THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XIV MECHANICHAL ENGINEERING, ISSN 1224-5615
2017
https://doi.org/10.35219/im.2017.2.04

DYNAMIC ANALYSIS OF SHAFTLINE PROPULSION OF VESSELS USING FINITE ELEMENTS METHOD

Drd. Eng. Nicolae IACOB
Institute of Solid Mechanics,
Romanian Academy, Bucharest
Assoc. Prof. Dr. Eng. Nicusor DRAGAN
MECMET - The Research Center of Machines,
Mechanic and Technological Equipments
Engineering and Agronomy Faculty of
Braila
"Dunarea de Jos" University of Galati

ABSTRACT

The article presents a dynamic calculation method for long shafts, especially for the shaft line of vesselpropulsion. The calculation is based on finite elements method ANSYS 14.5 WORKBENCH software, where the authors simulate 3 events: static analysis, modal analysis, transient analysis. From all numerical simulations are issued calculation reports and general conclusions.

KEYWORDS: static analysis, dynamic analysis, modal analysis, transient analysis, finite elements method ANSYS 14.5

INTRODUCTION. CALCULATION METHOD

The FEM analysis with ANSYS 14.5 WORKBENCH encompasses the following steps [1]:

►modeling the analysis geometry;
►choosing the material;
►meshing to geometry, 11047 node and 2639 finite elements in this case;
►putting limit conditions;
►post-processing the results.

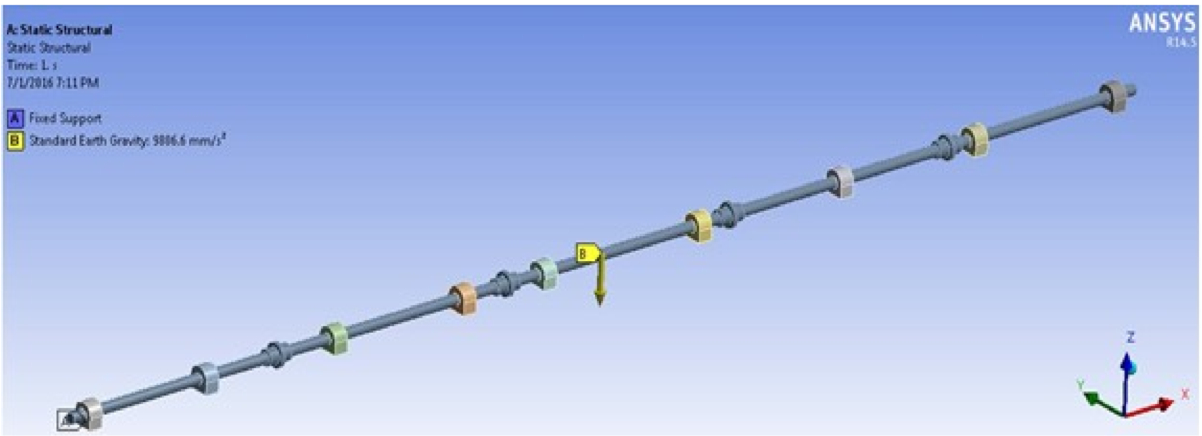


Fig. 1 Calculation scheme for static analysis

17

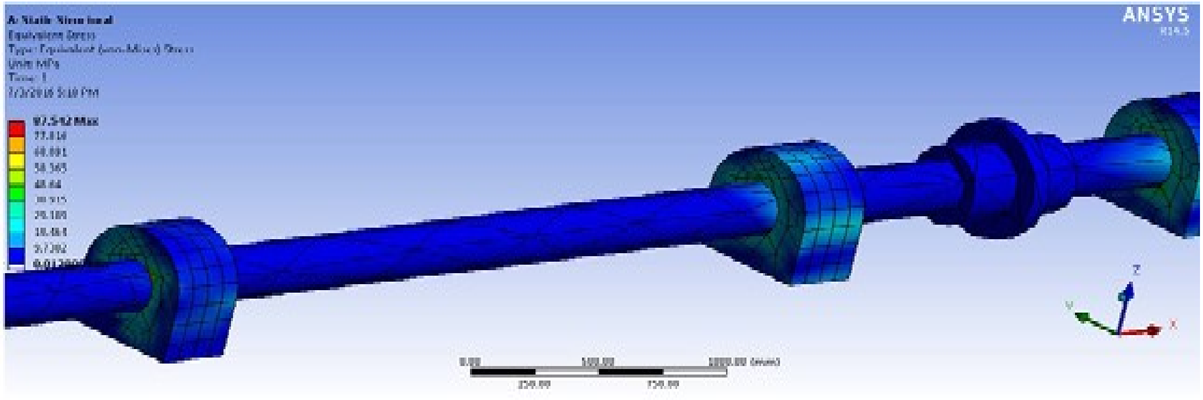


Fig. 2 Tensions distribution

STATIC ANALYSIS

Figure 1 presents the calculation scheme for static analysis. The 28 m length shaft has nine rigid bearings.

Between the shaft and each bearing, the contact friction coefficient is 0.1. The bearings are fixed on the base and the shaft loads are due to the proper weight only.

Figure 2 shows the tensions distribution in mechanic system (shaft + bearings). The local tension inside the bearing is 87.5 N/mm^2.

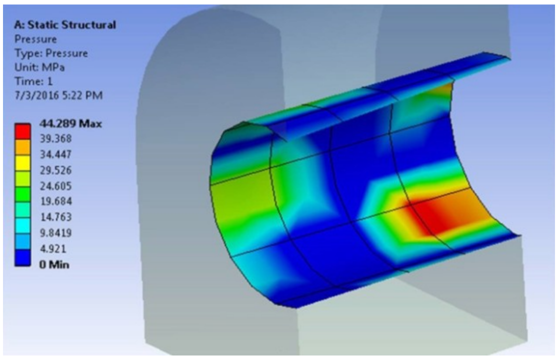
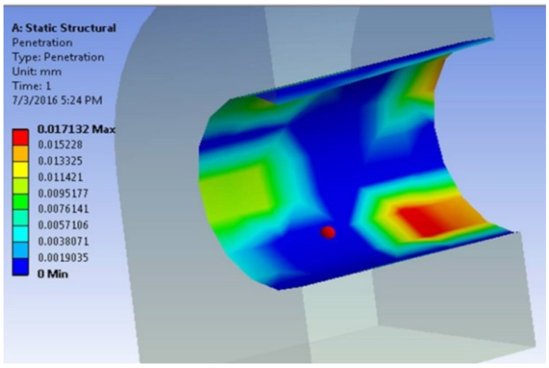


Fig. 3 Contact tensions

Figures 3 and 4 show the tension distribution and the deformations distribution of the internal surface contact of the bearing. The values of the tensions are between 40 MPa and 87 MPa. The deformation in contact zone is 0,01 mm.

MODAL ANALYSIS

Modal analysis is released with MODAL module from the workbench ANSYS platform. The limit condition was imposed on the cylinder articulation contact of the bearing. In table 1 are presented the modal frequencies of the shaft. The third vibration mode is torsional, the first, the second, the fourth, the fifth and the sixth vibration modes are flexional.



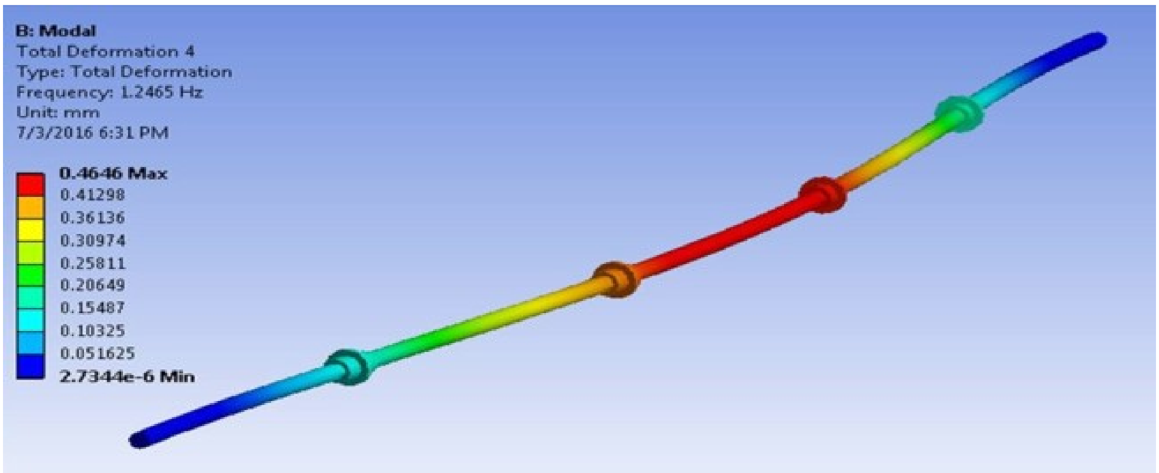


Fig. 5 First mode of vibration (1.2465 Hz)

Fig. 4 Contact deformations

Table 1. Modes of vibration

Mode	Deformation	Natural frequency [Hz]
1	flexional	1.2465
2	flexional	1.2487
3	torsional	1.5395
4	flexional	3.4628
5	flexional	3.4672
6	flexional	6.8403

Figures 5 and 6 show the first two modes of vibration. The deformation at resonance has a maximum value of approx. 0.46 mm.

Figure 7 shows the third mode of vibration. In this mode of vibration, the shaft has maximum 0.6 rad angular deformation.

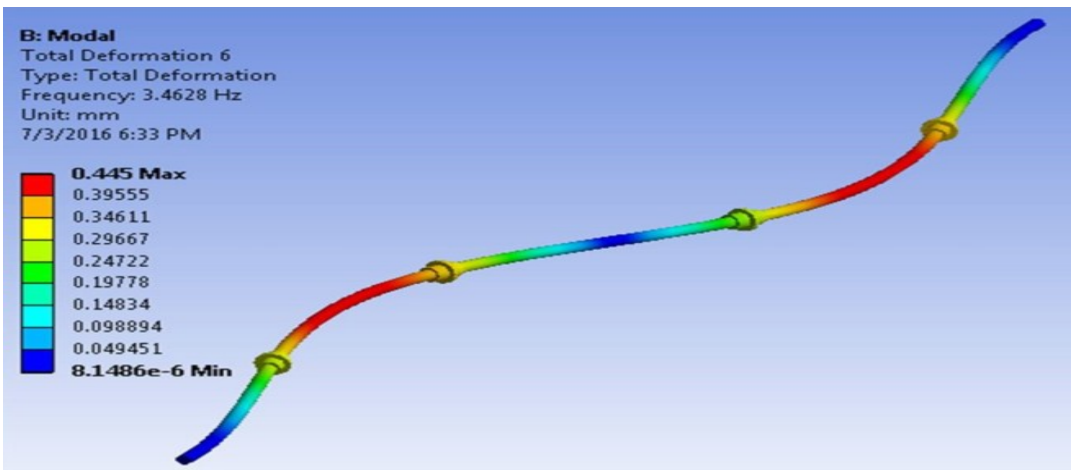


Fig. 8 Fourth mode of vibration (3.4628 Hz)

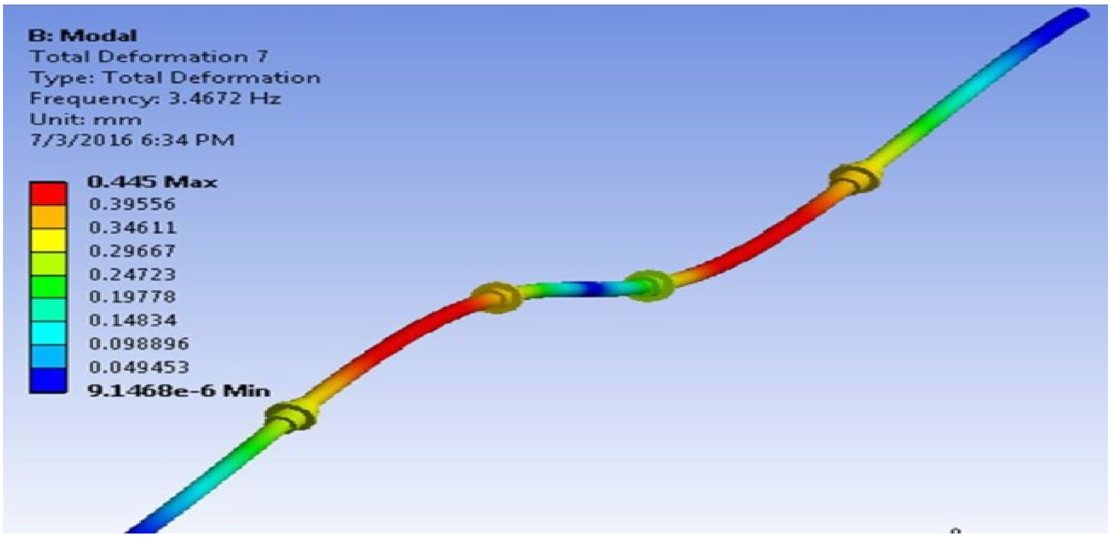


Fig. 9 Fifth mode of vibration (3.4672 Hz)

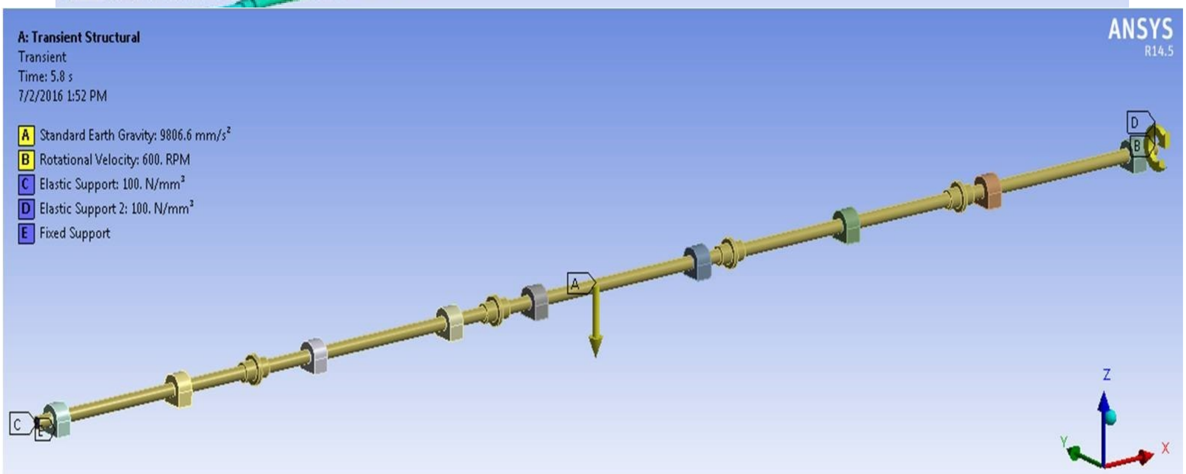


Fig. 10 The limit condition for transient structural analysis

Fig. 7 Third mode of vibration (1.5395 Hz)

Figures 8 and 9 show the fourth and the fifth modes of vibration. The deformation at resonance has a maximum value of approx. 0.45 mm.

TRANSIENT STRUCTURAL ANALYSIS

Transient structural analysis was released with TRANSIENT STRUCTURAL module from the workbench platform from ANSYS 14.5. The limit conditions (see Fig. 10) are:

►the bearings are fixed at the base;
►the static load is the shaft gravity force;
►the dynamic load is the centrifugal force with variable angular velocity (acc. to diagram from Fig. 11); the angular velocity increases from 600rpm to 800rpm and then decreases to 600 rpm.

The program gives the possibility to visualize the graphs of the displacement, the velocity and the acceleration of vibration. The amplitude of vibration is 0,18 mm at the beginning of angular motion then is stabilized to 0.125 mm. Figure 12 shows the time-variation of the displacement of vibration at critical angular speed.

Figure 13 shows the time-variation of velocity of vibration at critical angular speed. The maximum value is 10,89 mm/s at the beginning of the motion.

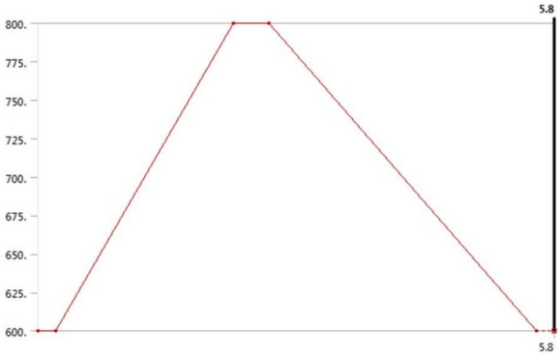


Fig. 11 Variable angular velocity diagram

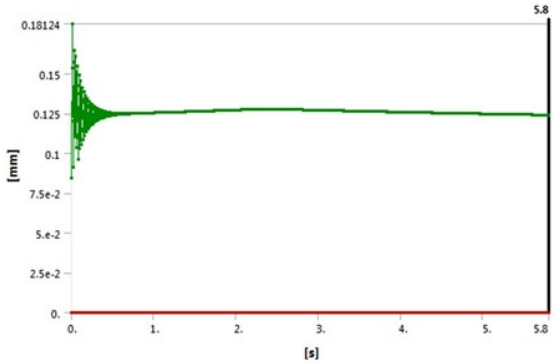


Fig. 12 Time-variation diagram of the velocity at critical angular speed

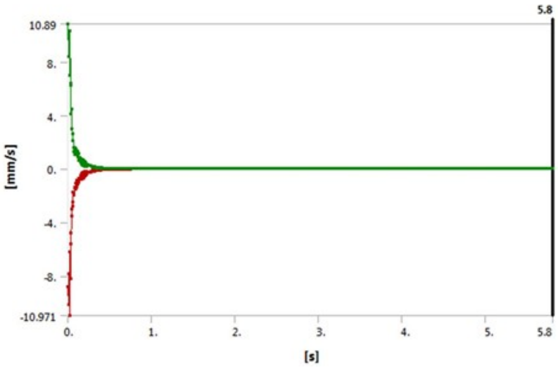


Fig. 13 Time-variation diagram of the velocity at critical angular speed

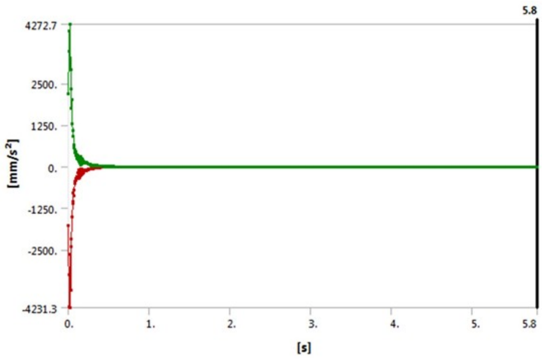


Fig. 14 Time-variation diagram of the acceleration at critical angular speed

Figure 14 shows the time-variation of acceleration of vibration at critical angular speed. The maximum value is 4,89 m/s^2 at the beginning of the motion.

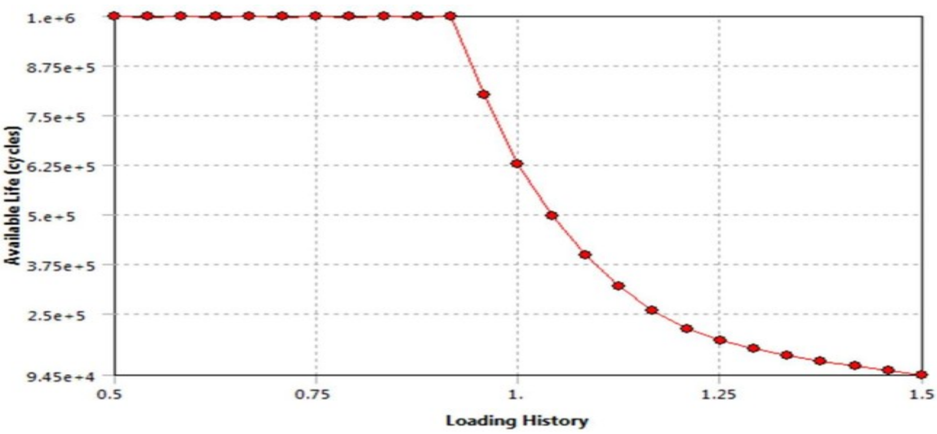


Fig. 17 Life cycle diagram

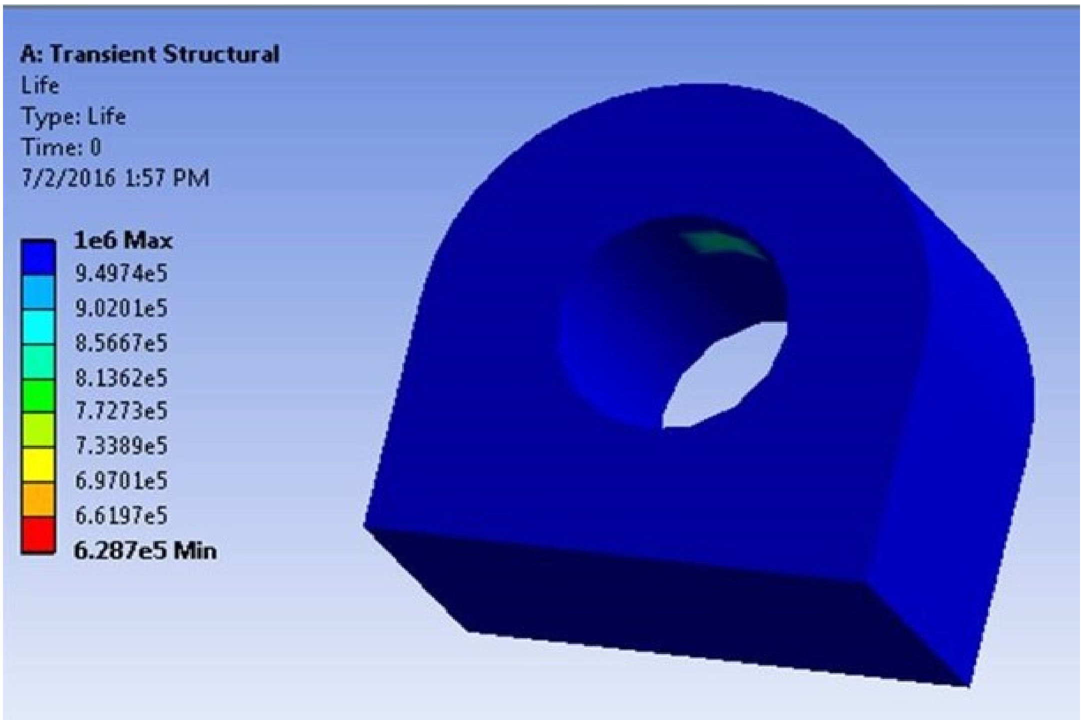


Fig. 18 The diagram of the bearings damage

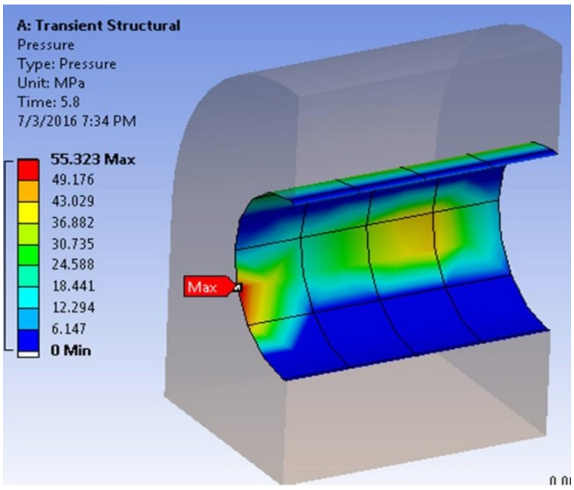


Fig. 15 Time-variation diagram of the acceleration at critical angular speed

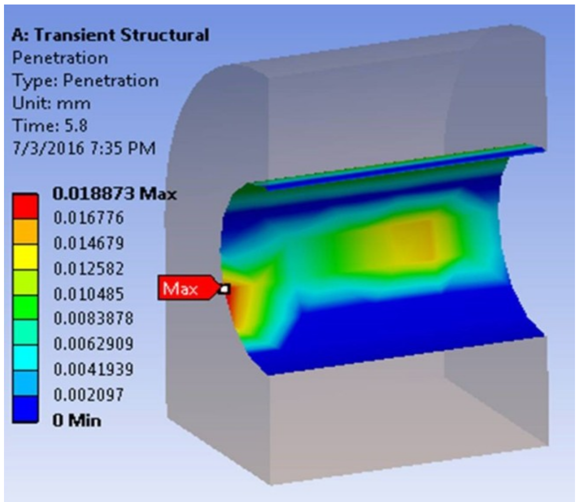


Fig. 16 Time-variation diagram of the acceleration at critical angular speed

With the module of dynamic analysis, can be calculated and shown the diagram of the tensions distribution in the interior of the bearing at critical velocity speed. The tensions in dynamic regime are double than in static regime (93,4 MPa as against 40 MPa), thus the bearing can be damaged. The distribution of tension is not uniform on the internal surface of the bearing. Figures 15 and 16 show the tension distribution and the contact deformation distribution in dynamic mode.

CONCLUSIONS

For the correct operation of the system, it is necessary to avoid reaching the mechanical system natural frequencies in order for them not to resonate and compromise the mechanical system. The parametric model technique can, in some circumstances, be a way to increase the efficiency of modeling and FEM analysis for mechanical structures. The parametric model has a remarkable insight because it can be introduced with minimal changes in any FEM software and can be used as such or can be defined as a substructure or as a component submodel of a model as complex as possible.

Under the terms of design workshops or computational groups, different parametric models developed over times can be integrated in libraries, in order to be used in a CAD system.

The FEM models used for optimization are partially parametric because some model dimensions, namely design variables, are defined by algebraic notation and the optimization process will be determined. We can easily imagine the situation where all dimensions of the model are defined as design variables, which leads to a fully parametric model that will have the functions and advantages presented above.

REFERENCES

[1] https://www.ansys.com/academic