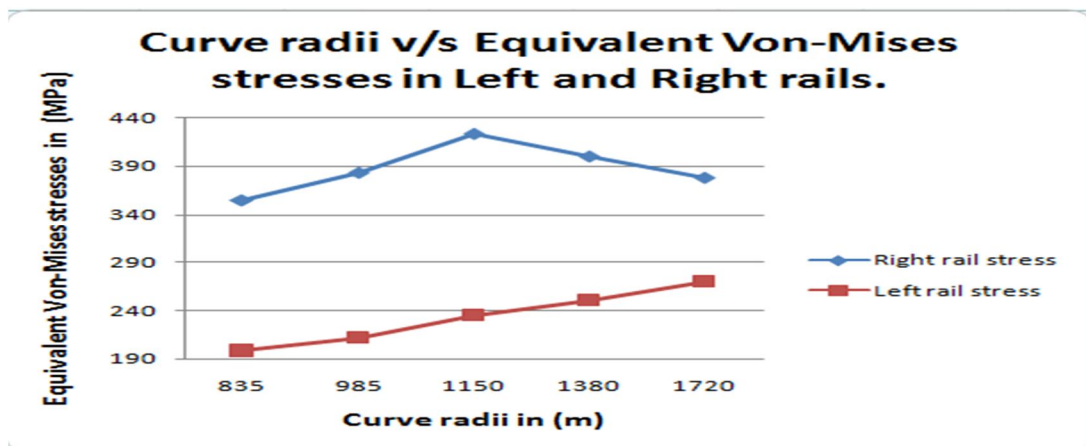
Figure 9. Variation of the maximum Von Mises equivalent stress in the left & right rail with different curve radii.

The wheel and rail are having a rolling contact. The normal load is supported by the contact forces between the wheel and rail. The repeated type of high magnitude of contact stresses at the rail wheel interface causes a fatigue loading. For obtaining the contact condition and the stresses at the interface, the model proposed by Li Ping Sun is used[10].
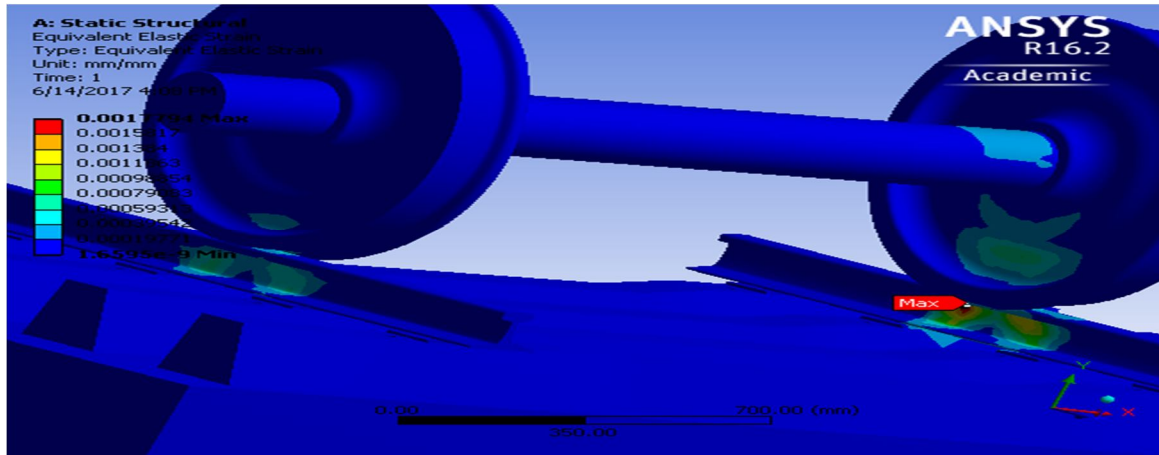
Fig 10. Maximum Equivalent elastic strain in rail-wheel assembly with 835m curvature.

The material of wheel is assumed to be forged steel. The yield surface is described by the von Mises yield criterion. Material parameters with Bi-linear kinematic hardening used for the FE simulations have been given in Table 1.

Curve Radius(m)	Equivalent elastic strain
835	0.00177
985	0.001922
1150	0.00212
1380	0.00201
1720	0.00190

Table 2. Equivalent elastic strain for different Curve radii.

fatigue analysis uses results obtained from the static structural analysis. Equivalent von-mises stress, equivalent strains are used for calculating life, damage, safety factor. Here Goodman theory, equivalent von-mises stress values are used for conducting fatigue analysis.
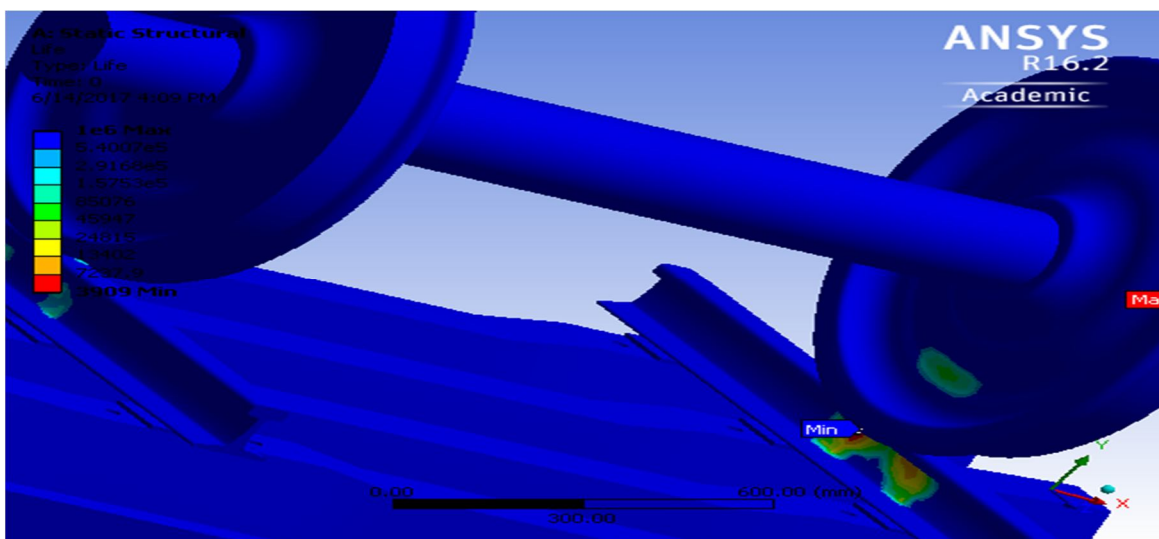
Fig 11.Life of rail-wheel assembly with 835m curvature.

Fig9.Shows that the life of rail-wheel assembly with 835m curvature the minimum life is obtained at point of contact of right rail-wheel. The value is 3909 cycles.
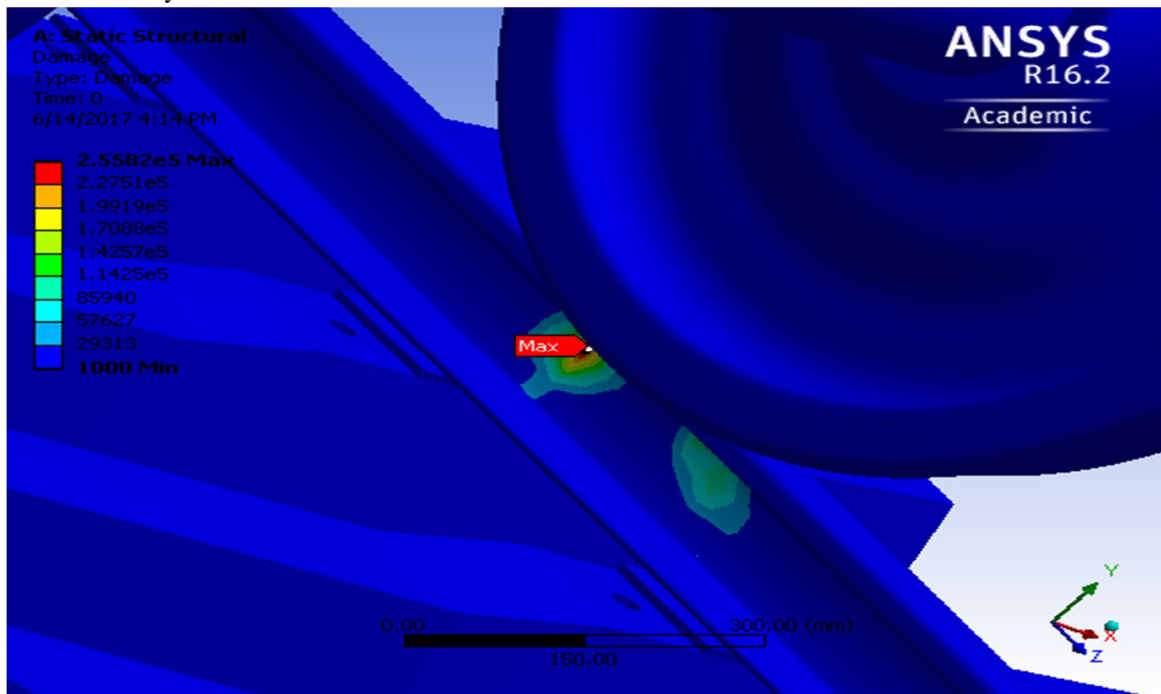
Fig 12.Damage of rail-wheel assembly with 835m curvature.

Fig 10 shows that the damage of a rail-wheel assembly with 835m curve radius. The maximum damage is happened at 2.558×10^5 cycles. And maximum damage is obtained at the point where maximum stress is obtained.
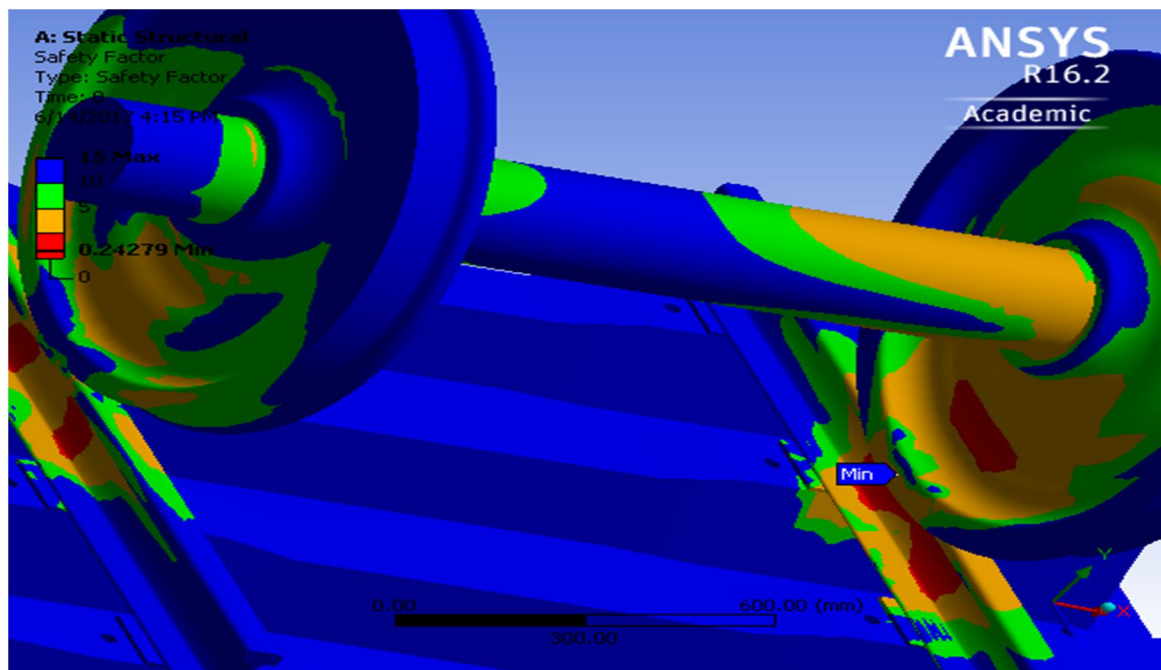

Fig 13.Safety factor of rail-wheel assembly with 835m curvature.

Fig11. Shows that safety factor of a rail-wheel assembly with 835m curve radius. The minimum factor of safety is obtained at the contact of left and right rail-wheels. The value of minimum factor of safety is 0.2427.
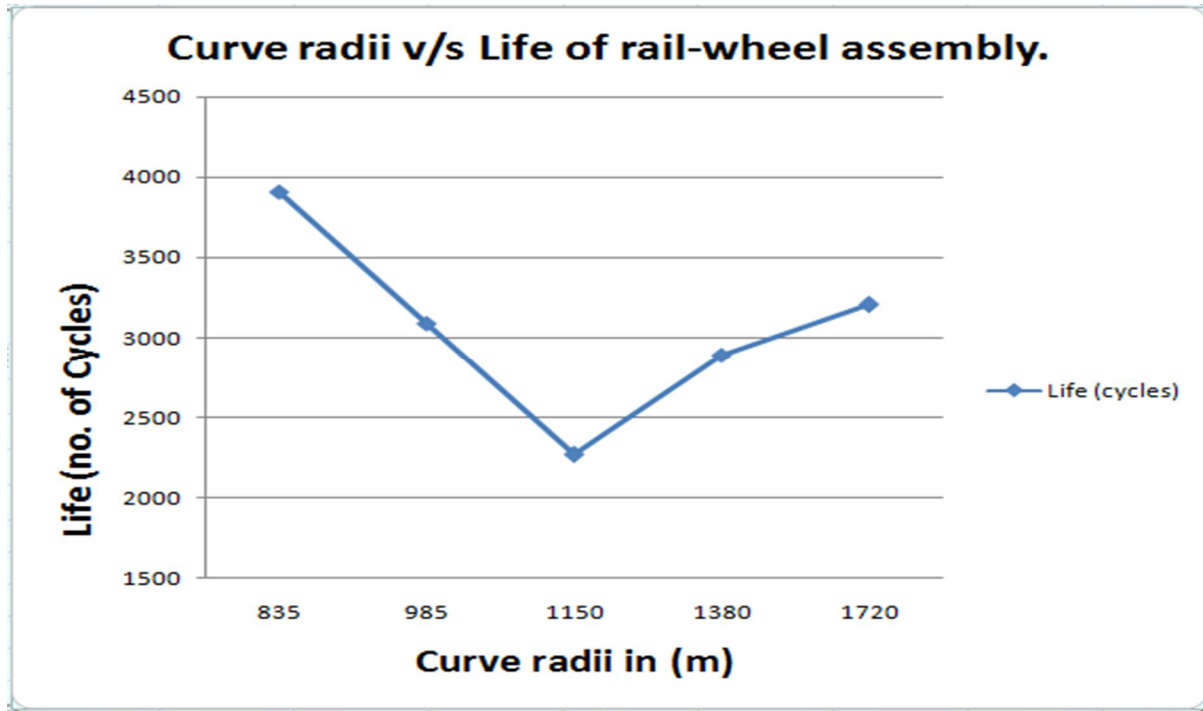
Figure14. Variation of the Life of rail-wheel assembly with different curve radii.

Fig12. Shows the variation of life with different curve radii. It is clear that stress is vary with the curve radius, and life of an assembly is varies with the stress. Super-elevation is inversely proportional to the Curve radius.
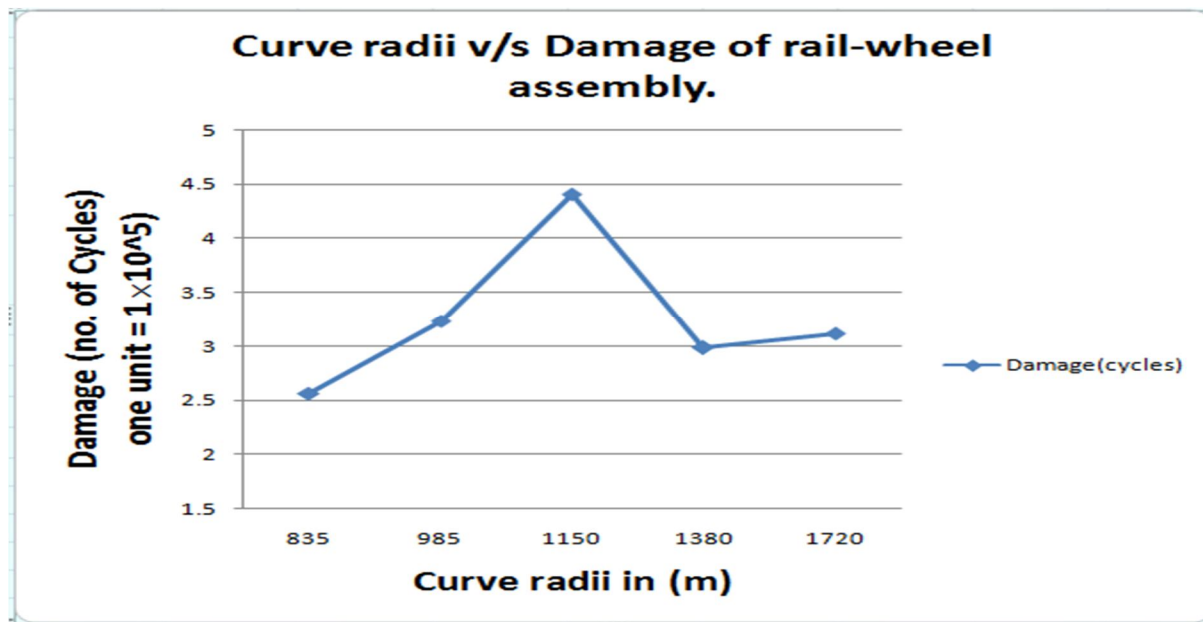

Figure15. Damage of a rail-wheel assembly with different curve radii.

Fig12. Shows that the variation of damage with the curve radii. The maximum damage is occurs at the 1150m curve radius and the value is 4.411×10^5 cycles.
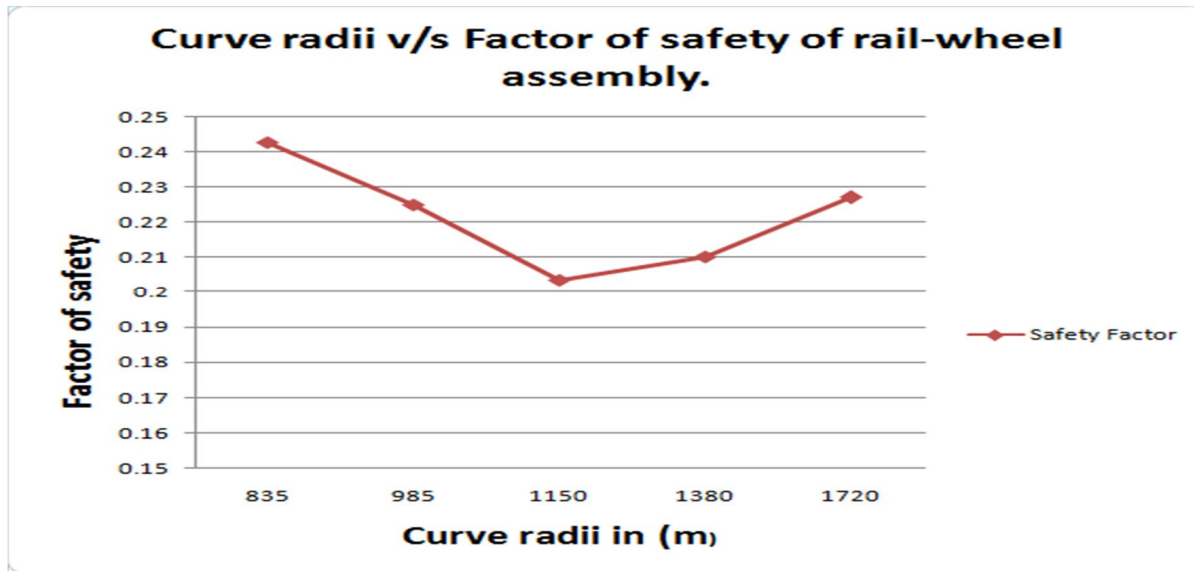
Figure16. Safety factor of rail-wheel assembly with different curve radii.

Fig13. Shows that Factor of safety for different curve radii. Minimum factor of safety is obtained at the 1150 curve radius because the maximum stress is obtained at that point.

A. Modal Analysis Results

First six mode shapes and frequencies are calculated and maximum deformation is also calculated. Analysis is carried for different curve radiuses. Fig14.shows the six mode shapes.
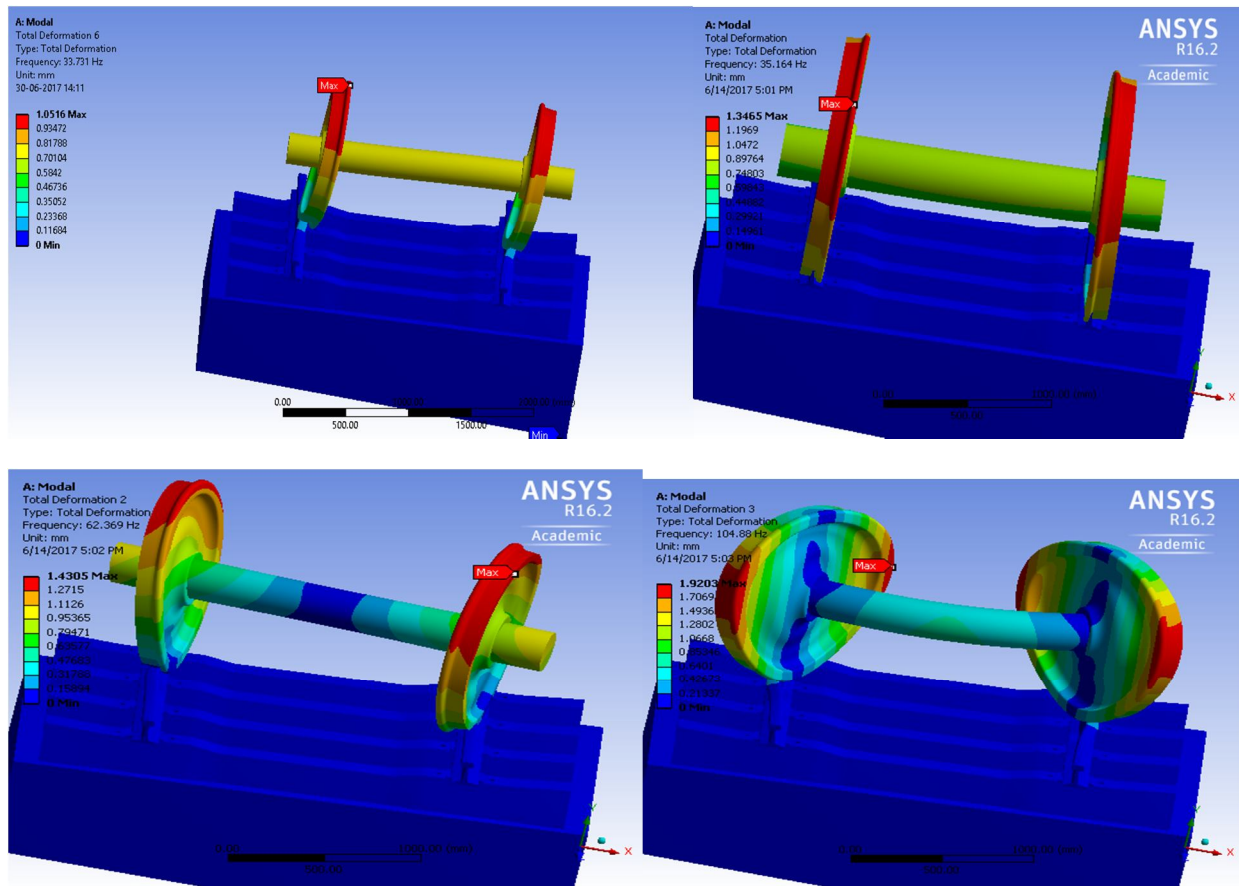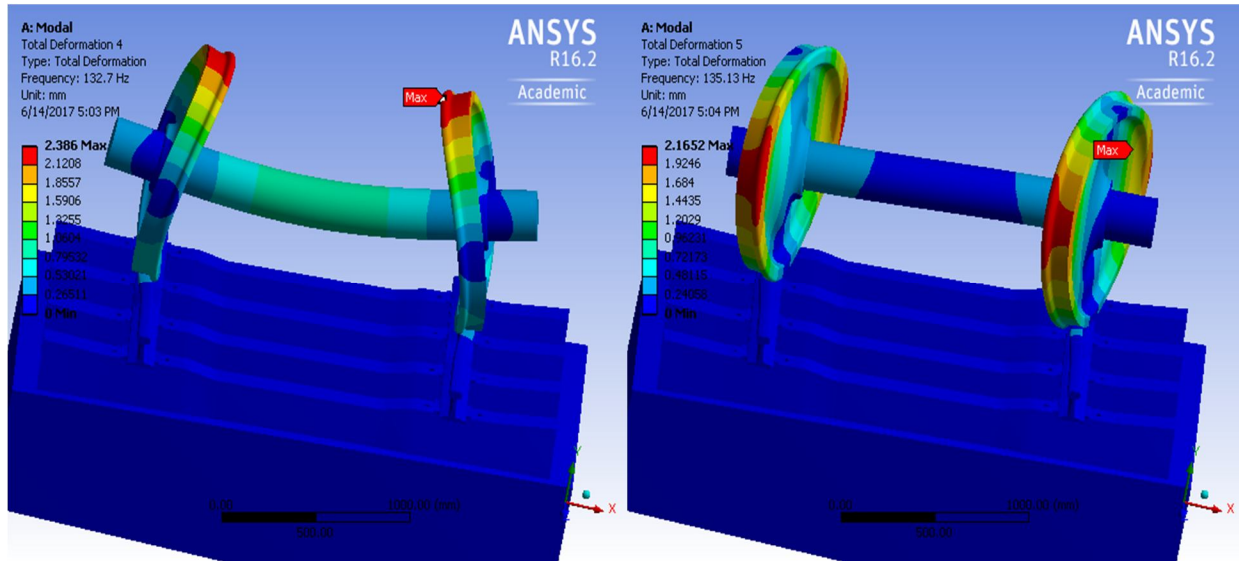
Fig 17. Shows the first Six mode shapes of Rail-wheel assembly with 835m curve radius.

Mode shape number.	Frequency (Hz)	Total deformation (mm)
First mode shape	33.731	1.0516
Second mode shape	35.164	1.346
Third mode shape	66.369	1.430
Fourth mode shape	104.88	1.920
Fifth mode shape	132.7	2.386
Sixth mode shape	135.13	2.165

Table3. Frequencies & total deformation for first six mode shapes of a 835m curve radius rail-wheel assembly.

IV. CONCLUSIONS

In this study, a numerical analysis of Equivalent von-mises stress, equivalent elastic strain and fatigue, modal analysis of 3D Rail–Wheel contact is successfully carried out using Finite Element Analysis with ANSYS Workbench 16.2.Application of realistic FE loading and boundary conditions has played an important role in getting accurate results.

The results from the present investigation are summarized as follows:

When the wheel is in contact with rail corner during train passing through curved track, the maximum Von-Mises equivalent stress in the left rail occurs at a depth approximate 3-4 mm below the rail head, and that of the right rail takes place at the surface of rail corner. The maximum Von-Mises equivalent stress in the right rail is bigger than that of the left rail. It is a natural result that failure chances of the right rail are more than that of the left rail.

It is observed that fatigue life depends on the Equivalent stress, and the stress values are varying with the curve radiuses. The minimum life of an assembly is observed at right side of an assembly.

Damage of an assembly is also calculated. From the results damage is varying with the curve radius, the maximum damage is occurs at the right side of an rail-wheel assembly. And the damage failure chances of an right side rail is more.

Safety factor results shows that the minimum value of safety is obtained at the both left and right rail-wheel assembly. And it is affected by the curve radius.

Frequencies obtained from the modal analysis are within the standard rolling contact frequency range[12]. If we increase this frequency there will be reduction in unbalanced forces.

V. ACKNOWLEDGMENT

REFERENCES

[1] Vinod Kumar and Gurdeep Singh "Investigation on Fatigue Life of Rail-Wheel Assembly using Finite Element Analysis" 978-1-5090-1555-9/16/2016 IEEE.

[2] Mr.Prabhakar Purushothaman and Dr.Prashanth Thankachan "Hertz Contact Stress Analysis and Validation Using Finite Element Analysis",Volume 2 Issue XI, November 2014, ISSN: 2321-9653.

[3] T. Telliskivi, U.Olofsson, "Contact mechanics analysis of measuredwheel-rail profiles using the finite element method", Proc Inst MechEng Part F J Rail Rapid Transit, vol215, pp. 65-72, 2001.

[4] M.A. Arslan and O. Kayabaşi, "3-D Rail–Wheel contact analysis usingFEA", Advances in Engineering Software,Vol. 45, pp.325–331, 2012.

[5] Aleksander Sladkowski "Analysis of wheel–rail interaction using FE software", Wear 258 (2005) 1217–1223,Elsevier

[6] Y. Liu, L. Liu, and S. Mahadevan, "Analysis of subsurface crackpropagation under rolling contact loading in railroad wheels usingFEM",Engineering Fracture Mechanics, Vol. 74 , pp. 2659-2674, 2007.

[7] W. Yan, F.D. Fischer, "Applicability of the hertz contact theory to rail-wheelcontact problems", Arch Appl Mech, vol70, pp. 1267-1273,May 2000.

[8] M. Pau, F. Aymerich, F. Ginesu, "Distribution of contact pressure inwheel–rail contact area", Wear, vol. 253, pp. 265-274, July 2002.

[9] Sunil Kumar Sharma and Anil Kumar "Dynamics Analysis of Wheel Rail Contact Using FEA" Procedia Engineering 144 (2016) 1119 – 1128, Elsevier.

[10] Li Ping Sun and Jian Zhang "Analyses on Wheel-Rail Contact Using Finite Element Method During Passing through curved track" 978-0-7695-3583-8,IEEE

[11] Nirmal K Mandal "Finite element analysis of the mechanical behavior of insulated rail joints due to impact loadings", J Rail and Rapid Transit2016, Vol. 230(3) 759–773.

[12] Naveen Garg & Omkar sharma "Noise emission of transit trains at curvature due to track lubrication" Indian J pure & apply Physiscs,Vol 48, pp. 881-885,December 2010.

[13] J.P.Srivastava and P.K.Sarkar "Contact Stress Analysis in Wheel–Rail by Hertzian Method and Finite Element Method" J. Inst. Eng. India Ser. C (October–December 2014) 95(4):319 -325,Springer.

[14] Institution of Permanent Way Engineers (India),TECHNICAL DIARY 2008 .

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089   (24*7 Support on Whatsapp)