Static Structural Analysis Analytical and Numerical of Ball Bearings

S. Belabend¹, V. Paunoiu², N. Baroiu², R. Khelif¹ and I. Iacob²
¹ “Badji Mokhtar Annaba” University of Annaba, Faculty of Engineering Science, BP 12, 23000 - Annaba, Algeria
² “Dunarea de Jos” University of Galati, Faculty of Engineering, Department of Manufacturing Engineering, 47 Domneasca St., 800008 - Galati, Romania

E-mail: b.belabend@gmail.com

Abstract. FEM analysis is a very efficient method for achieving results of stresses at different loading conditions according to forces and boundary conditions applied to the component the static analysis. The purpose of the study is to collect data's using two different softwares and then to compare them with analytical results. This work aims at analysing the behaviour of the ball bearings under a static load, using Solidworks, ANSYS and MESYS software. The comparison is done between the analytical results using the Hertzian theory, ANSYS and MESYS, for two different cases of loading.

1. Introduction

Bearings are important mechanical components that help to maintain both linear and rotational movement of a machine. The loads which the bearings are supposed to carry are crucial if we seek to ensure a long operational life to them. Loading is considered to be static in several cases such as for stationary bearings, low oscillations when working at low speeds or taking large short shocks while rotating [1-4].

Hertz's theory [1] is generally used to calculate the contact pressure and radial stiffness of ball bearings. The limitation of the hertz theory is transcribed in the fact that it can only consider a small portion of the contact surface of the ball bearings and it cannot include the total structural deformation of the ball, the outer ring, and the inner ring, as well as housing, this limit the engineering calculations when the total structure of the aforementioned deformation is considered. The finite element method remains very practical for structural analysis and gives convincing results in many types of engineering calculations. The contact condition between the rollers and the rings is complex and the use of FEM analysis a delicate task. These contact problems are too difficult. First, before the problem is solved the specific contact area is unrecognized. With the change in load, material, boundary condition, or other external factors, contact or separation will occur between surfaces. When the load is 0, the contact zone is reduces to a point, i.e. the point contact. As the load increases during operation, the inner ring of the bearing, the outer ring and the rolling elements cause deformation in the contact area so that the point contact becomes a surface contact. Besides, the contact area gradually becomes elliptical and generates residual stress. This phenomenon is not easy to predict. Furthermore, the effect of friction which as for the contact is not easy to take into account. They may be variable with loads of change.

This paper, presents a simplified analysis of the ball bearing static behaviour using the finite element method for determining the values of the contact pressure. The research methodology is presented in figure 1. The obtained FEM results were compared with the results obtained through the methodology
of Hertz theory based on specific analytical equations and also with those obtained with dedicated MESYS software.

Figure 1. Research methodology

2. Theoretical analysis of ball bearing static behaviour
In the elastic area, it is possible to calculate the contact pressure and the resulting deformation at the contact points of the rolling elements and orbits by means of Hertz theory, concerning the contact of the volume elastic bodies.

This theory is based on these following assumptions, [8, 9].
1. The material of the bodies in contact is considered homogeneous and isotropic.
2. The tangential forces are not considered in the contact area.
3. Contact stresses and deformations satisfy the differential equations for stress and strain of homogeneous, isotropic, and elastic bodies in equilibrium.
4. Contact is limited to a small portion of the surface (the stress disappears at a great distance from the contact zone).
5. In the absence of an external force, the contact zone degenerates to a point.

When two elastic solids are in contact under load F, figure 2, a contact area develops. For a point contact, the area, in general is assumes to be an elliptical shape and has a semi-major (a) in one direction and a semi-minor (b) in the perpendicular direction.

Figure 2. Contact between two spheres [2].

When the ball bearing is operating, there are usually more than one axis ball that is supporting the load axes. The condition is more complex between the rollers and the rings.
When the load increases in running, the bearing inner ring, outer ring, and rolling elements bring forth plastic deformation in the contact area, so the point contact becomes face-contact. Furthermore, the contact area gradually becomes an ellipse and generates residual stress [5, 9, 10].

The equations used in the theoretical analysis of a ball bearing are presented in the table 1:

Table 1. Equations used in the static analysis of contact pressure [10,11].

| Contact radius a, b | a and b are the dimensions of the contact ellipse, see the figure 1 |
|---------------------|---------------------------------------------------------------------|
|                     | $a = 1,145, n_a. (F. K. γ)^{1/3}$                                   |
|                     | $b = 1,145, n_b. (F. K. γ)^{1/3}$                                   |

$n$ and $e$ are used to simplify the equations

| $n_a = \frac{1}{k} \left( \frac{2kE(e)}{\pi} \right)^{1/3}$ |
| $n_b = \left( \frac{2kE(e)}{\pi} \right)^{1/3}$ |
| $n_d = \frac{1}{E(e)} \left( \frac{\pi^2 k E(e)}{4} \right)^{1/3}$ |

$e = \sqrt{1 - \left( \frac{a}{b} \right)^2}$

$E(e)$ Elliptic Integral

$E(e) = \int_0^\frac{\pi}{2} \sqrt{1 - e^2 \sin^2(\phi)} d\phi$ (7)

Coefficients in equation for locus of contacting points

$K = \frac{1}{r_1 r_2 + r_3}$ (8)

Relation between the elastic parameters of the two solids in contact

$\gamma = \frac{(1-\psi_1^2)}{E_1} + \frac{(1-\psi_2^2)}{E_2}$ (9)

Rigid distance of approach of contacting bodies

$d_r = 0.655. n_d. \left( \frac{F^2 \psi^2}{\pi} \right)^{1/3}$ (10)

Ratio of $b$ to $a$

$k = \frac{b}{a}$ (11)

The maximum pressure [7, 11]

$P_{Hertz} = \frac{3F}{2 \pi a b}$ (12)

where: $a$ - semi-major axis of elliptical contact; $b$ - semi-minor axis of elliptical contact; $d_r$ - rigid distance of approach of contacting bodies; $E$ - Young’s modulus; $\psi$ - Poisson’s ratio; $P_{Hertz}$ - maximum pressure (or stress) at the center of an ellipsoidal; $E(e)$ - elliptic Integral; $R_1$ - ball radius; $R_2$ - external race radius; $R_3$ - inner race radius; $F$ - maximum load on a bearing ball, [10,11].

Table 2. Parameters of the contact between the balls and the race.

| Parameters          | Sphere | Circular Race | Unit    |
|---------------------|--------|---------------|---------|
| Poisson’s ratio [$\nu_1, \nu_2$] | 0.29   | 0.29          | ---     |
| Elastic modulus [$E_1, E_2$]     | 210000 | 210000        | MPa     |
| Radius of objects      | $R_1, R_2$ | Ball radius | 8.01 mm |
|                       | $R_1, R_3$ | Ball radius | $7.54 \times 7.665$ mm |
| Force [$F$]            | 10     |               | KN      |

2.1. Analytical results

Based on the parameters presented in table 2, the analytical calculations results are presented in the table 3.
Table 3. Analytical results.

| Parameter                                      | Unit    |
|-----------------------------------------------|---------|
| Maximum Hertzian contact pressure, \( P_{Hertz} \) | 3110 MPa |
| Rigid distance of approach of contacting bodies, \( d_r \) | 0.038 mm |
| Semi major axis of contact ellipse, \( a \)      | 0.312 mm |
| Semi minor axis of contact ellipse, \( b \)      | 4.930 mm |

3. Simulation of static behaviour in ANSYS

ANSYS is commonly used and enjoy extended use in structural analysis. ANSYS consists of three main phases: pre-processing, conducting or importing of the solid model system that is to be analysed, solid meshing design in finite elements, implementation of boundary conditions and loads limit, processing, numeric solving of the characteristic equations of the system and getting the solution. Post-processing, viewing the results to analyse system reaction and identification of areas with critical applications [6].

In this paper it was considered a ball bearing according to the URB RULMENTI S.A. Barlad company catalogue [12], designation 6308 C3L, which have an inner diameter (d) of 40 mm, an outer diameter (D) of 90 mm, and a width (B) of 23 mm. The model is made in SolidWorks (CAD (*.SLDASM. format), figure 3.a, and the meshing are done after importing the model in ANSYS. Because of 3D geometry, the tetrahedron element was used, figure 3.b. In the present analysis, the effect of the cage is neglected.

![Figure 3.a SolidWorks model.](image)

![Figure 3.b ANSYS meshing model.](image)

For both rings and balls the material used was ASTM 52100 Bearing Steel. The material properties are presented in table 4.

Table 4. ASTM 52100 Bearing Steel Properties.

| Property                                      | Value         |
|-----------------------------------------------|---------------|
| Bulk modulus (typical for steel)              | 140 GPa       |
| Shear modulus (typical for steel)             | 80 GPa        |
| Elastic modulus                               | 190-210 GPa   |
| Poisson’s ratio                               | 0.27-0.30     |
| Hardness, Rockwell C                          | 60-67         |
| Hardness, Vickers (converted from Rockwell C hardness) | 848 |
| Machinability (spheroidized annealed and cold drawn. Based on 100 machinability for AISI 1212 steel) | 40 |

3.1. Boundary conditions

The value of external load is 10,000 N as Bearing Load in the negative y-axis direction with a bearing time of one second for simulation of impact load. (The value of the force is used in multiple papers and references in the literature [3;19])
The FEM analysis was performed in ANSYS Workbench. The simulation of the investigated bearing was solved as a linear isotropic material model with given material properties of linear structural steel $E = 210,000$ MPa, $\nu = 0.29$.

The boundary conditions of the bearing, table 5, have been specified for each part as follows:

The outer ring and also inner ring displacement were removed in the X-axis direction and allowed in the direction of the Z and Y axes in the global Cartesian coordinate system. The coordinate systems are marked in figure 4.

- The first case (figure 4.a): the load is applied on the inner ring and the outer ring is fixed, as boundary condition
- The second case (figure 4.b): the load is applied on the outer ring; and the inner ring is fixed, as boundary condition

The contact between the elements, figure 5, was given as friction contact $f=0.002$ (the chosen of the friction contact coefficient for Deep Groove Ball Bearing (0.0010 ~ 0.0015)) [13][18].

| Displacement | X | Load | Y | Intensity of the load | 10,000N | Friction Contact | $f=0.002$ |
|--------------|---|------|---|----------------------|--------|------------------|-----------|

**Table 5. Boundary conditions.**

3.2. Meshing

The mesh structure is the subdivision of the mathematical model which has a simple geometric form, and does not overlap, called a finite element. The answer to each finite element simulation is expressed on a finite number of freedom degrees that represent the values of unknown function (movement function) in several crucial points. Thus, the mathematical model will result in an approximation of the response obtained by assembling a discrete model with all other model elements [14]. Optimization of the mesh was carried out in order to obtain a continuous mesh and a correlation between the number of finite elements and the number of associated nodes. Choosing a simple type, here we used “Tetrahedrons”, will result a mesh of top quality with 28716 nodes and 97422 elements in view of the structure and complexity, figure 6 and table 6. Also, for rings were established a custom mesh [15].

| Type       | Tetrahedrons |
|------------|--------------|
| Element    | 97422        |
| Nodes      | 28716        |

**Table 6. Meshing.**
The simulation parameters are similar to those used for theoretical analysis shown in Table 2.

3.3. Simulation results
After running the model, we obtain important information about the contact pressure.

The results of the simulation are presented below:

The first case of loading:

![Figure 7. Contact pressure (case 1).](image)

**Table 7. Results for the first case of loading.**

| Object Name | Pressure |
|-------------|----------|
| Minimum     | 0 MPa    |
| Maximum     | 1211.5 MPa |

The second case of loading:

![Figure 8. Contact pressure (case 2).](image)

**Table 8. Results for the second case of loading.**

| Object Name | Pressure |
|-------------|----------|
| Minimum     | 0 MPa    |
| Maximum     | 1212.8 MPa |

In both cases, the biggest contact pressure appears at the contact point to correspond the line which radial force acts.

4. Simulation with MESYS
The rolling bearing analysis software MESYS calculates the life of a rolling bearings according to ISO/TS 16281 considering the inner geometry of the bearing, the load distribution within the bearing, the clearance and the tilting angle will affect the resulting bearing life. The inner geometry of the rolling bearing is provided by the user, but it can also be approximated from the load capacities by the software. The calculation returns the pressure distribution between the rolling elements and the reference life according to ISO/TS 16281 for a given loading (force and moment or tilting) [14, 16].
The input of the bearing geometrical parameters on MESYS and the loading $F (0 \text{ N}; 10,000 \text{ N}; 0 \text{ N})$ are presented in figures 9 and 10.

**Figure 9.** Application of the load in MESYS.

**Figure 10.** Contact stresses

In MESYS software the user could not choose where to apply the load or the external load, unlike Ansys software where the chosen of the point of the application of the load is easily configured. The user can only define the amplitude of the load. Results are provided as a small result overview directly in the software, a main PDF text report, a tolerances report, and a separate graphic window. In figure 11 are presented the results of simulations.

**Figure 11.** Results of MESYS simulations
5. Comparison between the results

Through the analysis, it was found that the value of the maximum contact pressure gets different size between the first case applying the load on the inner ring and fix the outer ring and the value of for the second case, applying the load on the outer ring and define the inner the ring as fixed support with a deviation of 0,001%.

Table 9. Comparison between the obtained results.

|                        | Analytical result contact pressure, MPa | ANSYS result contact pressure, MPa | MESYS result contact pressure, MPa |
|------------------------|----------------------------------------|------------------------------------|-------------------------------------|
| Maximum pressure       | 3110.00                                | 1211.5                             | 1212.8                              |
| Error MESYS result     | 1.6%                                   | 59.5%                              | 59.4%                               |
| Error ANSYS Result     |                                        | 61.04%                             | 61.00%                              |
|                        |                                        | Case 1                             | Case 2                              |

The results of the simulation by finite element method, and their comparison with the analytical results, in terms of contact pressure, see table 9, show a deviation 61% as percentage which is in accordance to the work of P. Šulka, A. Sapieťová [3].

6. Conclusions

In the paper, we made a comparison between the results obtained from the analytical model, obtained with the ANSYS software and those obtained with the MESYS software Hertzian contact pressure.

The comparison between the MESYS results and the analytical calculation results are closer, because MESYS uses the same equations as the analytical ones to calculate the maximum Hertzian pressure.

Analysing the results between MESYS and ANSYS led to the following conclusions: the observed differences may be because MESYS does not take into account the presence of the cage; at the same time, ANSYS does not take into account the roughness of the surfaces.

The results and data obtained from this study can be used to improve the characteristics and geometries of the bearings of the next-generation products.

7. References

[1] T A Harris and M N Kotzalas 2006 Essential concepts of bearing technology: CRC press
[2] S Belabend 2019 Maintenance based on simulation of static structural analysis for bearing behavior Dunare de Jos Galati. Badji Mokhtar Annaba University, Galati,
[3] J-P Gerval, G Morel and C Querre, Behaviour modelling and vibrations analysis applied to predictive maintenance: Different approaches leading to the same conclusions 1 pp 1/544-1/546
[4] P Šulka, A Sapieťová, V Dekýš et al. Comparison of analytical and numerical solution of bearing contact analysis p 02022
[5] H Hertz 1987 Le mémoire de hertz sur les contacts ponctuels: Ecole nationale supérieure d'arts et métiers
[6] A M Pușcașu, O Lupescu and A Bădăncăc Study regarding the structural response of standard cylindrical roller bearings using ANSYS and MESYS p 0512
[7] B Zengin, S Taşkaya and K Kaymaz 2018 Investigation of force and moment effect of St 37 and St 70 roof lattice steels in Ansys program Middle East Journal Of Science, 4(1) pp 23-35
[8] P Šulka, A Sapieťová, V Dekýš et al. Static structural analysis of rolling ball bearing p 01023
[9] A Jones 1960 A general theory for elastically constrained ball and radial roller bearings under arbitrary load and speed conditions
[10] K Chennakesavulu and D Rao 2015 Structural analysis of ball bearings in ANSYS International Journal of Scientific Engineering and Technology Research, 4(35) pp 7086-90
[11] P Guay, and A Frikha Ball bearing stiffness. A new approach offering analytical expressions pp 23-25
[12] URB 2017 URB Ultra Reliable - General catalog
[13] V P Selma Belabend and R Khelif 2019 *Simulation Of Ball Bearing Behaviours Under Static Loading* Scientific Conference of Doctoral Schools SCDS-UDJG Perspectives and challenges in doctoral research, Galati.

[14] Z Yongqi, T Qingchang, Z Kuo et al. 2012 *Analysis of Stress and Strain of the Rolling Bearing by FEA method* *Physics Procedia* 24 pp 19-24

[15] M Bucki, C Lobos, Y Payan et al. 2011 *Jacobian-based repair method for finite element meshes after registration* *Engineering with Computers*, 27(3) pp 285-297

[16] I. O. 1990 Standardization, *Rolling Bearings: Dynamic Load Ratings and Rating Life*: International Organization for Standardization

[17] D Collins 2020 *What are Hertz contact stresses and how do they affect linear bearings?* https://www.linearmotiontips.com/what-are-hertz-contact-stresses-how-do-they-affect-linear-bearings/

[18] *** 2020 *Selection of bearing type* https://koyo.jtekt.co.jp/en/support/bearing-knowledge/3-0000.html.

[19] W Jacobs, R Boonen, P Sas et al. 2014 *The influence of the lubricant film on the stiffness and damping characteristics of a deep groove ball bearing* *Mechanical Systems and Signal Processing* 42(1-2) pp 335-350

**Acknowledgements**

This work was supported by Erasmus+ Programme, KA2 Capacity Building in Higher Education, project no.586035-EPP-1-2017-1-DZ-EPPKA2-CBHE-JP, entitled Algerian National Laboratory for Maintenance Education - ANL Med.