ANALELE UNIVERSITĂȚII
"EFTIMIE MURGU" REȘIȚA
ANUL XX, NR. 1, 2013, ISSN 1453 - 7397

Tiberiu Mănescu jr.

Considerations about how the Numerical Methods are Choose in Purpose to Calculate the Stress – Deformations from Rolling Bearings

The paper present the simulations made with finite element method for a simplified model of rolling bearing balls. The simulations were made with Ansys software. The simplified model of the bearing was made from a ball and two parallel planes

Keyword: *rolling bearing, finite element, Ansys, contact, stress*

1. Introduction

The paper is based on analytical studies of the author's for a specific case of a 12.7 diameter ball subjected to a 1250 N force. From geometrical point of view, the model is composed from (fig. 1):

- superior plane (inner rolling way of the bearing);
- the ball (identical with the bearing's ball);
- inferior plane (exterior rolling way of the bearing).

The geometrical model can be done 2D or 3D. For the 3D model, we have two symmetry planes so the geometry (and analysis) can be done only for a quarter, half or the entire simplified model. Because of it's dimensions, even for small dimensions of the finite elements the model solution doesn't require a long time from the point of view of solution's convergence.

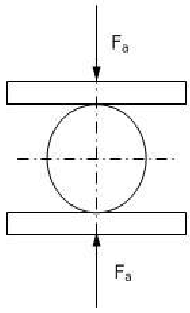

Figure 1: The simplified model

For 3D model, a Cartesian reference system was choose with the origin in ball's centre, Ox axis is the horizontal in paper's plane, Oz axis is perpendicularly on Ox axis and it's orientation is down and axis Oy perpendicularly on Ox and Oz axis is horizontal.

2. Geometrical model

In figure 2 is presented an isometric view of the model that was used for simulations. The geometry of the finite element model was done as follows:

- first I made the ball (12.7 mm diameter);
- then I build the upper plane (cube with 10 mm edge);
- finally I made the inferior plane (cube with 10 mm edge).

In purpose to generate contact elements between the ball and the superior/inferior plan only near the ball's "poles" I intersected the ball with two parallel planes, so that to cupolas were birth (fig. 3).

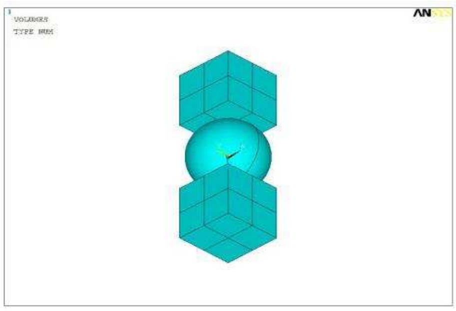

Figure 2. Isometric view

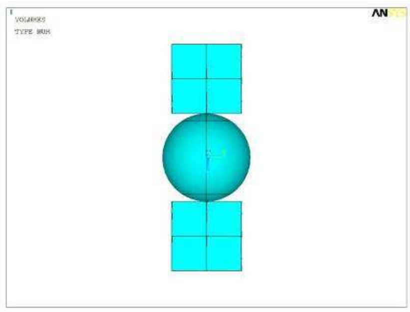

Figure 3. Lateral view

3. Finite element model

Based on geometrical model I meshed with Solid45 elements (bricks with nodes at corners). For a medium imposed size of 0.5 mm, 35193 nodes and 186661 elements result. In figures 4-7 is presented the meshed model.

Between the ball's cupolas and the inferior face of the superior cube and the superior face of the inferior cube, contact elements were generated. The contact elements were surface to surface elements. From the point of view of Ansys, the pair of contact surfaces is of two kinds: Conta and Target. The "Conta" surfaces were the surfaces of the cubes (fig. 8) and the "Target" elements were those from the cupolas (fig. 9). The Conta elements were Conta175 (970 elements) and the Target elements were Target 170 (794 elements).

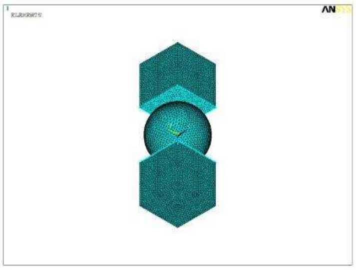

Fig. 4. The mesh of the model

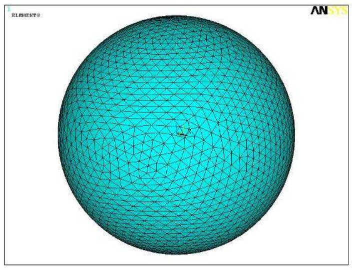

Fig. 5. The mesh ball

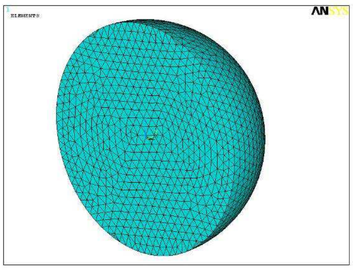

Fig. 6. Detail for half of the ball

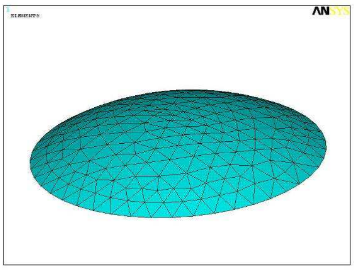

Fig. 7. Detail for upper cupola

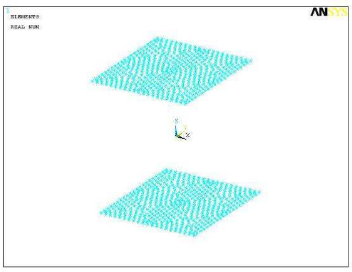

Fig. 8. Conta elements

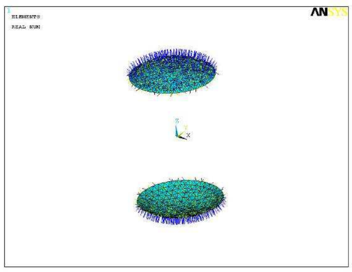

Fig. 9. Target elements

4. Force and constrains

The force was applied distributed on all the nodes of the upper surface of the superior cube. For the superior cube, translations after Ox and Oy axis were blocked (like guidance). The inferior cube was constrain on it's inferior surface after all axes (translations equal with zero). Plus, the ball was constraining in one node on the "Equator" (far enough from the interest points who are near the ball's "poles").

5. Contact setting

After contact pairs generation made from elements who are in contact (Conta and Target – fig. 10), Ansys allow setting different parameters, depending what the solving would be different and the solutions can be different. After I study the technical documentation of the software and specialized technical books (presented at paper's references) I choose "standard" for the contact mode (behavior of contact surface) and "Augmented Lagrange method" for contact algorithm.

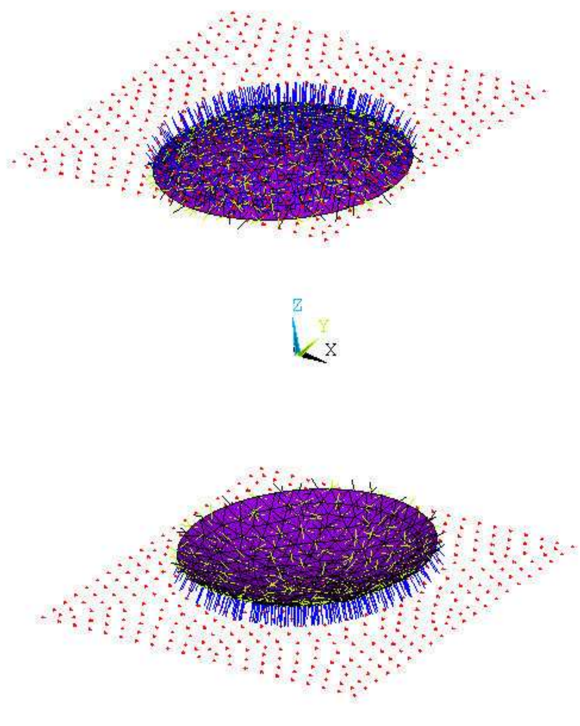

Figure 10. The contact pairs of the model

6. Results

After the solving of the problem, the results can be plotted as stress (after an axe, von Mises), deformations etc. both in nodes and on elements. The maximum stress is oriented after the vertical axe (Oz). in figures 11-12 are presented the results. The maximum deformations (after Oz axis) are $0.15 \cdot 10^{-3}$ mm and the maximum stress (after Oz axis) is -4215 N/mm^2, as it is shown in figures 11-12.

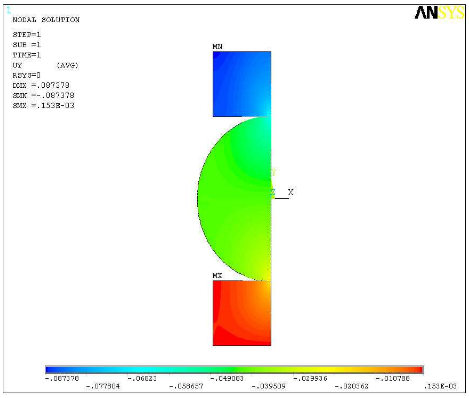

Figure 11. Vertical deformations

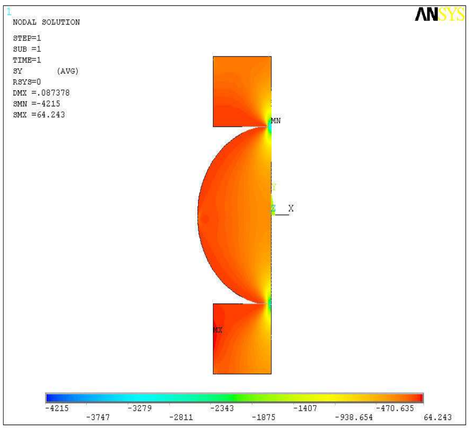

Figure 12. Vertical stress

7. Conclusions

The stress and deformations state of a model can be studied analytical, numerical and experimental. In the case of Hertzian contact, the analytical studies don't involve special complications. Based on Hertz theory mathematical formulas many applications (software) that are on free on the Internet can be used. But, such programs offer only the values of the stress or the dimensions of the contact patch without the possibility to see the stress gradient for example. In such conditions, the usage of the finite element software, even with the request of a higher level of the knowledge from the user, offer the advantage to calculate numerical the values for different parameters but also can us to see how the stress is propagated in the model.

For the studied case, the results difference between the analytical model and the numerical model is around 2%, so I can concluded that the finite element model is accurate.

References

[1] Mănescu T.Şt., Jiga G., Zaharia L., Bîtea C., *Noţiuni Fundamentale de Rezistenţa Materialelor*, Editura Eftimie Murgu, Reşiţa, 2008

[2] Mănescu T.Şt., Nedelcu D., *Analiză Structurală prin metoda Elementului Finit*, Editura Orizonturi Universitare, Timişoara, 2005.

[3] Mănescu T.Şt., Zaharia L.N. Mănescu T.jr., Iacob Mare C., *Noţiuni Fundamentale de Rezistenţa Materialelor şi Teoria Plasticităţii* Editura Eftimie Murgu, Reşiţa.

[4] Gillich G.R., *Dynamic machines. Modeling of technical systems*, Publishing House AGIR, Bucharest, 2003.

[5] Pop D., Tudose L., Haragâş S., *Lagăre cu Rulmenţi, Proiectare*, Todesco, 2006.

[6] INA, *Radial insert ball bearings, Housed bearing units*, Catalogue 520, INA Schaffer KG, 2002.

Address:

- PhD. stud. Eng. Tiberiu Manescu, "Eftimie Murgu" University of Reşiţa, Piaţa Traian Vuia, nr. 1-4, 320085, Reşiţa, t.manescu@uem.ro