Validation of Hertzian Theory of a Cylinder Contact with a Plane Surface Using Finite Element Analysis

Thanin Chanmalai[1,*] Byungik Chang[2] and Jirawat Yuttapravest[3]

[1,3] *Civil Engineering Department, Chulachomklao Royal Military Academy, THAILAND*
[2] *Department of Civil and Environmental Engineering, University of New Haven, West Haven, Connecticut, 06516, USA.*
**Corresponding author; E-mail address: thanin.ch@crma.ac.th*

Abstract

In general, contact stress is generated when a force is applied between solids in contact. Contact stress addressed sufficiently severe defects that occur within the construction design, and the end material may fail to qualify. Hertz's theory illustrates the contact stress issues at the contact regions between two elastic solids in frictionless contact. As a result, analyze the structural behavior, it requires Hertz contact stress to calculate the contact pressure between two elastic solids. In addition, the issues including complicated structural systems need to rely on numerical solutions to create an accurate solution for contact stress between two elastic solids. Therefore, the finite element method (FEM) is a numerical solution to achieve many complicated problems relevant to the contact mechanic.

The primary objective of this research is to achieve the validation between Hertz contract stresses of a cylinder contact with a flat surface by using FEM.

Keywords: Contact Stress, Hertz's theory, Hertz Contact Stress, Finite Element Method (FEM)

1. Introduction

Hertzian contact theory is one of the methods being accepted for contact mechanics. The contact area between two solid surfaces, there are significant forces that may occur between the single line contacts of a ductile material. For instance, significant stresses would be generated in the contact area between the steel cylinder and the flat steel surface.

However, the solutions of Hertzian contact stress profiles are not regularly led to the proper analytical solution because of the complicated geometries, forces, and material properties. Therefore, the issues, including complicated structural systems rely on computational solutions such as the finite element analysis by ANSYS®. This method can be used to achieve solutions to many complicated problems, including geometric and material nonlinearities.

The contact stress between a cylinder and plane surface using finite element analysis depends on the modeling, loading, and property of materials. Typically, the complicated structural systems are required numerical analysis. The finite element technique can be applied to obtain solutions to many complex issues, including geometric and material nonlinearities. Heinrich Hertz presented the concept of contact mechanics in 1896. [1] Hertz contact stress introduces to the pressure caused on two cylindrical rollers in contact under the applied loading. The contact pressure is significant, as a single line contact arise between the rollers. As the loading flow lines will be intersecting at the contact region stress concentration takes place and high stress generated between two solid surfaces. In this research, Hertzian contact stress obtained by computational simulation between the steel cylinder and plane surface is validated by using FEM.

The primary objective about the research is to achieve the validation between Hertz contract stresses of a cylinder contact with a flat surface by using finite element analysis.

2. Methodology

2.1 Hertz contact stress theory

The analysis of the contact process between two elastic materials has been solved with the method of Hertzian contact stress [1]. The researchers have designed a reasonable approach based on Hertz's theory to explain the contact pressure between steel materials to determine the Hertzian contact process from contact dimension and penetration from contacting geometries [2]. In addition, many authors explained the solution of the contact zone by examining the proportion of the elliptic axis. Tanaka [3] designed an accurate method to calculate elliptical Hertzian contact pressure in which the elliptic integral is not required for calculation. Moreover, Antoine et al. [4] illustrated the replacing of elliptical integral with the polynomial system has taken a considered of the Hertzian contact theory. Also, Norden, N.B. [5] provides many Hertz equations regarding the contact stress have been developed to calculate the maximum compressive stress between the steel cylinder and steel plane.

2.2 Assumptions of Hertz theory

The research attempts to illustrate the cylindrical steel contact with the plane surface to obtain stress values caused by the contact point. Accordingly, the contact stress can be generated by the load parameter and the cylinder size. The validation of the finite element analysis result by ANSYS® can be compared with the results obtained using the Hertzian theory equation. The Hertzian contact process made the following accepting [4].

- At the contact point, the contacting surfaces can be represented by a similar quadratic polynomial between two geometries.
- The contacting surfaces between two geometries are frictionless.
- Stress and strain of homogeneous material occur in the elastic limit.
- Load is applied perpendicular in the surface.
- The contact stress reduces when far away from the contact point.

2.3 Result obtained in Hertz theory

The result obtained in Hertzian contact [10] is a rectangle shape with the presence of a contact region between two structure steel can be calculated by equation (1) and (2).

The area of contact stress is a rectangle of width (2b), and the length of the cylinder (l). The contact patch width (b) between steel cylinder and the plane surface can be found as the following equation (1).

$$b = \sqrt{\frac{2F\left(\left(1-v_1^2\right)/E_1+\left(1-v_2^2\right)/E_2\right)}{\pi l\left(\frac{1}{d_1}+\frac{1}{d_2}\right)}} \quad (1)$$

Where;

b is rectangular contact area width
F is Force
ν is Poisson's ratio
E is Elastic modulus
l is Length of contact
d is diameter of object

The maximum contact pressure (p_{max}) located at the center of the contact point, shown in equation (2).

$$p_{max} = \frac{2F}{\pi b l} \quad (2)$$

2.4 Finite element method for contact problems

Contact problems are complicated to solve without using a numerical solution, such as the finite element method. Chang, B [6] created a recommendation based on the Iowa River Bridge (IRB) Girder C was developed with a mapped mesh using the girder ratio - equal to six. The simulation was performed using the girder-end, force-based approach. Similarly, the computational beam examination was examined by ANSYS®. Also, the boundary conditions such as bending moment and shear force were applied to the geometry.

In addition, Wriggers, P [7] approached the results of finite element methods for the computational contact stress analysis with friction. In this research, an overview of the simulation of frictional modeling is performed when common equations are formulas for contact between elastic solids. Also, the paper provides the underlying technical derivations.

2.5 Mesh generation for finite element analysis

Shiao, C.M., and Chamis, C. C. [8] illustrates the mapping method was created to improve the accuracy and efficiency results. So, the mapped mesh can be generated to the complex geometries in finite element analysis.

The meshing technique is an elemental component of the computational analysis for the engineering simulation method. The proper meshing of the geometry is very important to consider due to mesh styles control the accuracy, efficiency, and time of the results. In ANSYS®, mesh styles can be divided to be tetrahedral shapes meshing and mapped mesh. The tetrahedral shapes meshing does not have constraints in terms of component styles. Also, tetrahedral shapes meshing does not have a specified pattern generated to the modeling of the structure.

In the mapped mesh style, hexahedral is proper to generate for the geometry. So, the geometry has elements less than tetrahedral shapes meshing. The volume of mapped mesh includes only hexahedron elements and typically has general features.

The tetrahedral shapes meshing also requires more elements than a mapped mesh. However, the mapped mesh has a limitation in terms of the element shape. The mapped mesh includes the pattern of the meshing and technically takes more effort to generate accurate results. In this research, the mapped mesh is appropriate for the modeling simulation that can be seen in Fig 1. Generating a suitable mesh for the geometry is significant. The geometry has too fine mesh results in excessive data processing time and memory space, while the creating of too coarse a mesh might result in inaccurate results.

In this research a fine mesh is typically required in the area where higher stress-strain may be generated. Furthermore, in the less significant part of the geometry are meshed with a coarser mesh. Also, sub-modeling may recognize methods to fix the area to mesh with finer mesh.

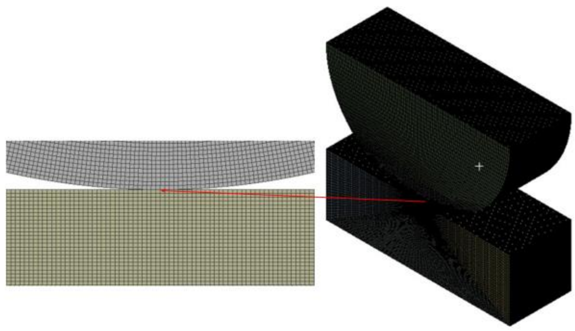


Fig. 1 Mapped mesh for finite element analysis

2.6 Sub modeling

The results from consider modeling can show proper research between the contact point that the load is applied in the boundary due to the finer mesh in computational analysis. The primary step is analyzing the entire geometry with coarse mesh. After that creating a sub-modeling is finer meshed of only the model of interest to get an accurate result. Sub-modeling is a technique used to obtain more accurate results in a specific part of the geometry. This technique is also specified as the cut-boundary displacement process or the specified-boundary displacement method. Sub-modeling represents a cut specified boundary through the coarse meshing model. The sub-modeling technique is based on St. Venant's principle. [9] This method illustrates that the distribution of stress and strain is changed only near the areas of pressure applying. St. Venant's principle [9] represents the sub-modeling technique states that a statically equivalent system replaces an actual distribution of forces. The distribution of contact stress is the most significant at the regions of load application between contact points. Therefore, the boundaries of the sub geometry are far enough away from the contact point then the accurate solutions can be obtained by the sub-model technique.

Sub-modeling is a finite element technique used to obtain accurate results in a specific part that can be obtained from sub modeling. Also, the contact performance between the two modeling can be employed in the simulation. The first step is to simulation the entire geometry with greater mesh refinement. The second step is creating a specific part with a finely meshed model analyzing the model by using the sub-modeling technique as shown in Fig.2.

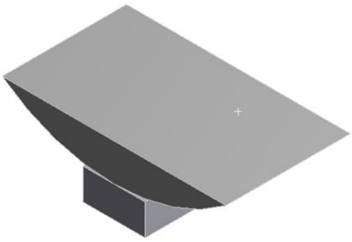


Fig. 2 Sub-modeling geometry in ANSYS®

3. Computational results using ANSYS®

The contact mechanic is generated under applied force (F) at the top surface of the cylinder and applied fixed support at the bottom surface of the plane surface as shown in Figure 3. Contact stress at the contact point depends on the load and type of non-friction contact that applied for finite element analysis by ANSYS®.

The small scale of the geometries has been created to analysis the contact stress between the cylinder and plane surfaces to minimize the simulation-processing time. Also, the most proper modeling may be determined by the small geometry to reduce the limitation during the simulation and get accurate results from ANSYS.

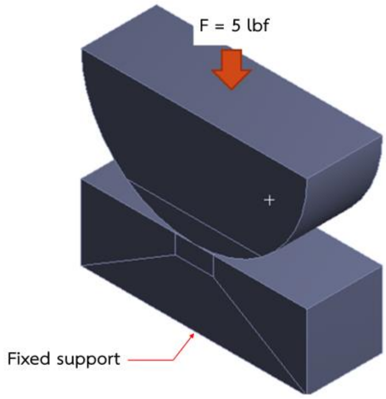


Fig. 3 Force and fixed support were applied in ANSYS®

The data assumed for validation of Hertzian theory of a cylinder contact with a plane surface using finite element analysis was followed in Table 1.

Table 1 Data assumed for Hertz contact stress.

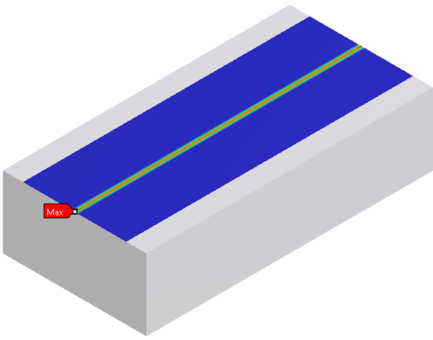
Description	Variable	Value	Unit
Force	F	5	lbf
Length of contact	l	0.1	in
Diameter of object 1 (cylinder)	d_1	0.3	in
Diameter of object 2 (plane)	d_2	∞	in
Elastic modulus object 1 (cylinder)	E_1	2.9E+07	psi
Elastic modulus object 2 (plane)	E_2	2.9E+07	psi
Poisson's ratio object 1 (cylinder)	ν_1	0.3	
Poisson's ratio object 2 (plane)	ν_2	0.3	

The percentage difference of Hertz contact stress between finite element analysis and Hertz theory between contact and target surfaces is about 0.33 % and 0.52 % shown in Table 2, respectively. The accurate result from the simulation can be obtained by meshing, and the type of contact areas are properly defined to the complex geometry. Also, the contact stress between cylinder and plane surfaces is required to ensure the stress-strain phases are elastic limits. In addition, the performance of the cylinder on the plane surface shown in simulation software is reasonable due to the percentage difference of the pressure between Hertz theory and numerical analysis from ANSYS is less than 1 percent.

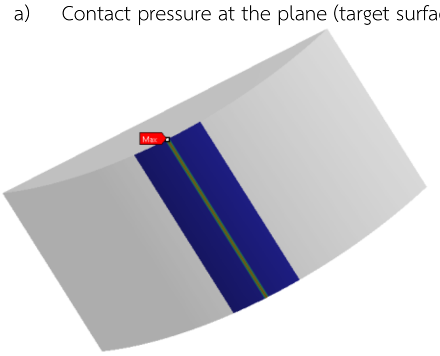
Table 2 Contact pressure comparison

Description	Hertz contact stress (psi)	Result from ANSYS®	Percentage difference
Maximum pressure at plane surface (target surface)	41,117.6	40,983	-0.33%
Maximum pressure at cylinder (contact surface)	41,117.6	40,904	-0.52%

In the numerical analysis from ANSYS® illustrates the maximum contact pressure at the plane (target surface) is 40,983 psi, as shown in the red region in Fig. 4 a). Also, the maximum contact pressure at the cylinder (contact surface) is 40,904 psi, as shown in Fig. 4 b). In the Hertz theory, the result obtained by the maximum Hertzian contact pressure is 41,117.6 psi that obtained from equation (2).



a) Contact pressure at the plane (target surface)



b) Contact pressure at the cylinder (contact surface)

Fig. 4 Contact pressure result from ANSYS®

4. Conclusions

The validation analytical result between Hertz theory and finite element method by ANSYS® is acceptable. The value of contact pressure is significant for the contact mechanic between two steel surfaces, as the stress value effect in the contact point. The accurate result from the simulation can be obtained by meshing, and the type of contact areas are properly defined to the complex geometry. Also, the contact pressure between cylinder and plane surfaces is required to ensure the stress-strain phases are elastic limits. In addition, the performance of the cylinder on the plane surface shown in simulation software is reasonable due to the percentage difference of contact pressure between Hertz theory and numerical analysis from ANSYS® is less than 1 percent.

Hertzian theory presented by Hertz can represent the basis for many contact problems in engineering. Hertzian contact proposes between two elastic solids that are frictionless contact. Also, the proportion of the contact area is small when compared to the dimensions of the model. During the contact stage, two solid surfaces are touched together; there will distort between each other. The deformation of the material may be an elastic or plastic phase. Some deformation between the structures occurs on a small scale. Moreover, the two curved modeling of various radii of the curve are made into contact they initially touch at a contact point or along a contact line.

Acknowledgement

The author is thankful to the Department of Civil and Environmental Engineering, University of New Haven, Chulachomklao Royal Military Academy who have provided the most cooperative assistance to this research work.

References

[1] Hertz, H. (1896). *On the Contact of Two Elastic Solids*. Macmillan & Co., pp. 146–162.

[2] Fischer, F.D. and Wiest, M. (2008). Approximate analysis model for Hertzian elliptical wheel/rail or wheel/crossing contact problems. *ASME Journal of Tribology*, 130, pp.1–3.

[3] Tanaka, N. (2001). A new calculation method of Hertz elliptical contact pressure. *ASME Journal of Tribology*, 123, pp.887–889.

[4] Antoine, J.F., Visa, C., Sauvey, C. and Abha, G. (2006). Approximate analytical model for Hertzian elliptical contact problems. *ASME Journal of Tribology*, 128, pp.660–664.

[5] Norden, B. N. (1973). *On the Compression of a Cylinder Contact with a Plane Surface*. U.S. Department of Commerce National Bureau of Standards.

[6] Chang, B.K. (2004). *Recommendations for the Analysis of Contact Stresses for Launched Plate Girder*. Master's Thesis, Iowa State University, U.S.A.

[7] Wriggers, P. (1996). Finite element methods for contact problems with friction. *Tribology International*, 29, pp. 651-658.

[8] Shiao, M.C. and Chamis, C.C. (2004). Mapping methods for computationally efficient and accurate structural reliability. *In its Structural Integrity and Durability of Reusable Space Propulsion Systems*, pp.145-158.

[9] Lee, H.H. (2019). *Finite Element Simulations with ANSYS Workbench 2020*. SDC Publications, pp. 390-417.

[10] Budynas, R.G. and Nisbett, J. K. (2014). *Shigley's Mechanical Engineering design*. The McGraw-Hill Companies, Inc.