iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 2 **Issue:** XI **Month of publication:** November 2014

DOI:

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

Hertz Contact Stress Analysis and Validation Using Finite Element Analysis

Mr. Prabhakar Purushothaman[1], Dr. Prashanth Thankachan[2]

[1] *Design Engineer- FEA*, [2] *Chief Technology officer - R & D, UCAM Pvt Ltd, Bangalore*

Abstract— ***In general machines are designed with a set of elements to reduce cost, ease of assembly and manufacturability etc. One also needs to address stress issues at the contact regions between any two elements, stress is induced when a load is applied to two elastic solids in contact. If not considered and addressed adequately serious flaws can occur within the mechanical design and the end product may fail to qualify. Stresses formed by the contact of two radii can cause extremely high stresses, the application and evaluation of Hertzian contact stress equations can estimate maximum stresses produced and ways to mitigate can be sought. Hertz developed a theory to calculate the contact area and pressure between the two surfaces and predict the resulting compression and stress induced in the objects. The roller bearing assembly and spur gear pair assembly is an example were the assembly undergoes fatigue failure due to contact stresses. This paper discusses the hertz contact theory validation using finite element Analysis.***

Keywords— ***Hertz Contact stress, Contact FE analysis, Stresses within rollers, Roller contact analysis and Hertz contact stress calculations.***

I. INTRODUCTION

Study of deformation of solids under contact is called contact mechanics, comprising of mechanics of material and continuum mechanics. Contact mechanics provides the information for safe and energy efficient design of mechanical elements in contact, while continuum mechanics provides for analysis of the kinematics and the mechanical behavior of materials modelled as a continuous mass rather than as discrete particles. Heinrich Hertz introduced the idea on contact mechanics in 1982. Hertz stress refers to the stress and deformation generated on two cylindrical rollers in contact under applied load. The stresses in between two rollers are critical, as a single line contact takes place between the rollers as shown in Fig. 1(a). As the force flow lines will be intersecting at the contact region stress concentration takes place and high stress generated at contact occurs as shown in Fig. 1(b). In this paper Hertz contact stress values obtained by analytical calculation and Hertz contact stress calculator is validated using finite element analysis.[1]

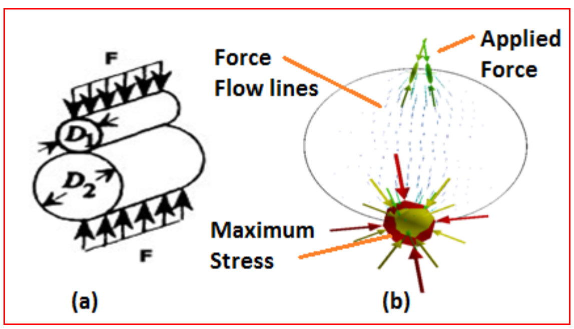


Fig.1(a) Rollers in contact (b) Force flow lines in rollers

II. ASSUMPTIONS AND IDEALIZATIONS

The following idealization considerations for the analysis of the problem

A. It has been considered that both the cylinders are parallel to each other and single line contact takes place between cylinders. The representation of the same is made in Fig.1(a)
B. Two half cylinders are used instead of full cylinder for modelling assuming that each cylinder contains a uniform cross section along its length.
C. It is being considered that both the cylinders are made of homogeneous isotropic materials and is considered to be made of steel
D. Both the cylinders namely the one in contact and the one considered as target are considered to be deformable bodies

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

and are considered to be behaving in linearly elastic manner and are considered to be exhibiting plane stress behaviour

E. It is also assumed that both surfaces that are in contact has a smooth surface and shows no frictional behaviour when in contact.

III. SOLUTION STRATEGY

The paper attempts to perform contact analysis and obtain appropriate stress values induced at the contacts. Based on the data the stress values can be optimized by modifying either the load parameter or the roller size. Validation of the finite element analysis result is done with values obtained using manual analytical calculation and a Hertz stress calculator.

Data Assumed:

Diameter of roller 1 in mm (D1)	=188.12
Radius of roller in mm (R1)	= 94.06
Diameter of roller 2 in mm (D2)	= 183.78
Radius of roller in mm (R2)	= 91.89
Force in N (F)	= 1049
Young's modulus in MPa (E_1 & E_2)	= 2 x 10^5
Poisson's ratio (v)	= 0.3

A. Finding contact patch width (b) [2]

$$b = \sqrt{\frac{2x\,F/t}{\pi} x \frac{D_1 \, x \, D_2}{D_1 + D_2} x \left(\frac{1-V^2}{E_1} + \frac{1-V^2}{E_2}\right)}$$

$$b = \sqrt{\frac{2x1049}{\pi} x \frac{188.12 \, x \, 183.78}{188.12 + 183.78} x \left(\frac{1-0.3^2}{2 \, X \, 10^5} + \frac{1-0.3^2}{2 \, X \, 10^5}\right)}$$

$$b = \sqrt{667.81 \, x \frac{34572.69}{371.9} x \, 9.1 \, x \, 10^{-6}}$$

b = 0.751 mm

Patch width = 2b
= 2 x 0.751
= 1.502 mm

B. Finding Maximum Pressure (P) [2]

$$P = \sqrt{\frac{2 \, x \, F}{\pi \, b \, l}}$$

$$= \frac{2 \, x \, 1049}{\pi \, X \, 0.751}$$

= 889.23 MPa

C. Stress induced in the cylinder along Z-axis is given by[2]

$$\sigma_z = -2 \, P \, x \, v \left[\sqrt{1 + \left(\frac{y}{b}\right)^2} - \frac{y}{b}\right]$$

= -2 x 889.23 x 0.3

σ_z = -533.53 MPa

D. Stress induced in the cylinder along X-axis is given by[2]

$$\sigma_x = -P \left[\left[\sqrt{1 + \left(\frac{y}{b}\right)^2}\right] x \left(2 - \left(1 + \frac{y}{b}^2\right)^{-1}\right]\right]$$

= -889.23 (1) at y = 0

σ_x = -889.23 MPa

E. Stress induced in the cylinder along y-axis is given by[2]

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

$$\sigma_y = -2\,P\,x\left[1+\left(\frac{y}{b}\right)^2\right]^{-1/2}$$

$$\sigma_y = -889.23 \text{ MPa}$$

F. Shear stress is calculated by[2]

$$\tau_{xy} = 0.3 \text{ x } P_{max}$$
$$= 0.3 \text{ x } 889.23$$
$$= 266.97 \text{ MPa}$$

G. The approach or centre between two cylinders are[2]

$$\delta = \frac{2 \text{ x } F\,(1-\nu^2)}{\pi E l}\left(\frac{2}{3} + \ln\frac{D_1}{b} + \ln\frac{D_2}{b}\right)$$
$$= \frac{2 \text{ x } 1049\,(1-0.3^2)}{\pi \text{ x } 2 \text{ x } 10^5}\left(\frac{2}{3} + \ln\frac{188.12}{0.751} + \ln\frac{183.78}{0.751}\right)$$
$$= 3.03 \text{ x } 10^{-3}(0.66 + 5.52 + 10.10)$$
$$= 0.4884 \text{ mm}$$

IV. RESULT OBTAINED IN HERTZ STRESS CALCULATOR SOFTWARE

In Hertz stress calculator the data assumed for calculation provided as input for calculating contact stress is shown in Fig. 2

INPUT PARAMETERS				
Parameter	Symbol	Object-1	Object-2	Unit
Object shape		Cylinder	Cylinder	
Poisson's ratio	ν_1,ν_2	0.3	0.3	
Elastic modulus	E_1,E_2	200	200	GPa
Diameter of object	d_1,d_2	188.12	183.78	mm
Force	F	1049		N
Line contact length	l	1		mm
Calculate				

Fig. 2 Input provided in Hertz Stress calculator

For the provided input the results of maximum shear stress, contact patch width, contact pressure and approach distance obtained is shown below in Fig 3.

RESULTS				
Parameter	Symbol	Object-1	Object-2	Unit
Maximum Hertzian contact pressure	p_{max}	888.5		MPa
Max shear stress	τ_{max}	266.8	266.8	
Depth of max shear stress	z	0.591	0.591	mm
Rectangular contact area width	2b	1.503		

Fig. 3 Output result of Hertz Stress calculator

The graph below in Fig. 4 shows the result of distribution of stress for various contact depth. The X axis is plotted as depth of contact surface in mm and Y-axis refers to the stress in MPa

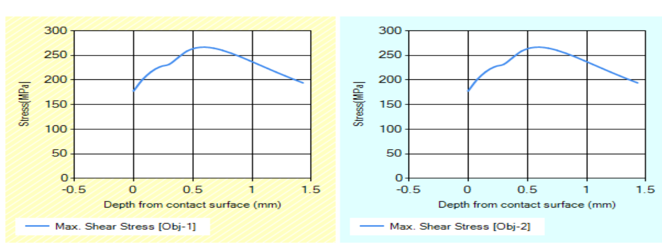


Fig.4 Stress distribution in roller 1 and roller 2

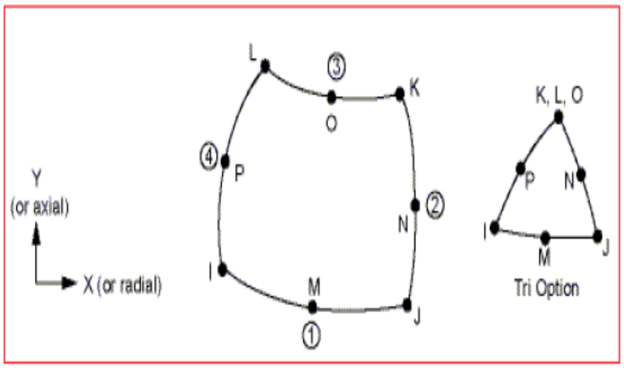
ISSN: 2321-9653

V. FINITE ELEMENT MODELLING

The two rollers considered to be in contact is idealized as per plane stress theory criteria, in order to improve the accuracy of result and reduce computation time. While plane stress provides for 2D approximation condition to simplify the finite element problem.[3] When the problem is approximated as per this theory, need not to be analyse the entire 3D model instead only a plane of cross section needs to be considered with certain thickness. The result of entire model is calculated using result obtained from 2D finite element analysis result. The conditions to be satisfied are the sections to be uniform throughout length and forces should not act along the thickness. In order to find the contact stress between the rollers satisfying the above criteria, idealization of the problem as per plane stress criteria is needed. The analysis carried out considering one plane of the cylinders and loads are applied as the cylinders will not be changing orientation during analysis. While both the cylinders are in contact only a line contact exist in order to do analysis. A very fine mesh is required around the region of contact in order to capture accurate results. The conditions of flexible to flexible contact with deformable bodies are applied to the nodes in contact.

VI. ELEMENT SELECTION FOR FINITE ELEMENT ANALYSIS

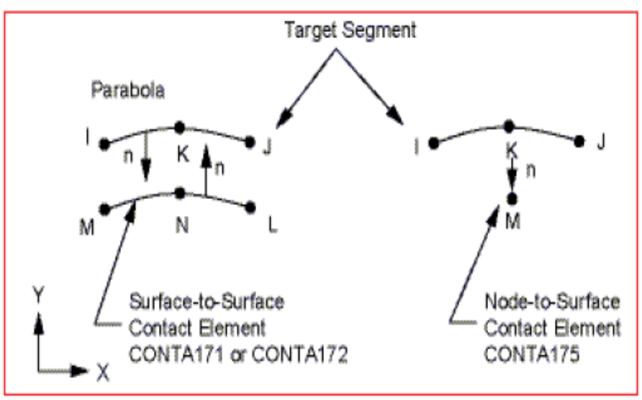
As the problem is simplified as per plane stress criteria, the element should be 2D element and should be capable of capturing stress. Therefore the element selected is solid- 8node82 (Plane 82). The Plane 82 is 2D higher order 8 nodded element. It provides more accurate results for irregular shapes and is most suited for curved geometries. The mapped mesh is created only



around the region of contact and the free mesh is created in the region of less concern.

Fig.5 Plane 82 Element

The other element selected for FE modelling is Contact – 2D target169 is selected for TARGE169. It is used to represent target surface for contact elements describing the boundary of deformable body which are in contact with target surface. It provides flexible to flexible contact system which is required for Hertz contact analysis. As the problem being analysed is contact issues between two cylindrical rollers were node to surface contact is used by selecting CONTA175. Contact-point to surface 175 is selected for CONTA175. It is used to represent the contact and sliding between the two surfaces between node and surface or



between line and surface.

Fig. 6 Contact 175 Element

The h-method is adopted for solving, while h-method provides for obtaining accurate results by converging the result with respective to number of iterations. The fine mesh is required to obtain a better result, however use of fine mesh throughout the

ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

section increases the computation time. In order to reduce the computation time, a small boundary is created around the contact and meshed to the size of 0.1 mesh size as mapped and rest of the portion is free meshed to larger element size 5 mm size as shown below in the Fig. 7

.

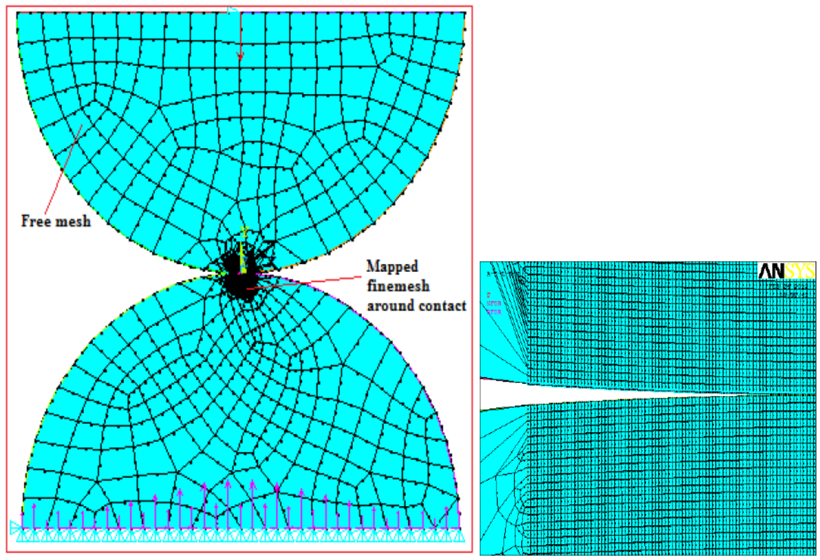


Fig. 7 Mesh generated for FE analysis

VII. BOUNDARY CONDITIONS

For this analysis only the two half portion of the cylinders are considered in which the load is applied to one half portion of the cylinder, in the other half no load is applied. In order to avoid the displacement of the bottom cylinder, the nodes at the bottom most line at boundary of bottom cylinder is constrained at X and Y axes together. In the top half cylinder at the middle node were load is to be applied, only the X axis is constrained in order to transfer the entire load axially. The load of 1049 N is applied in negative direction on the same middle node as a constant load that is transferred from the top half of the cylinder to bottom half of the cylinder.

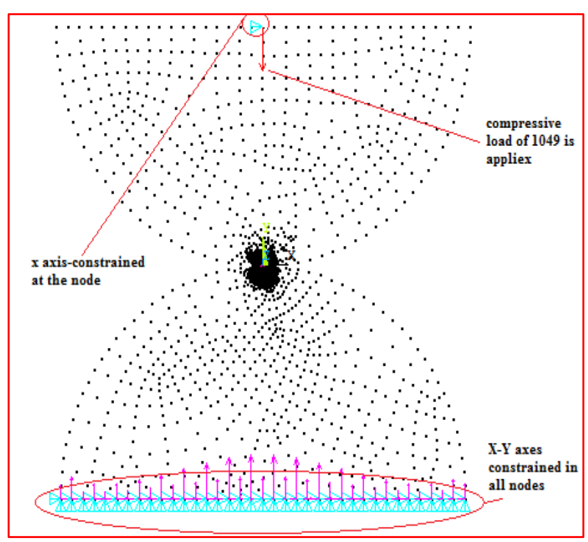
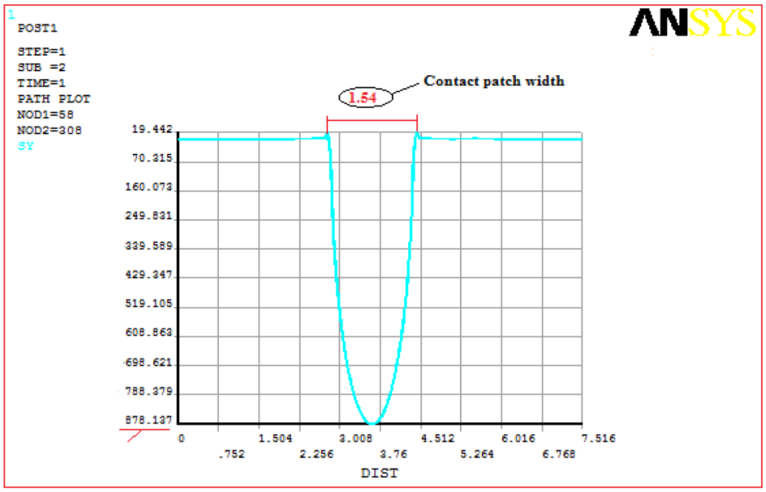


Fig. 8 Boundary condition for analysis

VIII. CONTACT PATCH WIDTH RESULT

ISSN: 2321-9653

The contact patch width result provides the result of length of contact or the length of line of contact between two rollers for applied load. In Ansys the result is plotted by selecting the series of nodes at the contact region. The Fig. 9 shows the



graph obtained from Ansys were in X-axis represents dimension in mm and in Y-axis represents node number selected in finite element model. The result obtained for contact patch width is 1.54 mm, the difference in result from analytical calculation is 2.46 % and difference in result from Hertz contact stress calculator is 2.46%.

Fig. 9 Contact Patch width graph

IX. MAXIMUM SHEAR STRESS RESULT

Shear stress develops under applied load as the rollers in contact oppose each other and will be shear stress developed and will maximum at 45°. The magnitude of shear stress depends on the force and type of contact. In Fig. 10 it can be seen that stresses occur at an angle as shown in the blue and red region. The maximum shear stress obtained from Ansys is 268.61MPa and the result obtained by the analytical solution is 266.97MPa therefore the difference is about 0.61 %. The result obtained from Hertz contact stress calculator is 266.8 MPa, the difference in result is about 0.67 %.

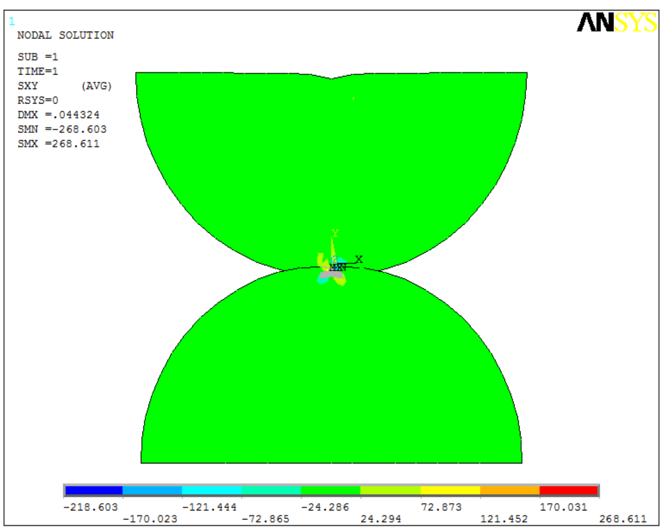


Fig. 10 Maximum Shear Stress Result

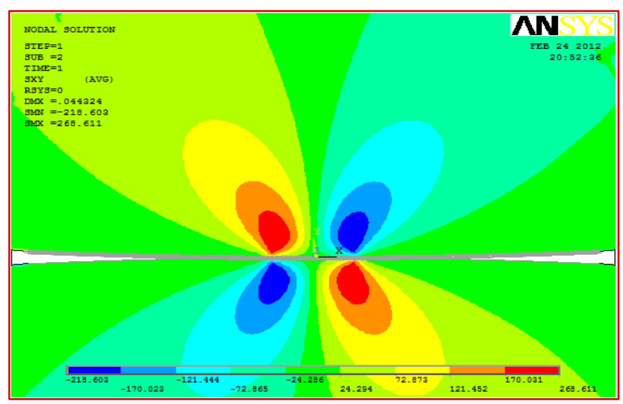


Fig11. Exaggerated view of deflection pattern

X. APPROACH DISTANCE BETWEEN CENTERS OF THE CYLINDERS

The approach distance between the cylinders refers to the two cylinders in contact and the load applied to one of them. It undergoes either elastic or plastic deformation. This depends primarily on the yield strength of the material, as the stress value exceeds the yield strength for a given applied load the deformation will be permanent and if the stress value is lesser than yield strength for the load applied the deformation will be temporary and the rollers will regain the original shape on removal of load. However in this case for analysis proper material yield property is not defined, therefore the deformation result of 0.044 mm is valid but it may not be defined as elastic or plastic deformation. The observation from the Fig. 11 is, for the compressive load of 1049 N applied to the upper cylinder and as the bottom cylinder is fully constrained the upper cylinder tries to penetrate inside the bottom cylinder and the bottom cylinder will resist the upper cylinder but still the deformation of 0.044mm takes place. As the result obtained is compared with the analytical solution the result obtained from the Ansys is 0.044mm and the result obtained by the analytical solution is 0.048 the difference in the result is 8.33%.

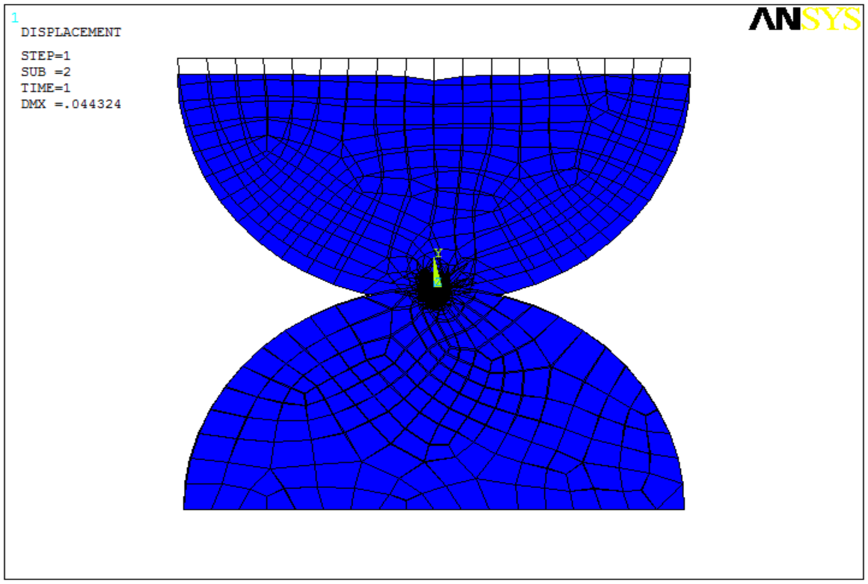


Fig. 12 Displacement result of Analysis

XI. RESULTS AND DISCUSSIONS

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. As the difference in result is within 10% this difference is due to the approximation made as two half cylinder instead of full cylinder. In case of contact patch width finite analysis result shown in graph was plotted by selecting nodes at contact region the element size and node selection for plotting graph was the main criteria for result and the contact patch width is arrived manually from graph therefore this difference is acceptable. Since only half cylinder is modelled instead of full cylinder load is applied with constrain on node was the influence for the difference in approach distance result. The result shows in order to obtain proper results in finite element analysis, proper elements has to be selected with suitable degrees of freedom, assigning appropriate contacts between element and by providing accurate boundary conditions helps to compute most accurate contact stress values. The value of contact stress is very important, as the stress value changes with contact area. The higher the contact area the stress generated will be less and for lesser contact area high stress will be generated. In various engineering applications line contact exists between bearing rollers, spur gears etc the contact stress are more critical and only point contact exists between ball bearing, ball screw etc finding the contact stresses are further critical therefore in order to capture accurate results proper care to be taken while meshing and assigning contacts depending on complexity of problem. The contact stress between rollers is important in order to ensure the stress generated is within elastic limits, this also helps predict accurately the fatigue life by plotting the value of stress in S-N curve (stress vs number of cycles) of the material. Based on required fatigue life the stress values can be optimized by modifying permissible load carrying capacity or by changing roller dimensions. The comparison of various results is shown below.

Table 1 Result comparison

Description	Analtical Calculation	Hertz Stress Calculator	FE analysis Result
Maximum Shear stress in MPa	266.97	266.8	268.61
Contact Patch Width in mm	1.503	1.502	1.54
Approach Distance in mm	0.0488	-	0.044

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

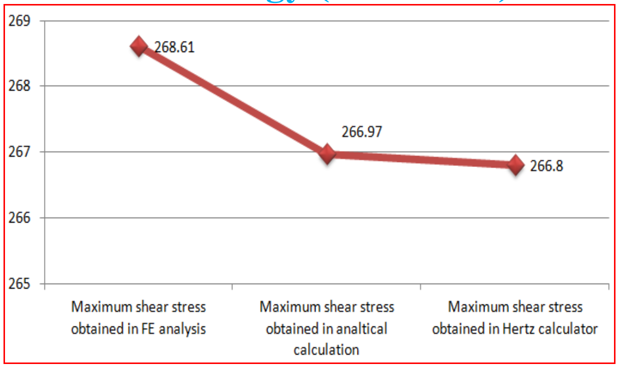


Fig.13 Shear stress result comparison

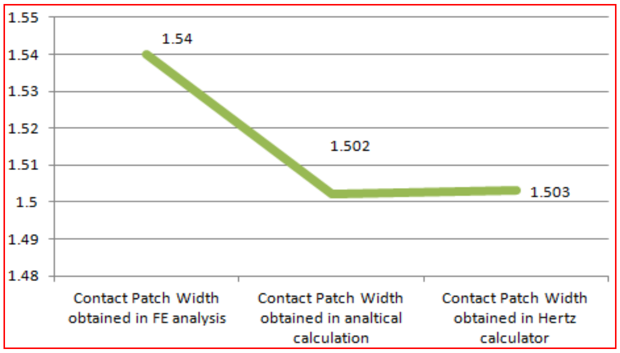
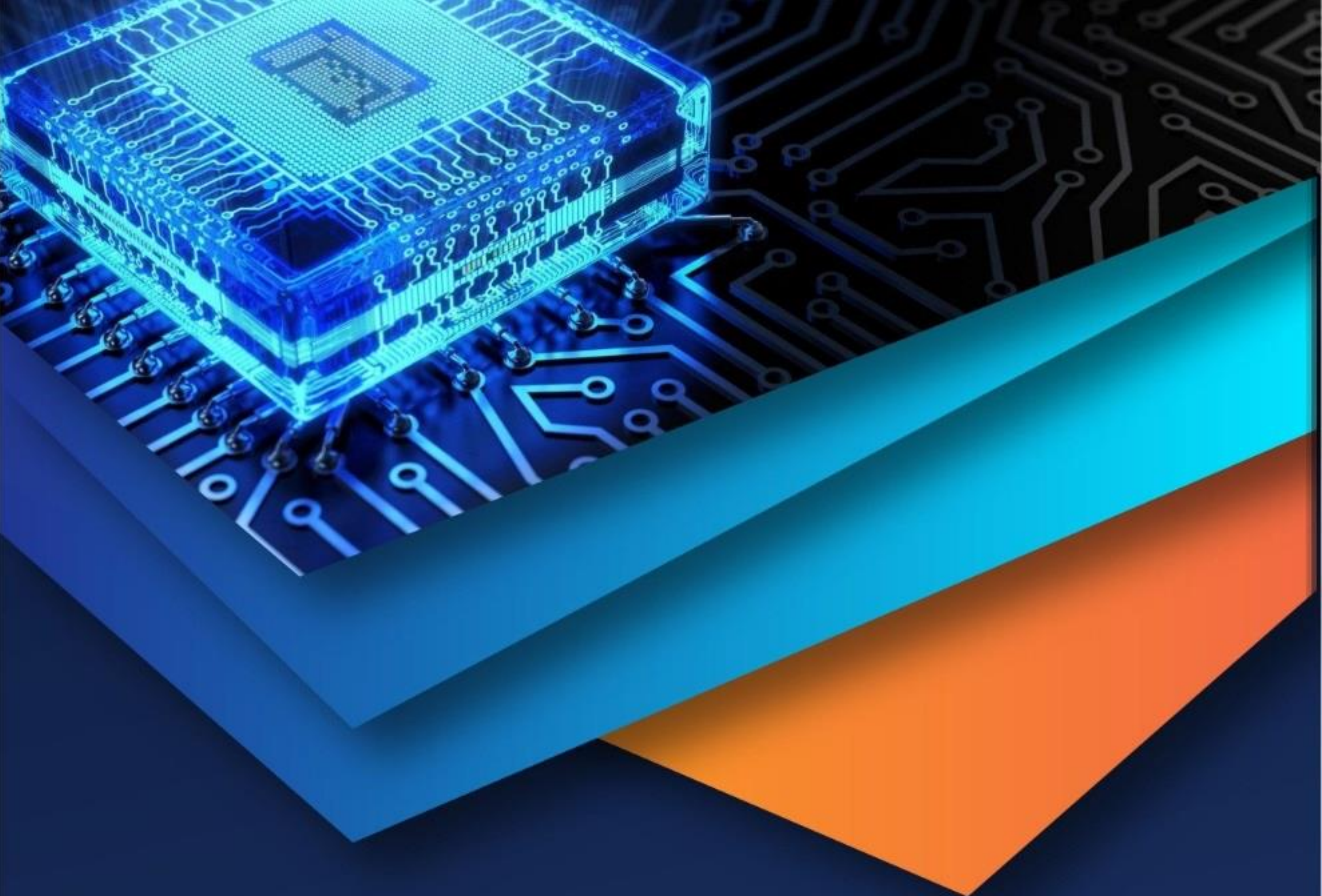


Fig.14 Contact patch width result comparison

REFERENCES

[1] Wikipedia article, *"Contact Mechanics"* available in web at http://en.wikipedia.org/wiki/Contact_mechanics
[2] Wei, Zeping *"Stresses and deformations in involute spur gears by finite element method"* Master Thesis, University of Saskatchewan, Oct 2004.
[3] Hertz Stress Calculator, available Online at http://www.amesweb.info/HertzianContact/HertzianContact.aspx#.VEtp5sXOns0 by advanced mechanical engineering solutions, retrieved on 20 10 2014.
[4] D Zeleniakiene "The comparative analysis of 2D and 3D micro structural models" internet.ktu.lt/lt/mokslas/zurnalai/mechanika/mech.../Leisys353.PDF Retrieved on 01 10 2014.
[5] Smith, J. O. Liu, C. K., "Stresses Due to Tangential and Normal Loads on an Elastic Solid with Applications to Some Contact Stress Problems", Journal of Applied Mechanics, June 1953
[6] Nicholas LeCain, "Tutorial of Hertzian Contact Stress Analysis" College of Optical Sciences, University of Arizona, Tucson, pp-1 to 6, 3 Dec 2011
[7] J.E. Shigley, C.R. Mischke, R.G. Budynas Mechanical Engineering Design, 7th Ed. pp 161-166, McGraw Hill, 2004.
[8] Vu-Quoc, L., Zhang, X., and Lesburg, L., 2000, ''A Normal Force- Displacement Model for Contacting Spheres Accounting for Plastic Deformation: Force-Driven Formulation,'' ASME J. Appl. Mech., 67, pp. 363–371.
[9] Liu, G., Wang, Q. J., and Lin, C., 1999, ''A Survey of Current Models for Simulating the Contact between Rough Surfaces,'' Tribal. Trans., 42, pp. 581–591.
[10] Bhushan, B., 1996, ''Contact Mechanics of Rough Surfaces in Tribology: Single Asperity Contact,'' Appl. Mech. Rev., 49, pp. 275–298.
[11] Reddy, J. N., "An Introduction to the Finite Element Method", 2nd Ed.,McGraw-Hill, New York. 1993.
[12] Owen, D. R. J., and Hinton, E., " Finite Elements in Plasticity: Theory and Practice", Pineridge Press, Swansea, UK. 1980
[13] Zhong, Z. H., "Finite Element Procedures for Contact Impact Problems", Oxford University Press, New York. 1993



10.22214/IJRASET

INDEX COPERNICUS

45.98

ISRA JIF

IMPACT FACTOR: 7.129

SJIFactor
TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

ISI

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY