UDC 519.61:539.3
Professional paper
Received: 20.10.2000.

Numerical model of a Hertz contact between two elastic solids

Budimir Mijovi}(1) and **Mustafa D`oklo**(2)
(1)Faculty of Textile Technology, University of Zagreb, Prilaz baruna Filipovi}a 30, HR-10000 Zagreb, CROATIA, e-mail: mijovic@hotmail.com
(2)Faculty of Mechanical Engineering, University of Mostar, Maršala Tita bb, Mostar, BOSNIA AND HERZEGOVINA

SUMMARY

This paper presents the analysis of stresses and deformations occurring when surfaces of two elastic bodies come into contact. Finite element method has been used as a numeric technique in order to obtain contact stresses between two elastic bodies. The mapped meshing method has been used for model meshing. When two bodies come into contact they touch initially in one point or along the line. Under the influence of loading they get deformed in the vicinity of the point of the first contact so that they touch along the surface whose size is small in relation to the body dimension. The contact problems are usually nonlinear and require computers of higher capacity. In a contact between two elements it is difficult to determine the contact area. This depends on loading, material, boundary conditions etc. Both combinations, i.e. when friction is taken into account and when it is not taken into account on contact surface have been observed. Normal and gliding stresses on the contact surface which have been caused by contact have been presented by reaction forces on nodes on contact surfaces. According to sizes of reaction forces and friction coefficient, conditions on nodes of contact surfaces have been classified as gliding or sticking. Contact stresses have been calculated in the paper for cases when cylinders and rectangular plates for materials with different elasticity moduli come into contact. A symmetrical contact model has been used in the numeric analysis.

Key words: *Hertz contact, elastic solids, finite element method, mapped meshing method, friction.*

1. INTRODUCTION

In many engineering structures there are cases where two or more components are in physical contact. Many examples in structures have been analyzed where breakup occurs due to the influence of contact stresses. It is very important for machine designers to know tensions and deformations in the vicinity or on the contact surface. Therefore many investigators have analyzed contact problems. However, these results have been limited to cases of contacts of elastic-elastic and elastic-rigid bodies. It is necessary to be informed about the results of contact stress between elastic bodies in structures, except when elastic modules are extremely different, as is the case in combinations of metal-plastic. Contact stress is highly concentrated in a closed contact region and decreases quickly by intensity with distance from the point of contact. The stress can be calculated approximately by observing each body as a semi-area restricted with flat surfaces. This idealization, where bodies with arbitrary surfaces of profile have been observed as semi-spaces, has been analyzed by Timoshenko & Goodier [1]. Hertz (1882) [2] performed the first satisfactory analysis of stress for contact between two bodies. He formulated conditions which should be met for the normal displacement of body surfaces. A simplification that each body can be considered as a elastic semi-space loaded on a small elliptic part of the surface is presumed. With these simplifications, highly concentrated contact stresses have been analyzed separately from the general distribution of stresses on two bodies. The calculation of contact pressure in gliding contact has been done by Johnson & Jefferis [3] by using Tresca and von Mises flow criterion. Contact surfaces convey shear stress causing additional friction by normal stress. During the analysis of stress and strain in elastic semi-space it is presumed that surface stresses outside of confined region are equal to zero. In the majority of contact problems shifts or combinations of displacement and surface stresses occur within the contact region. Elastic bodies in contact with sufficiently small deformations in the theory of elasticity of small linear deformations will be observed in contact, whose contact surfaces are small in comparison with dimensions and radius of curving for non-deformed surfaces.

2. METHOD DESCRIPTION

The paper presents the numerical procedure for two-dimensional elastic contact problem. Francavila & Zienkiewicz [4] described a simple technique for the analysis of elastic contact problems. Sachdev & Ramakrishnan [5] improved the method to include boundary conditions of forces. In many practical situations it is necessary to include the friction effect so that the presence of the friction force affects the nature and distribution of strain in the contact zone.

Two elastic bodies *A* and *B* have been analyzed which come into contact as presented in Figure 1. Body *B* is tied, as a boundary condition, while *A* body is loaded by a vector of force $\vec{f}$. The movement of particles of *B* body can be expressed with a vector $\{\Delta\}$ as:

$$\{\Delta\} = \begin{Bmatrix} U \\ V \\ \theta \end{Bmatrix} \qquad (1)$$

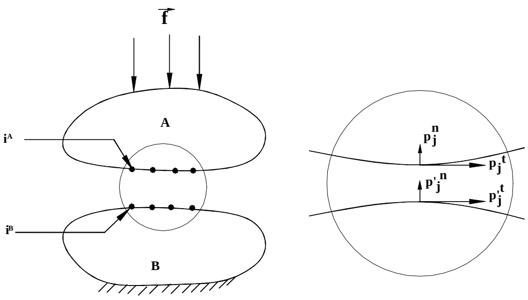

Fig. 1 Two elastic bodies in contact

A group of boundary points i^A and i^B *(i=1, 2, m)*, is taken, as presented in Figure 1, so that the points are where the contact is likely to occur. It will be assumed that the direction of the joint vertical and tangent on the boundaries of these points can be determined.

If each of these points of force components on a body *A* is $\vec{p}_j$ and $\vec{p}'_j$ on *B* body, then it can be written that vectors $\vec{p}_j$ and $\vec{p}'_j$ have two components as:

$$\vec{p} = \begin{Bmatrix} p_j^{\,t} \\ p_j^{\,n} \end{Bmatrix} \quad \text{and} \quad \vec{p}' = \begin{Bmatrix} \vec{p}_j^{\,t} \\ \vec{p}_j^{\,n} \end{Bmatrix} \qquad (2)$$

where p_j^t and p_j^n are force components in a node *j* in a tangential and normal straight line.

Let shifts of i^B be expressed with vector $(\vec{\delta}_i)_B$ as:

$$(\vec{\delta}_i)_B = \begin{Bmatrix} \delta_i^{\,t} \\ \delta_i^{\,n} \end{Bmatrix}_B \qquad (3)$$

where δ_i^t and δ_i^n are shift components in a node in a tangential and normal straight line.

Since *B* body is fastened and only action forces are contact forces $\vec{p}'_j$ *(j=1, 2, ..., m)* then displacement point of i^B is given with:

$$(\vec{\delta}_i)_B = \sum_{j=1}^{m} \left[C_{ij}^{\,B}\right] \vec{p}'_j \qquad (4)$$

where $[C^B_{ij}]$ (2x2) is submatrix of the corresponding shift coefficient in a tangential and normal straight line on node *i*, causing forces on node *j*. Matrix $[C^B_{ij}]$ is obtained by elimination of all nodes with the exception of those for which possible contact may occur. As a solution for *B* body, the motion of particles expressed by vector $\{\Delta\}$ and total shifts of points i^B is given as:

$$(\vec{\delta}_i)_B = \sum_{j=1}^{m} \left[C_{ij}^{\,B}\right] \vec{p}'_j + [\sigma_i]\{\Delta\} \qquad (5)$$

where $[\sigma_i]$ is transformation matrix, which is given as:

$$[\sigma_i] = \begin{bmatrix} \cos\varphi & \sin\varphi & (y\cos\varphi - x\sin\varphi) \\ -\sin\varphi & \cos\varphi & -(x\cos\varphi + y\sin\varphi) \end{bmatrix} \qquad (6)$$

The solution of Eq. (6) is determined if the contact zone is known and if friction is sufficiently high to prevent nodes sliding. In practice, the contact zone is not known in advance and some nodes may slide and others may not. Thus, in these cases the iterative method will esure control both for nodes which lose contact and for those at which sliding occurs. It is considered that in all such problems, the size of sliding is low in comparison with the mesh size. Then, the steps are repeated until all normal forces in the contact zone are positive and the ratio of tangent and normal forces for all nodes in the contact is lower or equal to μ. Once a normal force in the contact zone is known, the actual distribution of pressure and stress can be obtained correctly. The normal stresses along the contact surface correspond to the exact value of the constant pressure intensity.

3. ILLUSTRATIVE EXAMPLES

An elasto-elastic problem where cylinder comes into contact with a flat elastic plate has been analyzed in this paper. By using the computer program ANSYS, the contact problem has been solved by using 2-D structural solid elements PLANE42 and 2-D point-to-surface contact elements CONTAC48. The plane stress condition has been modeled by using a 10 mm thickness through the cylinder. The program uses contact elements in order to mark the relative position of the two surfaces. The contact elements are triangle, tetraheders or pyramids (in dependence on whether 2-D or 3-D), where the basis consists of nodes of other surfaces (target surface) and the peak is the node of the first surface (contact surface). It has to be identified where the contact may happen during the deformation of a certain model. Once, when the potential contact surfaces have been identified they are defined through contact elements. For an efficient solution of the problems (primarily for CPU) it is necessary to define smaller localized zones, but one has to be sure that the zones are adequately covered with all necessary contacts. In dependence on the model geometry (and potential deformations) several target surfaces may have contact with the same contact surface. In that case, more contact pairs should be defined. Under the condition that certain surfaces will never come into contact, those nodes can be omitted from the list of surface nodes, but usually more nodes are included than is really considered necessary. When types of contact elements have been defined then the phase of generating contact elements starts. This is to be done by command GCGEN. They have options for defining asymmetrical and symmetrical contact models. If an asymmetrical contact model is created, then one of the surfaces is a contact surface, a "contractor", and the other is "target" surface. Alternatively, two GCGEN commands can be used for the automatic generation of contact elements, in order to identify each surface simultaneously as a contact and a target surface. The symmetric contact model has certain advantages and, therefore, does not require special considerations when one surface is a contractor at one moment and a target at the other. On the contrary, the asymmetric contact model requires rules to be used which are valid for a certain surface topology.

The contact between a flat surface and a cylinder with normal loading has been presented in Figure 2. In the Coulomb basic model, two contact surfaces may convey shear stress before the beginning of the relative sliding. This condition is known as sticking. The Coulomb friction model defines an equivalent shear stress in which process sliding begins due to contact pressure p.

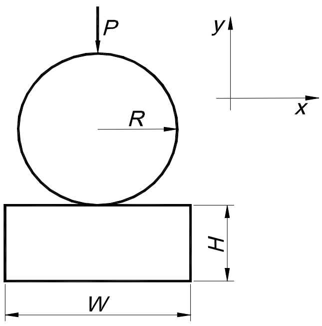

Fig. 2 Model of a cylinder and a flat elastic plate

In this model the steel-aluminium contact has been analyzed with the following parameters:

- normal force, $F=2.352\times10^6$ N,
- Poisson coefficient for aluminium, $\nu_1=0.22$,
- Poisson coefficient for steel, $\nu_2=0.29$,
- friction coefficient, $\mu=0.0$ and 0.005,
- unit loading, $P=F/L$.

Hertz formulation for elastic contact stress has been considered in the paper:

$$E^* = \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)^{-1} \qquad (7)$$

In this process the mean radius of cylinder and plane curving has been considered:

$$R = \left(1/R_1 + 1/R_2 \right)^{-1} \qquad (8)$$

The produced formula for the radius of the cylinder contact with the base is:

$$a = \left(\frac{4PR}{\pi E^*} \right)^{1/2} \qquad (9)$$

The maximum contact pressure between the cylinder and the plate can be calculated:

$$p_0 = \frac{2P}{\pi a} = \left(\frac{PE^*}{\pi R} \right)^{1/2} \qquad (10)$$

The maximum shear stress can be determined:

$$\tau_1 = 0.30\, p_0 \ \text{at} \ x=0,\ z=0.78a \qquad (11)$$

On the basis of previous formulas the following can be calculated:

$$E^* = \left(\frac{1-\nu_1^2}{E_1} + \frac{1-\nu_2^2}{E_2} \right)^{-1} = \left(\frac{1-0.22^2}{0.7\times10^8} + \frac{1-0.29^2}{2.06\times10^8} \right)^{-1} =$$

$$= 5.54\times10^7 \ Pa$$

$$a = \left(\frac{4PR}{\pi E^*} \right)^{1/2} = \left(\frac{4\times1000\,\frac{F}{L}}{\pi E^*} \right)^{1/2} = 73.52\ mm$$

$$p_0 = \frac{2P}{\pi a} = 2000\ Pa$$

where:

cylinder length, $L=10$ mm,
elasticity module for steel, $E_2=2.06\times10^8$ kPa,
modulus of elasticity for aluminium, $E_1=0.7\times10^8$ kPa,
radius of the plate, $R_2=\infty$,
cylinder radius, $R_1=1000$ mm,
plate dimensions, $W=600$ mm and $H=300$ mm.

When estimating the model it is necessary to determine the size of the contact rigidity *KN*. There is no standard value for *KN* but the value should be sufficiently high to cause sufficient rigidity of a model, i.e. to cause neither too high nor too small penetration. For the highest number of contact analyses, *KN* can be determined according to the following equation:

$$KN \approx f\,e\,h \qquad (12)$$

where:

f [-] - factor including the contact compatibility; this factor is taken from *0.01* to *100*; for the initial value $f=1$ is used,
E [N/mm^2] - Young modulus of material elasticity (if in a model there is contact between two various materials, the lower value of the module is taken),
h [*mm*] - characteristic contact length; value which is used depends on the problem geometry; for the plane strain condition, h is the thickness of the finite element).

The size of the elastic zone depends on the value, which is used for sticking rigidity (*KT*). As for the normal contact rigidity *KN*, it is ideal to use a higher sticking rigidity, but not so high as to disturb real convergence. It is taken that sticking rigidity *KT* should be 1, 2 or 3 times lower than the normal rigidity *KN*. If the friction model is modeled, the program will take the value for *KT*. The value for the sticking rigidity *KT* (*KT=KN/100*) was used in the program. However, as in case for *KN*, if *KT* has higher values it should be evaluated. This value for *KT* cannot be suitable for all situations. In the model, the area of plastification cannot be calculated since in that case it is necessary to define the curve of hardening.

The geometric model of the cylinder and plate prepared for simulation has been presented in Figure 3.

Figure 4 presents the meshed model for estimation, by applying the finite element method with boundary conditions, and the distribution of force and contact elements at the place of contact. The elastic cylinder and plate have been meshed by using the "mapped meshing" method.

4. NUMERICAL RESULTS

Figure 5 presents the distribution of normal stress in the direction of axis *y*, where it is obvious that the stress changes considerably during the process of contact. The minimum normal stress at the site of contact in direction of axis *y* is -1984.0 MPa, and the maximum 66.409 MPa.

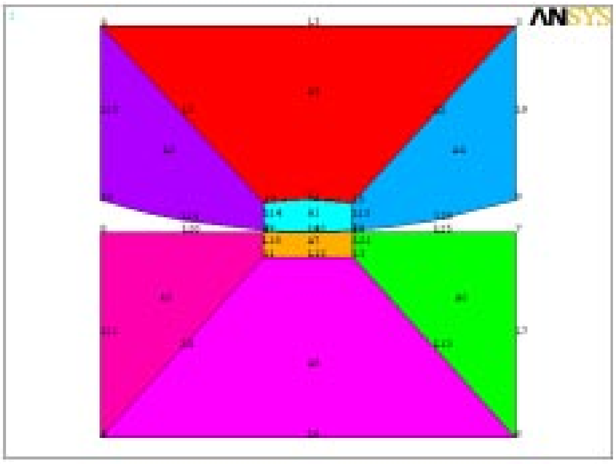

Fig. 3 Model of a cylinder and a plate prepared for simulation

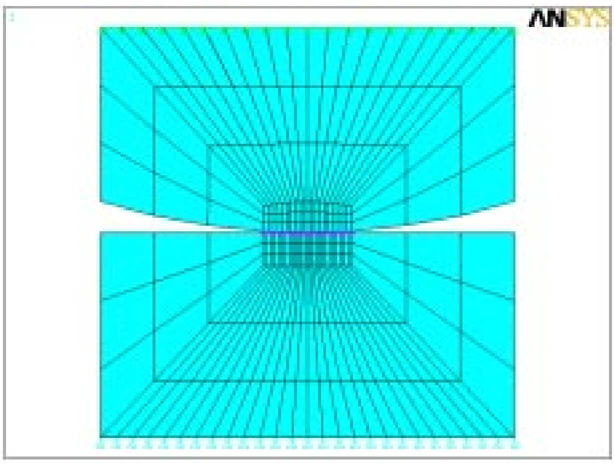

Fig. 4 Finite elements of the cylinder and plate

Figure 6 presents a significant change of normal stress on the contact site in direction of axis *x* from -1671.0 MPa do 256.015 MPa. From Figure 6 it is evident that the maximum normal stresses appears in the center of the contact surface and they get lower towards the end of the cylinder and plate.

Figure 7 presents the distribution of shear stress in plane *x-y* from -446.984 MPa up to 447.944 MPa. From Figure 7 it is evident that the size of the normal stress is decreased in the direction of axis *x* from the center of the contact surface towards the edges.

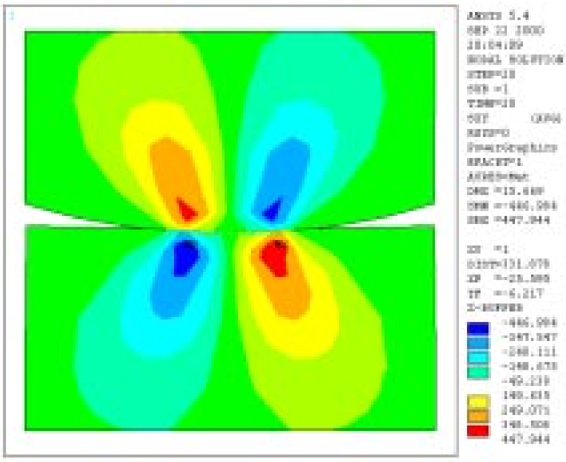

Fig. 5 Distribution of normal stress in direction of axis y

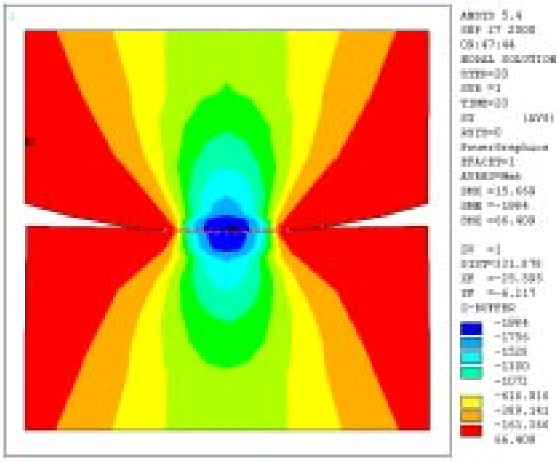

Fig. 6 Distribution of normal stress in direction of axis x

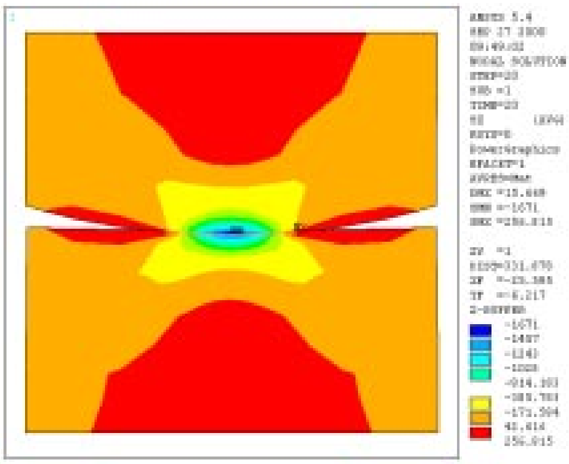

Fig. 7 Distribution of shear stress in plane x-y

According to Figure 8 the displacements of cylinder and plate in direction of axis *y* are highest in the center of the contact surface and they get lower towards the edges. At the sites of the maximum displacement the highest penetration occurs between the cylinder and the plate.

According to Figure 9 the displacements of the cylinder and plate in direction of axis *x* are the highest at the contact line. The displacements get smaller at the sites of greater distance from the contact line.

Figure 10 presents the distribution of the normal stress in the direction of axis *y*, as a function from distance *x* of the contact point. The maximum value of the normal stress 1981.205 MPa is at a distance *x*=5.4646 mm.

Figure 11 presents the distribution of shear stress as a function from distance *x* of the contact point. It is obvious from Figure 11 that the minimum values of the shear stress occur at a distance of 28.44 mm and the maximum values of shear stress at distance of 156.453 mm are visible.

Figure 12 presents the distribution of the normal stress in a straight line of axis *x* as a function from distance *x* of the contact point. It is obvious from Figure 12 that maximum normal stress occurs at distance of 9.1650 mm.

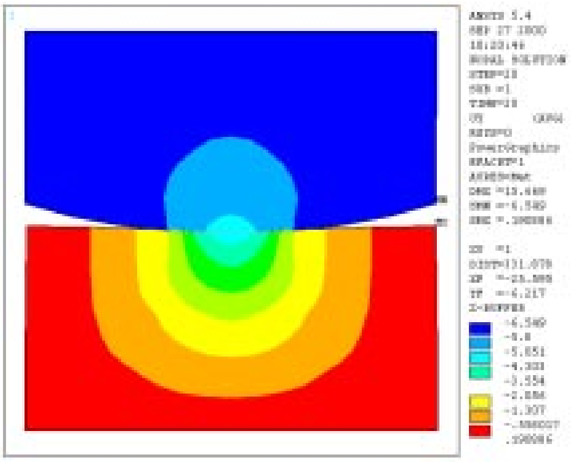

Fig. 8 Displacements of the cylinder and plate in direction of axis y

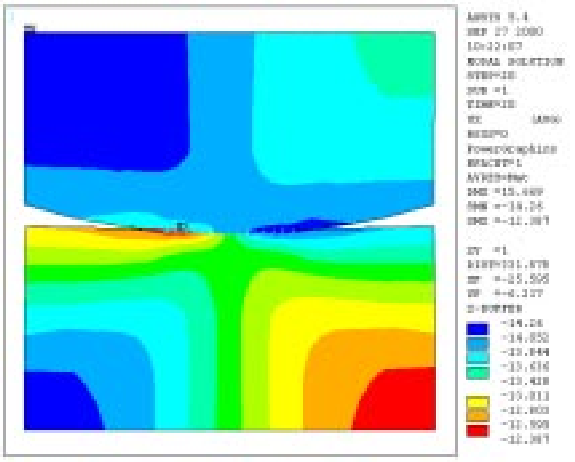

Fig. 9 Displacements of the cylinder and plate in direction of axis x

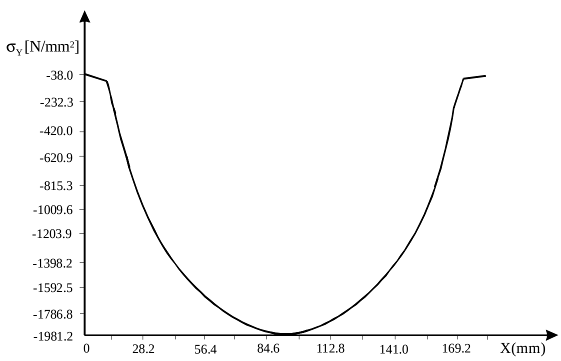

Fig. 10 Distribution of the normal stress σ_y as a function from distance x

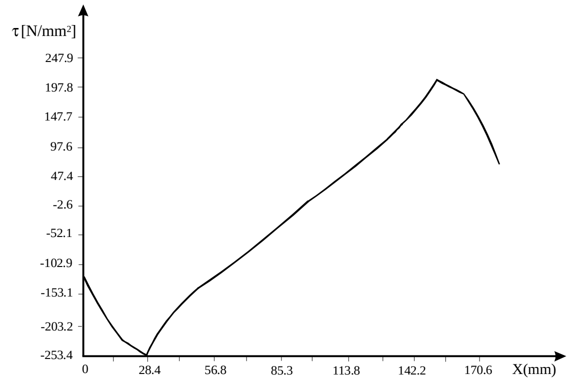

Fig. 11 Distribution of shear stress τ as a function from distance x

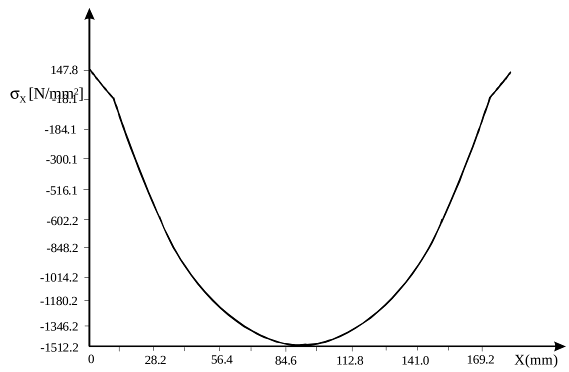

Fig. 12 Distribution of normal stress σ_x as a function from distance x

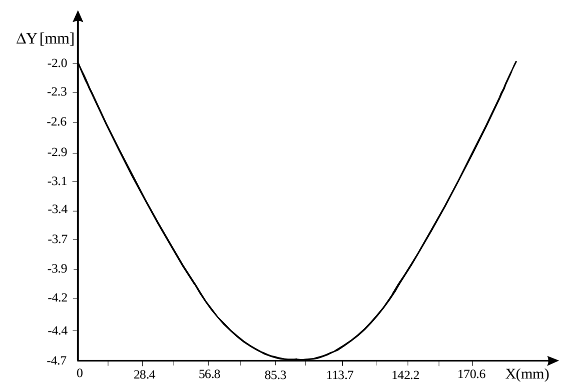

Fig. 13 Displacements in a straight line y as a function from distance x

Figure 13 presents the displacement in a straight line *y* as a function from distance *x* of the contact point. The maximum displacement in a straight line of axis *y* is 4.750 at a distance of 92.0 mm.

Figure 14 presents the displacement in a straight line *x* as a function from distance *x* of the contact point. From Figure 14 it is obvious that displacement in a straight line of axis *x* is the largest in the center and it decreases by increasing the distance from the center.

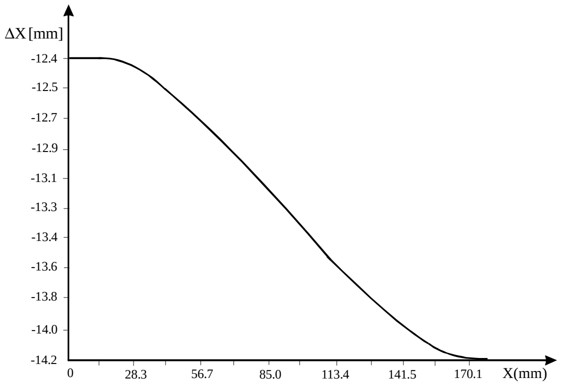

Fig. 14 Displacements in a straight line x as a function from distance x

5. CONCLUSION

A numerical solution of the cylinder contact on an elastic rectangular plate placed under linear load was obtained for a certain ratio of elastic constants and friction coefficient. The plane stress conditions have been applied. From Figures 5, 6, 10 and 12 it can be seen that the maximum normal stress always occurs in the center of the contact surface and it decreases unvariedly towards zero at the edges; hence, for various elastic constants it results in various contact surfaces. The friction effect has also been investigated for this configuration. The sliding and sticking behaviour has also been investigated for various elastic constants, for two bodies. This problem is known as non-conform problem because non-deformed contact surfaces have various profiles which represent complicated types of contact problems [6, 7, 8]. Non-conform problem depends on the loading and becomes highly non-linear. Problems of this category cannot be solved directly; they can only be solved approximately using successively small loading steps. The linear dependence was observed within each loading step. The cylinder and the plate were modeled by using PLANE42 finite elements. Linear loading was applied along the upper cylinder edge. Bodies have various elastic properties (steel-aluminium) and two techniques of loading have been used. In one case a technique of successive small loadings has been used (time-dependent-loading), and in another case loading in one step has been used. Comparing the results for μ=0.0 of these two techniques it is possible to note clearly that a change in the contact status of nodes occurs. Stated in other words, the nodes which were in the phase of gliding during the use of the first technique of loading have in large numbers moved into the sticking condition. This naturally occurs due to the loading mode, as if conditional high-impact loading has been used. It is also possible to notice that, when using the first technique, the sliding zone does not develop in the center of the zone of contact between two bodies, but the tendency is sliding development from the center towards the edges of the contact surfaces. For bodies with different elastic properties (steel-aluminium), the cylinder and the plate have been modeled also by using the PLANE42 finite elements. It is possible to explain that if the same density is used then a too high rigidity of finite elements occurs, and the nature of the problem is that one body is considerably softer than the other. As with the previously discussed case for μ=0.0, there are some nodes which are in the sliding status and several in the sticking status when using the first technique while in the case of applying the other mode of loading a large number of nodes are in the sticking status. Also, the occurrence of increased penetration of nodes is obvious, when applying the other loading technique. By introducing friction, in both techniques, no development of gliding occurs but a development of the node mass occurs which is in the sticking status [9, 10, 11]. Only slightly increased depth of penetration is visible by application of other loading technique. The diagrams of strain condition as well as the radius of contact obtained by a numerical method show a good accordance with the results obtained theoretically.

REFERENCES

[1] S. Timoshenko and J.N. Goodier, *Theory of Elasticity*, 3rd Edn., New York, McGraw-Hill, 1951.

[2] H. Hertz, Ueber die Berührung fester elastischer Körper, *J. reine und angewandte Mathematik*, Vol. 92, pp. 156-171, 1882.

[3] K.L. Johnson and J.A. Jefferis, Plastic flow and residual stresses in rolling and sliding contact, Proc. Institution of Mechanical Engineers Symposium on Rolling Contact Fatigue, London, p. 50, 1963.

[4] A. Francavilla and O.C. Zienkiewicz, A note on numerical computation of elastic contact problems, *Int. J. Num. Math. Engng.*, Vol. 9, pp. 913-924, 1975.

[5] T.D. Sachdeva and C.V. Ramakrishnan, A finite element solution for the two-dimensional elastic contact problems with friction, *Int. J. Num. Math. Engng.*, Vol. 17, No. 17, pp. 1257-1271, 1981.

[6] S. Ohte, Finite element analysis of elastic contact problems, *Bull. JSME*, Vol. 18, pp. 797-804, 1973.

[7] D. Nowell and D.A. Hills, Hertzian contact of ground surfaces, *J. of Tribology*, Vol. 111, pp. 175-179, 1989.

[8] A. Sackfield and D.A. Hills, Some useful results in the classical Hertz contact problem, *J. of Strain Analysis*, Vol. 18, No. 2, pp. 101-105, 1983.

[9] R. Gaertner, Investigation of plane elastic contact allowing for friction, *Computer & Structures*, Vol. 7, pp. 59-63, 1977.

[10] A. Sackfield and D.A. Hills, A note on the Hertz contact problem: A corection of standard formule, *J. of Strain Analysis*, Vol. 13, No. 3, pp. 195-197, 1983.

[11] K.L. Johnson, *Contact Mechanics*, Cambridge University Press, 1985.

[12] T. Tscita and S. Ymayi, *Finite Element Analysis of Elastic Contact Problems*, University of Tokyo Press, 1973.

[13] J.T. Stadter and R.O. Weiss, Analysis of contact through finite element gaps, *Computer & Structures*, Vol. 10, pp. 867-873, 1979.

[14] S.K. Chan and I.S. Tuba, A finite element method for contact problems of solid bodies - Part I. Theory and validation, *Int. J. Mech. Sci.*, Vol. 13, pp. 615-625, 1971.

NUMERI^KI MODEL HERTZOVOG KONTAKTA IZME\U DVAJU ELASTI^NIH TIJELA

SA@ETAK

U radu su analizirana naprezanja i deformacije, koja se javljaju kada površine dvaju elasti~nih tijela dolaze u kontakt. Metoda kona~nih elemenata korištena je kao numeri~ka tehnika, da bi se dobila kontaktna naprezanja izme|u dvaju elasti~nih tijela. Za diskretizaciju modela korištena je "mapped meshing" metoda. Kada dva tijela dolaze u kontakt, ona se dodiruju inicijalno u jednoj to~ki ili du` linije. Pod djelovanjem optere}enja, ona se deformiraju u blizini to~ke prvog kontakta, pri ~emu se ona dodiruju po površini ~ija je veli~ina mala u odnosu na dimenzije tijela. Kontakni problemi su naj~eš}e nelinearni i zahtijevaju ra~unala ve}ih kapaciteta. Kod kontakta dva elementa, teško je odrediti podru~je kontakta. To ovisi o optere}enju, materijalu, grani~nim uvjetima, itd. Tako|er su u radu promatrane kombinacije kada se uzima, i ne uzima, u obzir utjecaj trenja na kontaktnoj površini. Normalna i posmi~na naprezanja na kontaktnoj površini, koja su prouzrokovana kontaktom, predstavljena su silama reakcija na ~vorovima na kontaktnim površinama. U usporedbi s veli~inom sila reakcije i koeficijenata trenja, stanje na ~vorovima kontaktnih površina klasificirano je kao klizanje ili prijanjanje. Kontakna naprezanja ra~unala su se za slu~ajeve kada u kontakt dolaze cilindar i pravokutna plo~a za materijale s razli~itim modulima elasti~nosti. U numeri~koj analizi korišten je simetri~ni kontaktni model.

***Klju~ne rije~i**: Hertzov kontakt, elasti~na tijela, metoda kona~nih elemenata, "mapped meshing" metoda, trenje.*