CONTRIBUTIONS REGARDING THE DISPLACEMENT IN A HERTZIAN CONTACT

Stefan Ghimisi
University "Constantin Brancusi" of Targu Jiu, ghimisi@utgjiu.ro

Keywords: finite element method, displacement, hertzian contact

Abstract: Analysis by finite element method is a modern method for studying various contacts, allowing the determination of important parameters for the study of various contacts. In case of hertzian contact are obtained precise informations regarding the state of deformation at the contact level , it can be obtained information about the state of the existing tension at the contact level. We consider the sphere –plan contacts which represent the materialization of the point contacts from “Experimental Standfor the fretting study” which is in the Machine Elements Laboratory

1.Introduction

Finite element method (MEF) can be explained in two intertwined ways: mathematical and physical.

Basic concept in the physical interpretation is the partition (disassembly, partitioning, separation, division) of a mechanical complex in more simple dissociated components called finite elements. Mechanical response of an element is characterized by a finite number of degrees of freedom, represented like unknown function values as a set of nodal points. The response of the element is defined by algebric equations composed by physical arguments.

Original system's response is approximated by the response of discrete model constructed by connecting or assembling all the elements.

If a system is complex, its solution is by successive divisions, resulting subsystems which can be divided so that the splitting process will be continue until each subsystem behavior became simple. These "primitive pieces" called elements will interact with each other resulting the behavior of the system.

In the mathematical interpretation, finite element method is a procedure for obtaining the numerical approximations at the solution of the limit value set for a domain Ω, replace by the union U of separated sub domains called finite elements. Generally, his geometry is approximated only by the U.

Considering $U'(\xi)$ - functions which represent the displacements, deformations,tensions...,fonctins with U(x,y,z),coordinate,defined in the domain pointsin which is defined the structure. If the exact solution $U'(\xi)$ is unknown and $U(\xi)$ is an approximate of these function ,the error function $e(\xi)$ is (Fig.1):

$$e(\xi)=U(\xi)-U'(\xi) \tag{1}$$

To construct an approximate solution is sufficient to write an expression containing n - approximation parameters:

$$U(\xi)=U(\xi,a_1,a_2,a_3,...,a_n) \tag{2}$$

which can be determined based on the relationship (1) and a suitable convergence criterion.

In the finite element method the approximation is nodale and the form is:

$$U(\xi)=[N_1(\xi),N_2(\xi),...,N_3(\xi)]\cdot\begin{Bmatrix}u_1\\u_2\\...\\u_n\end{Bmatrix}=[N]^T(u_n) \tag{3}$$

where: u_i is the nodal parameters of approximation and they have a concrete physical meaning, a_i generalized parameters, $N_i(x)$ are interpolation functions, which are usually polynomial functions. Given the mathematical interpretation for the Ω domain,the approximate solutions u(x) can be constructed for un full size domain or on elementar subdomains $\Omega^{(e)}$.

2. Stages of the finite element method analysis

Using specialized software for finite element analysis (Cosmos, Ansys, Nice, Pro / driver, etc..) can proceed to study the structures comprising the steps: preprocessing and postprocessing.

Preprocessing - initial stage for the structure where we can define
1 - the system of units
2 - reference system;
3 - the geometry of the structure;
4 – the material that will make the structure;
5 - item type used to structure the digitization
6 - the analysis will be performed;
7 - structural loads;
8 - the contour conditions;
9 - softwer type used.

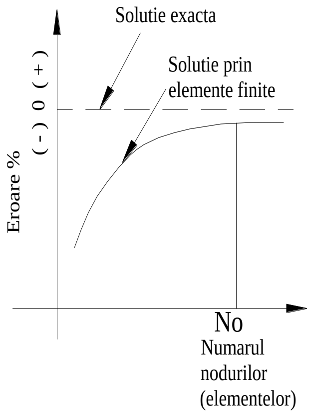

Figure1 Digitization error based on the number of elements

For realising the geometry of the system the was used Pro / ENGINEER program, version 2000i and the analyse by finite element method was done using Pro / Mechanica version 21 - which working in an integrated way with Pro / ENGINEER, thus reducing transfer errors.

For digitization of the structure are two methods available: the "p-element and method h-element ". Method "h-element-method is the " traditional "in which the structure is digitized with triangular elements - (2D) figure 2. and tetraedale (3D) fig.3. with the disadvantage that in digitization of the circular occurs an digitization error (fig. 4).

This error occurs neutral of the type of finite element used (of the order 1,2, or 3) even if on the circular area is made arefinement of the mesh.

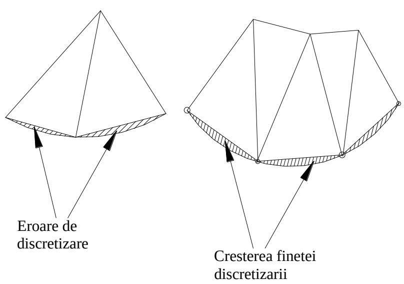

Figure.2.Triangular elements

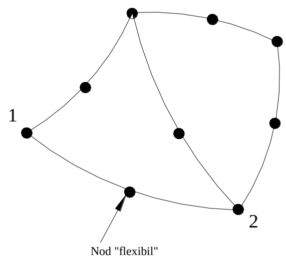

Figure.3. Tetraedale elements

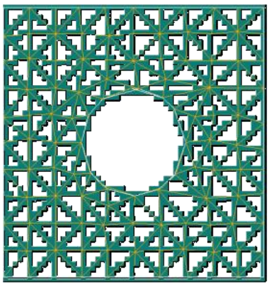

Figure 4.Fineţea discretizării

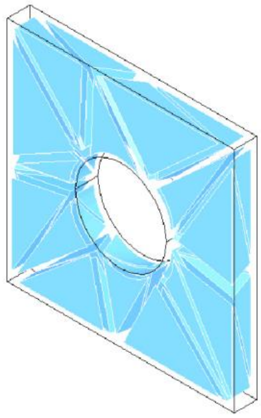

Figure 5.Metoda "p-element"

To remove these inconvenients is used the " p-element "- method which at digitizatization of the circular area use the "block" method (Figure 5).
The sensitivity of the digitization depends of the interpolation degree of the polynom.

3. Analysis of movements in a sphere-plane contact

We consider the sphere –plan contacts which represent the materialization of the point contacts from "Experimental Standfor the fretting study" which is in the Machine Elements Laboratory

Thus was considered the geometrical model shown in fig.6, the digitization of the elements being represented in fig.7.

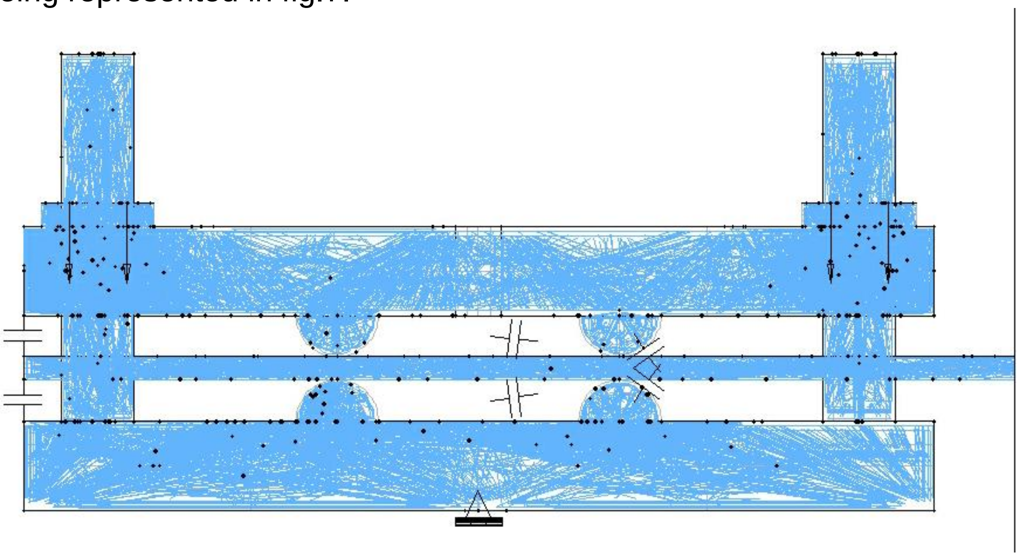

Figure 6. Theoretical model sphere-plan

The assumed model is represented by an elastic lamella assembled through 8 bearings ball ϕ 19 mm diameter, thus achieving eight point contacts. Lamella receives at the free end, an upward diplacement by 20 mm. The blade recess area is based on a fix lower part, top part being loaded by 4 screws with an uniform distributed force by 200 N on each screw.

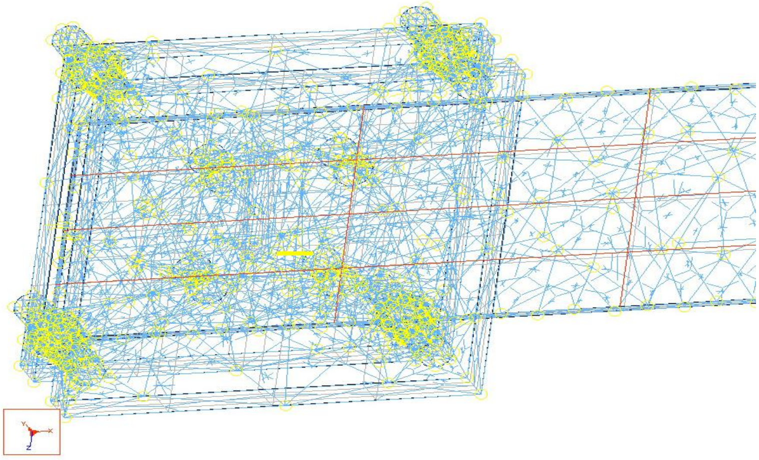

Figure 7.The digitization of the model

At the level of contact between the ball and lamella are obtained the total displacement given in fig.8 for balls on the top area and fig.9 for balls on the underside . At the level of the upper ball are obtained maximum total displacements between 7.52 mm and 6,20mm and the maximum total displacements for the lower balls are between 0.0175 mm and 0.0455 mm. Also, the pozition of the maximum displacements appropriately fonction by the the position of balls in contact. We analyzed the displacements at the level of the contact after the longitudinal axis X. These displacements are given in fig.10 and 11 and have maximum values between -3.21 mm and 2.10 mm for the superior balls and between 0.0455 mm and 0.0175 mm for the lower balls.

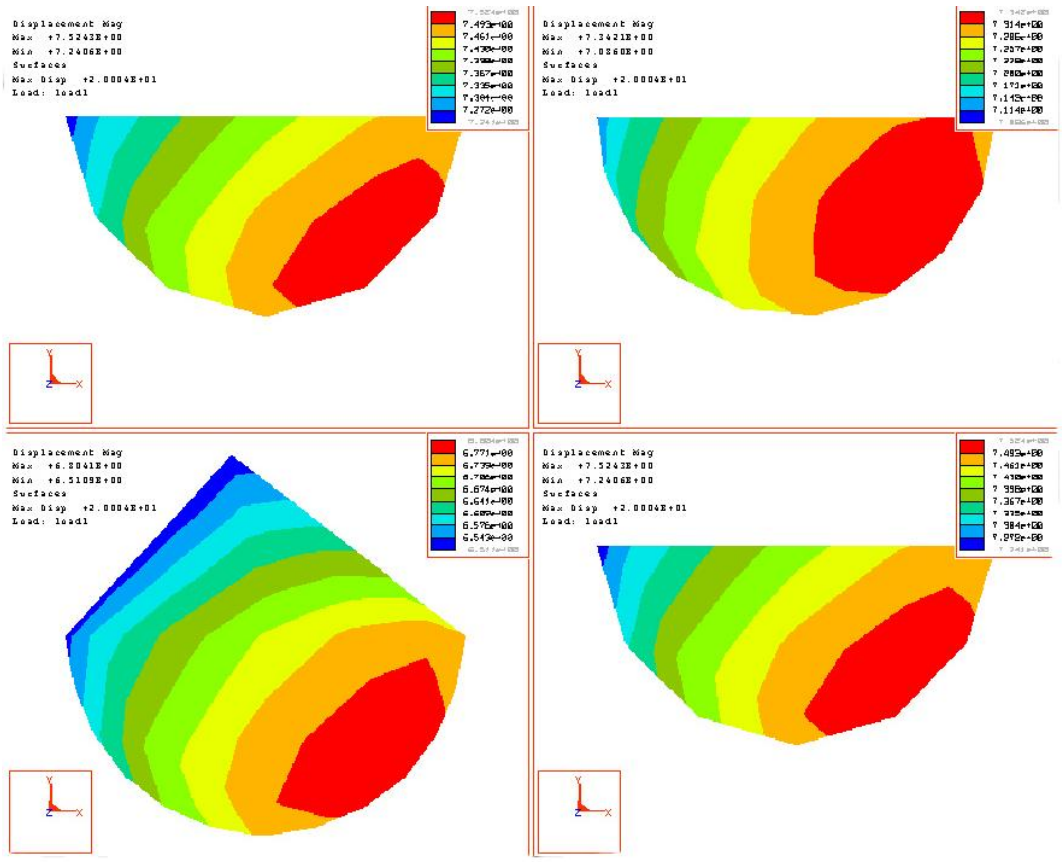

Figure 8. The total displacement at the level of the superior balls

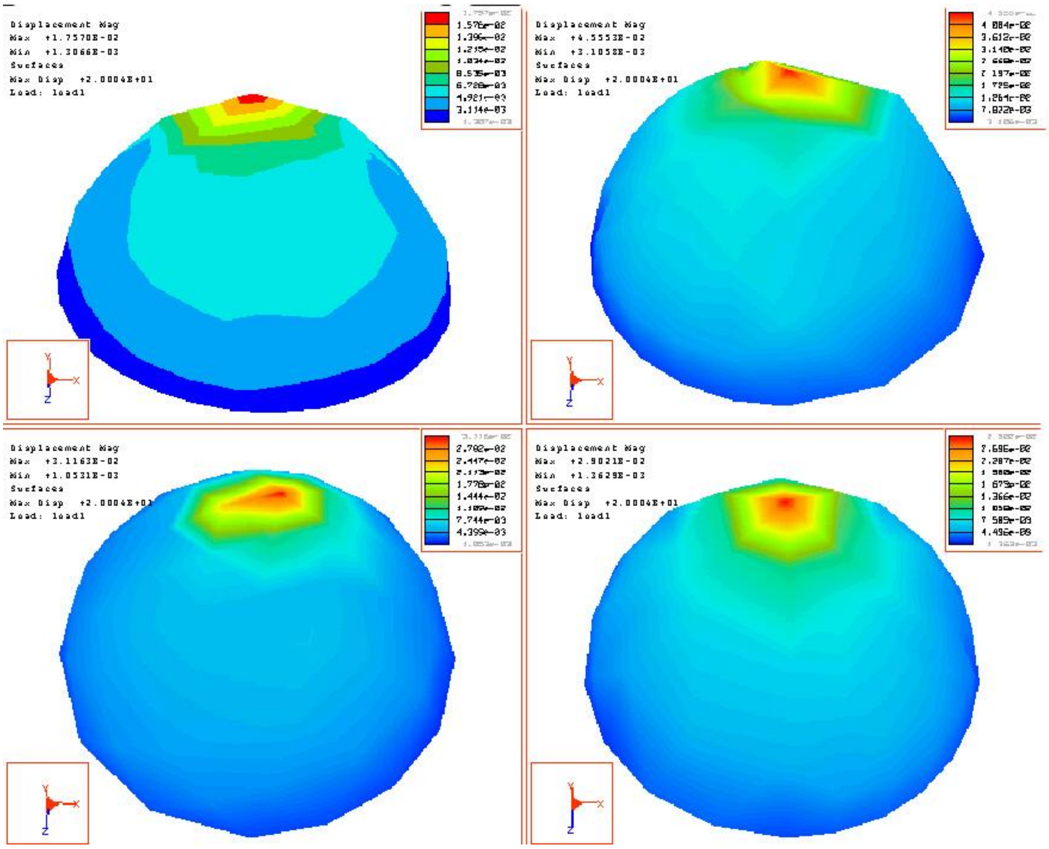

Figure 9.The total displacements at the level of inferior balls

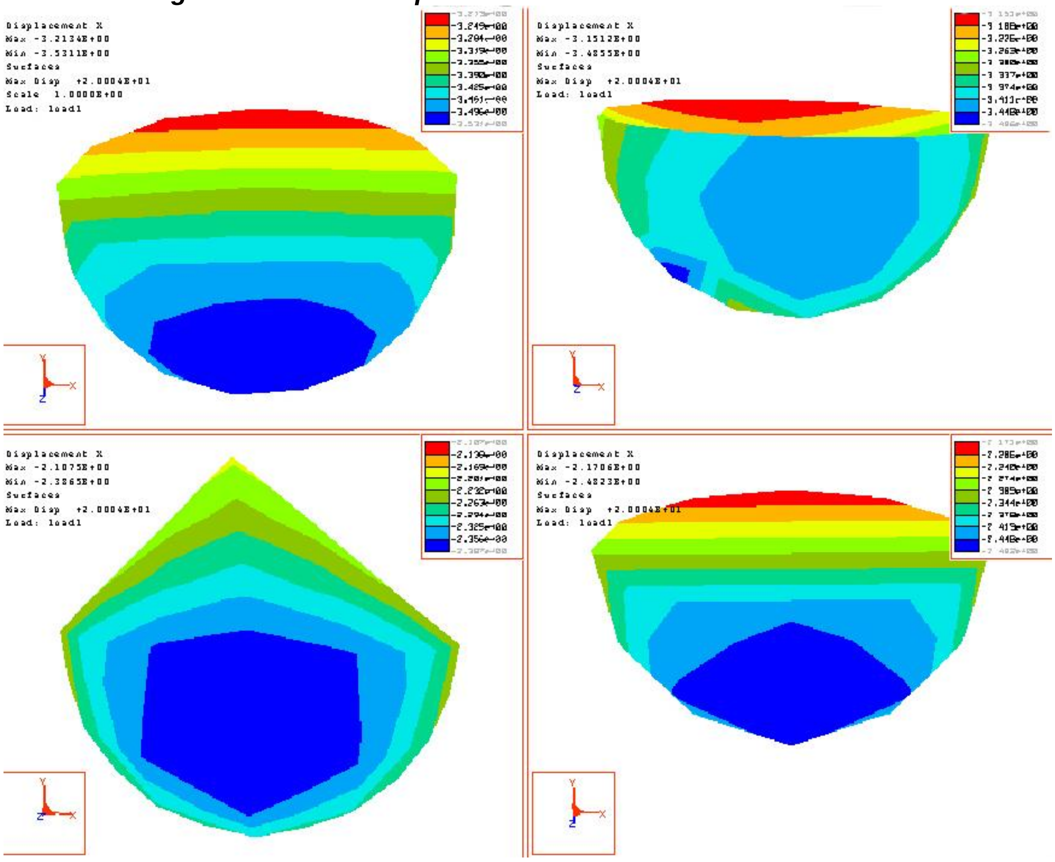

Figure 10.The displacement on the X axe at the superior balls level

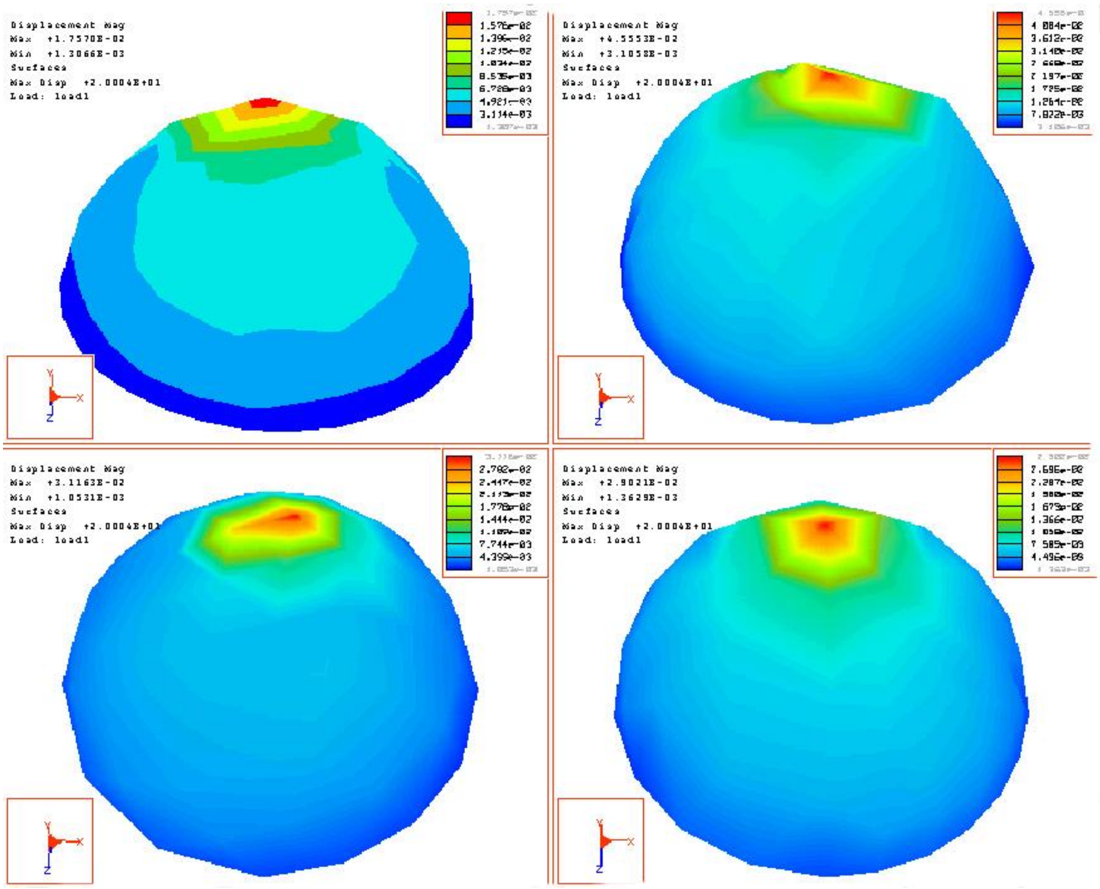

Figure 11.The displacement on the X axe at the inferior balls level

4.Conclusions

Analysis by finite element method is a modern method for studying various contacts, allowing the determination of important parameters for the study of various contacts. In case of hertzian contact are obtained precise informations regarding the state of deformation at the contact level , it can be obtained information about the state of the existing tension at the contact level. Analysis of the point contact using the classical method (Hertz theory) does not permit to to achieve the results of such precision .This analysis confirm the initial assumed characteristics ,characteristics which are proper the fretting test for the point contacts

References

1.Hertz,H. Uber die Beruhrung fester elastischer Korper . On the contact of elastic solids. J reine une angewandth, Mathematik n 92,p.156-171(voir traduction anglaise dans Miscellaneous Papers byH.Hertz,Eds Jones et Schott, London:Macmillan,1896, p.88-114

2.Ghimisi Stefan, Contribution regarding the wear by fretting with applications at the elastic assembling.Teză de doctorat, Universitatea Politehnica Bucuresti, 2000.