Analysis of contact stress between two cylinders using Finite Element Method

Prabhakar K. Sakhare[1], Joglekar Bhimashankar I[2]

Abstract- *The objective of the contact stress analyses was to gain an understanding of the modeling and solution difficulties in contact problems and examine the contact stresses in the two cylinders. This paper reviews and analyses to verify the FEM contact model procedure; contact between two cylinders was modeled. To reduce computer time, only half cylinders were meshed in the model as shown in Figure. The fine meshed rectangular shaped elements were generated near contact areas shown. The dimensions of the elements are based on the half width of the contact area. The contact conditions are sensitive to the geometry of the contacting surfaces, which means that the finite element mesh near the contact zone needs to be highly refined. Finer meshing generally leads to a more accurate solution, but requires more time and system resources. It is recommended not to have a fine mesh everywhere in the model to reduce the computational requirements.*

Keywords- Contact Stress, Contact Width, Element, poisons ratio, FEM, Modulus of elasticity, Maximum Pressure, Target element. Graphical User Interface.

I. INTRODUCTION

Contact stress refers to the localized stresses that develop as two curved surfaces come in contact and deform slightly under the imposed loads. This amount of deformation is dependent on the modulus of elasticity of the material in contact. It gives the contact stress as a function of the normal contact force, the radii of curvature of both bodies and the modulus of elasticity of both bodies. Contact stress forms the foundation for the equations for load bearing capabilities in bearings, gears, and any other bodies where two surfaces are in contact. Contact problems present two significant difficulties. First, you generally do not know the regions of contact until you've run the problem. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner. With the rapid development of computational mechanics, however, great progress has been made in numerical analysis of the problem. Using the finite element method, many contact problems, ranging from relatively simple ones to quite complicated ones, can be solved with high accuracy. The Finite Element Method can be considered the favorite method to treat contact problems, because of its proven success in treating a wide range of engineering problem in areas of solid mechanics, fluid flow, heat transfer, and for electromagnetic field and coupled field problems. The ANSYS program has a comprehensive graphical user interface (GUI) that gives users easy, interactive access to program functions study, commands, documentation, and reference material. An intuitive menu system helps users navigate through the ANSYS program. Users can input data using a mouse, a keyboard, or a combination of both. General purpose finite element modeling package for numerically solving a wide variety of mechanical problems. These problems include: static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems. Here in this work only static structural analysis is done.

II. HERTZ THEORY OF ELASTIC CONTACT

When two curved bodies brought in contact they initially contact at single point or along a line. With the smallest application of load elastic deformation occurs and contact is made over a finite area. A method of determining the size of this region was first discovered by Heinrich Hertz in 1881. He assumed:

a. The size of contact area is small and frictionless.
b. Both the contacting surfaces are smooth and frictionless.
c. The deformation is elastic and can be calculated by treating each body as an elastic half space.

III. PROBLEM DEFINITION

The main objectives for this is to become familiar with the use of contact elements in ANSYS and to compare the well-known solution of the classical Hertz problem for two parallel elastic cylinders in contact to the solution from ANSYS models the contact stress in two parallel cylinders as shown in Figure 1. The two cylinders are pressed together with force F, which is distributed evenly along the length l, of the cylinders.

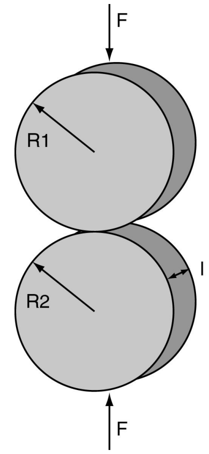

Fig.1 Parallel cylinders model

The radius of each cylinder is defined by R1 and R2. The radii can differ in size and sign or they can be equal. The parameters used in this analysis are defined as follows:-

R_1 =6.35mm (0.25 in)
R_2 =6.35mm (0.25 in)
ν_1=0.30,
ν_2=0.30,
E_1=206842.71($30*10^6$ Psi),
E_2=206842.71($30*10^6$ Psi),
F = 8.75 N/mm (50 lbs. /in (force per unit thickness),
L=25.4 mm (1inch).

Analytical solution to problem
Hertz Contact Stress Equations:

Contact stresses between two cylinders were shown in Figure 2. An ellipsoidal-prism pressure distribution is generated between the two contact areas.

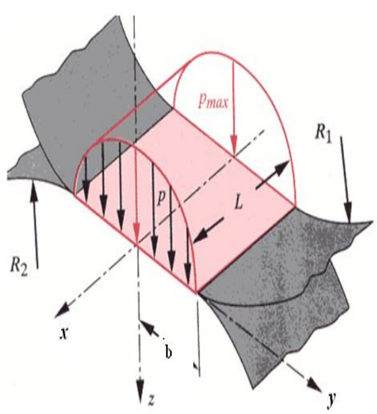

Fig.2 Ellipsoidal-prism pressure distribution

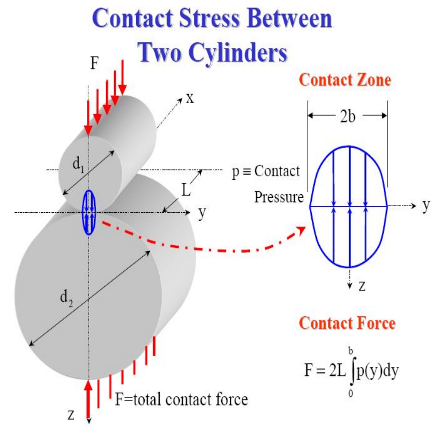

Fig. 3 Contact stress between two cylinders

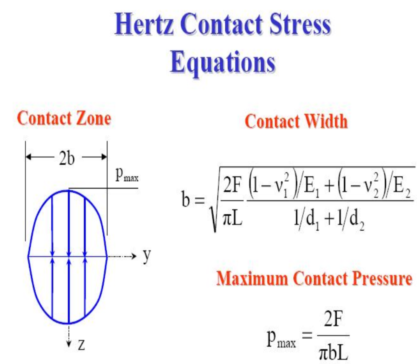

Fig.4. Contact Zone

From Figure 3 the width of the contact zone is 2b. If total contact force is F and contact pressure is p(x), there is a formula, which shows the relationship between the force F and the pressure p(x)

Maximum contact width and stress by contact width Equation:

$$\textit{Contact width, b} \sqrt{\frac{2F\left[\frac{1-\nu_1^2}{E_1}\right] + \left[\frac{1-\nu_2^2}{E_2}\right]}{\Pi L\left(\frac{1}{d_1} + \frac{1}{d_2}\right)}}$$

Where, d1 and d2 represents diameter of cylinders

$$b= \sqrt{\frac{50\times 2\times\left(\frac{1-0.3^2}{30\times 10^6}\right)+\left(\frac{1-0.3^2}{30\times 10^6}\right)}{\left[\Pi\times 1\times 2\times\left(\frac{1}{0.5}\right)\right]}}$$

b=0.000493in.

The maximum contact stress, $P_{max} = \frac{2F}{\pi b L}$

$$= \frac{2\times 50}{\Pi \times 0.000493 \times 1}$$

Maximum contact stress = 64466.35 Psi

IV. PROBLEM SOLVING PROCEDURE BY FEA

1. First of all select the preprocessor and then structural.
2. Then go to the preprocessor and then add element type and then select Solid as Quad 4 node 42. So as to consider thickness. Select second option plane stress with thickness as 0.1
3. Then add the real constant, click on add\edit\delete option and add the set 1 plane 42 and ok
4. To set as material property goes to material model, click on structural, then go to linear click elastic and then isotropic close the window.
5. Now go to modeling for creating the areas click create , areas as circle, select the type of circle as partial annulus.
6. Key points are created by input followings values.

x=0	X=0
Y=0	y=0.50
Rad=0.25	Rad=0.25
θ_1=-90	θ_1=90
θ_2=90	θ_2-=90
Apply	Ok

7. Now mesh the required area and then close the window for refining of mesh go to plot control select box zoom
8. Select modify mesh , refine at selected element to be refined mesh set the level of refinement as 1 minimal and ok close the window
9. For inputting the contact go to contact manager select contact wizard, select the target surface by choosing line and flexible choose surface to surface contact and then select the contact surface and then click to the next option for selecting coefficient of friction as 0.01. for setting as arrangement language method by create and close the window
10. Go to plot menu and select the re-plot option, now model is created in ANSYS.
11. For applying load various boundary conditions should be specified for constraint the motion select the displacement option in load menu, click all lines and select the line to which the motion is to be constrained. For constraining the motion along x-axis select UX option and set the value as zero. For fixing the lower end point of cylinder select the lower key point and click ok, and select all the degrees of freedom.
12. For applying load on the upper point of the cylinder select the force\moment option andapply the key point on the upper side.. the load is acting vertically downwards so input the FY force as -50
13. Now solve the problem by current LS and close the window.
14. For viewing the results go to general post procedure then go to plot results and choose the parameter as von-misses stress.
15. For viewing the result go to general post procedure then go to plot result and choose the parameter as von –misses' stress.

V. CONTACT ELEMENT ADVANTAGES, LIMITATIONS AND THEIR CONVERGENCE

Because of the simplicity of their formulation, the advantages of using contact elements are, They are easy to use they are simple to formulate, and they are easily accommodated into existing F.E. code.However, using contact elements poses some difficulties such as the fact that their performance, in term of convergence and accuracy, depends on user defined parameters.Contact nodes. Target surface. More detail and a finer mesh generally lead to a more accurate solution, but require more time and system resources. Nonlinear analyses add an extra factor, the number of load increments, which affect both accuracy and expense. Other nonlinear parameters, such as contact stiffness, can also affect both accuracy and expense. One must use their own engineering judgment to determine how much accuracy is needed versus how much expense can be afforded. Here these are neglected and only static structural analysis is done

VI. RESULT AND DISCUSSION

a. FEA Nodal Solution:

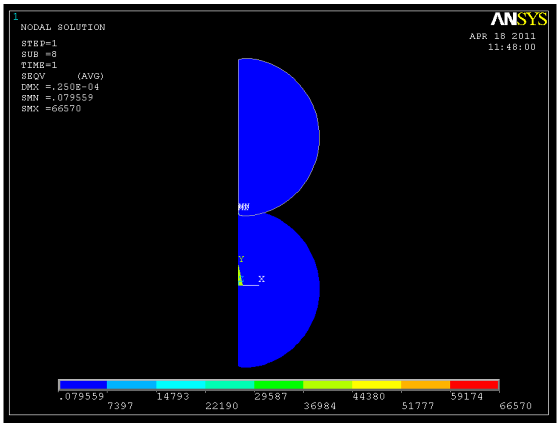

Fig 5.Nodal solution

The nodal solution averages the stress at element boundaries to display smooth contours. Fig 5 shows the stress contours at the contact zone. The elemental solution displays exact values of stress. The maximum contact stress occurred at the midline, whichwas 66570 psi.

b. FEA Elemental Solution:

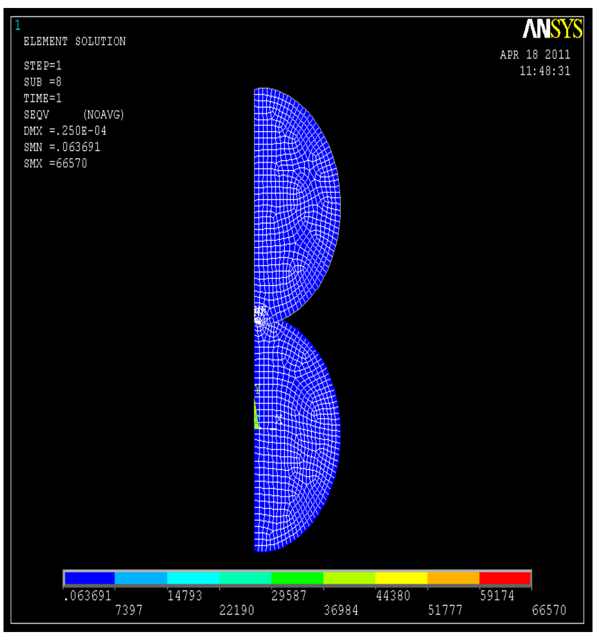

Fig 6. Elemental solution.

Comparison of results:

$$\%\,Error = \frac{\text{Contact stress}_{(Ansys)} - \text{Contact stress}_{(Hertz's)}}{\text{Contact stress}_{(Ansys)}}$$

$$= \frac{66570 - 64466.35}{66570}$$

$$= 3.160\%$$

The % error is small and within limit.

VII. CONCLUSION

Following conclusions can be drawn from the above work; F.E.M. provides approximate solution which is very close to analytical solution.

1. Contact stress between cylinders obtained by analytical method is 64466.35 Psi.
2. Contact stress between cylinders obtained by ANSYS is 66570 Psi
3. Percentage error 3.160 which is within acceptable limits.
4. Plane 42, contact 175 and target 169 element give good results for above contact problems.

REFERENCES

[1] “Variation Analysis of Involutes Spine Tooth Contact”;”Brian J.K. De Cares Brigham Young University”.

[2] “Stresses and Deformations in Involute Spur Gear by FEM “.”Zapping wei; University of Saskatchewan Saskatoon; Saskatchewan”.

[3] ”Cad Cam Theory and practice “;” Ibrahim Zaid”;”Tata McGraw Hill Edition; Seventeenth Reprint 2005”.

[4] ”An Introduction to finite element method “;”J.N.Reddy “;”Tata Mac Graw hill Edition eleventh reprint 2005”.

[5] ”Introduction to finite element in engineering”;”TirupathiR.Chandrapatala.Ashok D. Belegundu”;”Third Edition PHI New Delhi 2007”.

[6] ”Shubhankar Das Bakshi”;”Rolling contact Stress between two rigid axial flat cylinders”;”University of Cambridge; U.K”.

[7] ”R.S. DwerJoyce”;”Tribology Design Data”;”University of sheffielld; “First edition June 1997”.

[8] ”RaghvendraVijaywargiya”;”ItzahakGreen”;”A finite element study of thedeformations;forces;stress formation of energy losses in sliding cylindrical contacts”;”International Journals of nonlinear mechanics 42(2007)914-927”.

[9] ”Eugene I.Radzimovsky”;”Stress Distribution and strength condition of two rolling cylinders pressed

together";"university of Illinois Urbana; volume 50;feb-1953".

[10] "John a Williams and rob.s. Dwyer Joyee";"Modern Tribology Handbook".

[11] "Philippe de abreu Duque ;Dr-Ing Mauro MoraesdeSouza";"Jalionosavoy;carlosceelho";"Contact Pressure Between Cam and Roller Followers through FEM in Assembled cam shafts";"NeumayerTekfor Tech centre Brazil".

[12] "Ligiia cristina Brezeana";"contact stress between two cylindrical bodies with parallel axes analysis by FEM";"University of tiragumares; Romania".

[13] "Ali Rad Hassan ";"Contact stress analysis of spur gear teeth pair";"World academy of Science Engineering and Technology;volume 3;2009-10-20".