FINITE ELEMENT ANALYSIS OF A ROTARY SHAFT LIP TYPE SEAL OPERATING WITH A DYNAMIC ECCENTRICITY OF THE SHAFT

RADIALLINO SANDARINIMO ŽIEDO, DIRBANČIO ANT VELENO SU EKSCENTRICITETU, ANALIZĖ BAIGTINIŲ ELEMENTŲ METODU

Andreas Lucko, Stefan Meyer, Volkert Martin Wollesen

Technical University Hamburg-Harburg, Institute for Mechanical Engineering Design, Hamburg-Harburg, Denickestr. 17, 21071 Hamburg, Germany
E-mail: wollesen@tuhh.de

Gauta 2004-10-30, pateikta spaudai 2005-04-05

The research presented in this article is part of a project where several German universities and the German Institute of Rubber Research are developing a new constitutive model for rubber that shall be more precise than all existing ones. A thorough analysis of a technical rubber product, the rotary shaft lip type seal, is performed at TUHH in order to verify the new constitutive model. Experiments are carried out and FE-models of several operating conditions are developed. This article describes several ways how a rotary shaft lip type seal operating with a dynamic eccentricity of the shaft can be modeled with the commercial FE-program ABAQUS. In a first step the dynamic eccentricity is simulated with two 3D FE-models, one neglecting friction effects, the other one taking them into account. In a second step it is shown that the dynamic eccentricity can be simulated with an axisymmetric FE-model with a shaft that changes its diameter with respect to time. Results of the simulations are evaluated with a focus on computational efficiency and the quality of results.

ABAQUS, dynamic eccentricity, finite element analysis, rotary shaft lip type seal, rubber

Introduction

Finite Element Analysis (FEA), also referred to as Finite Element Method (FEM), is the preferred numerical method in engineering applications if partial differential equations are necessary to describe a technical system. FEA was originally developed for static structural mechanics; nowadays it is also used in the areas of structural dynamics, fluid dynamics, heat transfer and acoustics [8]. In the design process FEA is used to analyze and optimize preliminary designs before the first prototype is built. Development time and costs can thus be reduced [7].
An FEA model in structural mechanics consists of the geometry of one or more parts, external force and displacement boundary conditions, and a constitutive model. For a linear-elastic material the constitutive model is rather simple since the material behavior is fully described by Young’s Modulus and Poisson’s ratio. Assuming that the FEA is used properly the results from simulations for linear-elastic materials differ from the behavior of the real system by less than 10% [7].

On the other hand the use of FEA in the design of rubber parts is less reliable. Rubber has certain characteristics (huge reversible deformations, internal damping etc.) that makes it interesting for a wide range of technical applications. However, since its material parameters depend on frequency, time, velocity, temperature, aging processes, self-heating, etc. the formulation of a constitutive model is much more complicated than for a linear-elastic material. None of the constitutive models available in commercial FE-programs fully describe the behavior of rubber [4, 10]. Therefore, experimental techniques are still the preferred method in sealing design [2].

Several German universities and the German Institute of Rubber Research are working together on a research project with the overall aim of developing a new constitutive model for rubber that shall be more precise than the existing ones. It is planned to implement this new constitutive model into the commercial FE-programs ABAQUS and MARC, thus enabling designers to do more reliable rubber simulations.

At the Technical University Hamburg-Harburg (TUHH), the Institute for Design in Mechanical Engineering, Arbeitsbereich Konstruktionstechnik 1, is involved in this research project. In a first step several operating conditions of a rotary shaft lip type seal are analyzed experimentally. The rotary shaft lip type seal serves as example for a technical rubber product. Thus a database that can be used in order to verify the new constitutive model is created. In a second step FE-models of these operating conditions are prepared with the FE-programs ABAQUS and MARC. Currently these FE-models make use of existing constitutive models.

As part of the research it is analyzed how a rotary shaft lip type seal operating with a dynamic eccentricity of the shaft can be modeled with the FE-program ABAQUS. Therefore, three different FE-models are prepared, simulated and the results analyzed. As a first approach the dynamic eccentricity is modeled in 3D whereby friction effects are neglected. Thereafter, friction effects are taken into account and finally it is analyzed how a dynamic eccentricity can be described with an axisymmetric FE-model.

**Defining Dynamic Eccentricity**

Dynamic eccentricities cause cyclic stress variations in rotary shaft lip type seals. Therefore, this operating condition appears to be suitable for verifying the new constitutive law. The term dynamic eccentricity as it is used in this paper is actually a special case of a combination of a static eccentricity and a dynamic eccentricity. A definition is given below.
A pure static eccentricity of a rotary shaft lip type seal appears if the axes of symmetry of the shaft and the seal are not the same (Fig. 1a). The space between the centers of the two machine parts and possibly an additional angle between the axes yield an unsymmetrical stress distribution in the rotary shaft lip type seal, thus causing higher wear on one side and increasing the risk of leakage on the other.

A pure dynamic eccentricity, also called deviation of the cyclic running, may have several causes. Bending vibrations, a shaft that does not have an exact round shape and/or if the axis of revolution of the shaft differs from the axis of symmetry (Fig. 1b) yield in a stress maximum and a stress minimum in the radial lip seal that rotate. At high rotating speeds there is an increasing risk that at the position of the stress minimum the rotary shaft lip type seal cannot follow the surface of the shaft any longer. This can result in leakage.

At TUHH, experiments are done on a test rig where the static eccentricity and the dynamic eccentricity overlap such that the eccentric axis of revolution of the shaft falls together with the axis of symmetry of the rotary shaft lip type seal (Fig. 1c), thus causing a cyclically rotating stress distribution. In the following text the term dynamic eccentricity only refers to this special case.

**Non-Linearity and Constitutive Models for Rubber**

*Non-Linearity.* FE-simulations of rubber parts are non-linear in several aspects. Since large deformations may appear, special numerical formulations and solving algorithms are required. This is taken care of by choosing the non-linear simulation mode in ABAQUS [1]. Furthermore, contact may appear between surfaces of different parts or between surfaces of the same part. Contact is highly non-linear since it either exists or not, i.e., there is no smooth transition between these two states. Contact has to be taken into account by defining contact surfaces beforehand [1]. Finally, the stress-strain relationship for rubber is non-linear. This has to be included in the constitutive model. In ABAQUS the constitutive models for rubber are a combination of a so-called hyperelastic model and a viscoelastic model [1].
**Hyperelasticity.** The term hyperelasticity summarizes several characteristics and simplifications of rubber:

1. Rubber is homogenous and has a non-linear relationship between stress and strain. It shows large deformations under relatively low external loads. For some types of rubber these deformations can be as large as 1000 % [6].
2. Rubber materials are isotropic, i.e., there is no preferred direction on the molecular level [1].
3. All deformations appear instantaneously and are fully reversible. Time-dependent effects are not taken into account.

Some researchers also include incompressibility with the term hyperelasticity [5]. However, since the FE-program ABAQUS does include a description of compressibility in all available hyperelastic constitutive models this simplification is not done at this point.

In most commercial FE-programs the mathematical description of hyperelasticity is based on the so-called Cauchy-Green-Tensor. In a principal coordinate system the Cauchy-Green-Tensor has only entries on the main diagonal and is given by:

\[
C = \begin{pmatrix}
\lambda_1^2 & 0 & 0 \\
0 & \lambda_2^2 & 0 \\
0 & 0 & \lambda_3^2 \\
\end{pmatrix},
\]

where the \( \lambda_i \) describe the lengthening in the principal directions with

\[
\lambda_i = \frac{L_i}{L_{0i}} = 1 + \varepsilon_{ti},
\]

where \( L_i \) is the deformed length, \( L_{0i} \) the original length and \( \varepsilon_{ti} \) the technical strain. The hyperelastic constitutive models are based on the so-called strain energy potential \( W \). This can be either a function of the lengthening in the principal directions, \( W=\text{W}(\lambda_1, \lambda_2, \lambda_3) \), or a function of three invariants, \( W=\text{W}(I_1, I_2, I_3) \). The invariants are defined as:

\[
I_1 = \lambda_1^2 + \lambda_2^2 + \lambda_3^2,
\]

\[
I_2 = \lambda_1^2\lambda_2^2 + \lambda_2^2\lambda_3^2 + \lambda_3^2\lambda_1^2,
\]

\[
I_3 = \lambda_1^2\lambda_2^2\lambda_3^2.
\]

Since the entries of the Cauchy-Green-Tensor depend on the chosen coordinate system most hyperelastic constitutive models make use of this three invariants.
The partial derivatives of the strain energy potential to the strains finally give the elements of the stress tensor [5]:

$$\sigma_{ij} = \frac{\partial W}{\partial \varepsilon_{ij}}$$ \hspace{1cm} (6)

Hyperelasticity is included for all FE-models that are described in this paper with the most simple description of a strain energy potential, the so-called Neo-Hooke’s law. Neo-Hooke’s law is a pure phenomenological constitutive model that is numerically stable and requires little experimental effort. Therefore, at the time being it is the preferred choice since the aim of this research is only an analysis of how dynamic eccentricities can be modeled at all and how much computational time is necessary in order to solve the FE-models. It is not expected to produce quantitatively meaningful results. However, if the FE-models are solvable and provide valuable results more complex constitutive models and finally the new model can be included with little effort.

In ABAQUS Neo-Hooke’s law is given by:

$$W = C_{10} (I_1 - 3) + \frac{1}{D_1} (J_{el} - 1)^2,$$ \hspace{1cm} (7)

where $C_{10}$ and $D_1$ are temperature-dependent material parameters that have to be determined experimentally. The second summand takes volume changes due to material compressibility and temperature effects into account. $J_{el}$ is the elastic volume ratio defined as:

$$J_{el} = \frac{I_3}{(1 + \varepsilon_{th})^3},$$ \hspace{1cm} (8)

where $\varepsilon_{th}$ is the strain due to temperature effects [1]. For a constant temperature $\varepsilon_{th}=0$, thus compressibility depends only on $D_1$ and $I_3$.

The initial shear modulus $G_0$ and the parameter $C_{10}$ are related to each other by

$$G_0 = 2C_{10},$$ \hspace{1cm} (9)

The initial bulk modulus $K_0$ and the parameter $D_1$ are related to each other by:

$$K_0 = \frac{2}{D_1},$$ \hspace{1cm} (10)
Neo-Hooke’s law describes an almost linear stress-strain relationship. Fig. 2 shows the stress-strain diagram for the type of rubber of the rotary shaft lip type seal. The test data was gathered with an uniaxial experiment. In the uniaxial case Neo-Hooke’s law predicts stress values that are too small for small given strains and stress values that are too large for high given strains. In the biaxial case it predicts stress values that are always too large. The turning point of the experimental stress-strain curve is not described.

![Graph showing stress-strain diagram](image)

Fig. 2. Neo-Hooke’s law for rubber
2 pav. Naujas Hooke'o désnis gumai

Viscoelasticity. Pure hyperelastic constitutive models assume that the behavior of rubber is independent of time. In fact, time respectively the deformation velocity does have a significant influence on the momentary values of stresses and strains.

Typical behaviors of viscoelastic materials are stress relaxation, strain retardation, and self-heating. Stress relaxation means that for a given strain stress values in the material decrease. Strain retardation means that for a constant stress the strains increase, i.e., the material starts to creep. If a cyclic load is applied to a rubber part a hysteresis curve can be observed in the stress-strain diagram that describes an energy dissipation due to internal damping resulting in a temperature increase of the part.

In ABAQUS viscoelastic constitutive laws are a systematical expansion of the hyperelastic laws [1]. Material parameters are no longer constant but time-dependent. This is done by a so-called Prony-series, a sum of several exponential functions. For Neo-Hooke’s law the material parameter $C_{10}$ is now given by:

$$C_{10}(t) = C_{10}^0 \left(1 - \sum_{k=1}^{N} g_k \left(1 - e^{-t/\tau_k}\right) \right),$$

(11)
where \( C_{i0}^{0} \) describes spontaneous material behavior at high deformation speeds, \( g_k \) includes the material behavior at low deformation speeds, \( \tau_k \) is the so-called Prony-coefficient, \( t \) is the time and \( N \) the number of Prony-terms. Analogously, the material parameter \( D_i \) is defined as:

\[
D_i(t) = D_i^0 \left/ \left(1 - \sum_{k=1}^{N} k_k \left(1 - e^{-t/\tau_k}\right)\right)\right.
\]

(12)

For all FE-models described in this paper a Prony-series with three terms is used.

In all viscoelastic material laws that are based on a Prony-series it is assumed that every material parameter has the same dependency on time. Therefore, only one experiment is necessary in order to determine \( g_k \), \( k_k \) and \( \tau_k \). Furthermore, it is presupposed that the viscoelastic and the hyperelastic part of the stress have the same dependency on a given strain. This assumption is discussed controversially in the literature since on a molecular level for each of the two parts a different mechanisms is responsible [9].

Further Simplifying Assumptions. It has to be pointed out that in addition to the simplifications mentioned above even more effects that can be observed in rubber parts under usage conditions are neglected by a combination of a hyperelastic and a viscoelastic constitutive model. The influence of temperature on the material parameters is not included as such. This is especially critical since rubber gets warmer due to internal damping. The so-called Mullins effect that refers to an initial damage in the material during the very first usage cycles is not taken into account. Neither are the rather complicated mechanisms that cause rubber friction.

Simulation of a dynamic eccentricity

Test Rig Description. Three lamps emitting cold light are installed within the housing of the test rig such that they light up the rotary shaft lip type seal. On the outside a video camera is used to record the rotary shaft lip type seal. Rotating speed and the dynamic eccentricity are adjustable. All test series are carried out in a darkened room. Therefore, if the rotary shaft lip type seal cannot follow the surface of the shaft any longer light is emitted to the outside. During a preliminary test series it was found out that with a rotating speed of 2000 min\(^{-1}\) and a dynamic eccentricity of 0.4 mm good test results could be gathered. Therefore, these values are used for simulation purposes.

3D FE-Model Neglecting Friction Effects. The geometry of the rotary shaft lip type seal and the shaft is created by making a 2D sketch for each part and rotating it 360° afterwards. The diameter of the shaft is 80 mm and the original internal diameter of the rotary shaft lip type seal is 79 mm. The rotary shaft lip type seal is modeled as a deformable body with the constitutive law described above assigned to it. The shaft is a so-called analytically rigid part. This special
formulation available in ABAQUS saves computational time since an analytically rigid part is not meshed. It is recommended to use this formulation if one part is a lot stiffer than its contact partner [1].

The initial position of both parts at the beginning of the simulation can be seen in Fig. 3a. Half of the rotary shaft lip type seal is made invisible for illustrating purposes. Note that the shaft is not meshed during simulation, even though the picture may imply the opposite. In order to shorten the necessary calculation time the rotary shaft lip type seal is drastically simplified. As can be seen in Fig. 3b it is cut off close to the membrane. The marked surfaces are assigned to a fixed boundary condition. The influence of the spring is also neglected in this FE-model.

The 3D FE-model is divided into three analysis steps. In the first step the shaft moves in negative 2-direction. This step lasts 1 s and describes the assembly of the shaft and the rotary shaft lip type seal. Due to the difference in diameters the rotary shaft lip type seal is widened during assembly. In the second step the shaft moves 0.4 mm in negative 1-direction. Since the reference point that is used to describe the rotation of the shaft is off center 0.4 mm in positive 1-direction the axis of revolution of the shaft now falls together with the axis of symmetry of the rotary shaft lip type seal. This step also lasts 1 s. The third analysis step during which the shaft rotates 360° lasts 0.03 s, thus describing exactly one revolution at a rotating speed of 2000 min⁻¹. This last step is divided into 60 increments of equal size so that evaluation of the results can be done for every 6°.

Since the rotary shaft lip type seal is a rotationally symmetrical part the mesh is also created by rotation. In ABAQUS the user first seeds the part, thus describing the desired position of nodes. Afterwards, an automatic mesh generator creates the actual mesh. The cross-section of the rotary shaft lip type seal is seeded
with approximate distances of 0.35 mm and 360 seeds on the circumference. Solid 8-node brick elements (C3D8H) are then assigned to the part.

Using this information, the mesh generator for the so-called medial axis minimize mesh transition algorithm creates a mesh with 49,680 nodes and 37,800 elements. As can be seen in Fig. 3b there are elements in the area of the membrane where one side is about twice as long as the other side. Theoretically, this might cause numerical problems. However, since neither large stresses nor large strains are expected in this area, no further mesh optimization is carried out.

The solution of the FE-model takes about 11 hours on an HP Superdome parallel computer, with an overall amount of 16 GB of data being produced. The output database is 2.6 GB large. Fig. 4a shows the smoothened von Mises stress in the rotary shaft lip type seal at the end of the assembly step. The shaft is not shown in this picture. Note, by default ABAQUS smoothen all results. However, this serves visualization purposes only and does not make the results more precise. The original results of the calculation can be seen in Fig. 4b. Between adjacent elements jumps of the stress values can be observed. According to the method of stress band plots [3] this implies that the mesh is still too coarse. However, further optimization with a finer mesh and/or the use of 20-node brick elements (C3D20H) does not seem to be practical with the available computational power. Therefore, the results of the simulation are analyzed taking into account that at best they are only of qualitative nature.

Fig. 4. Von Mises stress at the end of assembly
4 pav. Von Mises'o įtempimai surinkimo pabaigoje

Since ABAQUS visualizes the actually round shaft with several edges (Fig. 3b) one cannot see directly whether there is contact between the rotary shaft
lip type seal and the surface of the shaft or not. Therefore, the shaft is not shown in Fig. 5. Instead, Fig. 5 displays the contact stress distribution at the position $\varphi=0^\circ$ at different times during one revolution.

Fig. 5a shows the stress distribution at the very beginning of the third step. Since the shaft is moved 0.4 mm in negative 1-direction the contact stress is rather small at $\varphi=0^\circ$. As time moves forward the contact stresses grow until a maximum displacement of the lip is reached at $t=0.0150$ s (Fig. 5b). At this point of time an uneven stress distribution on the surface of the lip can be seen. This does not resemble the typical stress distribution that can be observed during experiments. Obviously, contact in the FE-model appears primarily in one line of nodes on the circumference.

![Fig. 5. Contact stress at $\varphi=0^\circ$ at different times](image)

It has to be pointed out that the stress distribution at the end of the revolution differs from the one at the beginning. The color scheme in Fig. 5c suggests that the contact stress is equal to zero in every element so that there is no contact between the rotary shaft lip type seal and the shaft. However, a manual evaluation of the stress values in the elements around the lip shows that there are still non-zero stress values. These values are smaller than at the beginning of the revolution. On one hand this signifies that the viscoelastic influence is provable and on the other hand that a gap between the rotary shaft lip type seal and the shaft is not predicted.

3D FE-Model Considering Friction Effects. The 3D FE-model considering friction effects is based on the FE-model described above. The only difference is that now on all contact surfaces forces in tangential direction are allowed. Since
with this model it shall only be tested what kind of influence friction has on calculation time and the quality of results a constant coefficient of friction with $\mu=0.2$ is assumed.

In this case, dividing the third analysis step into 60 increments of even size leads to an abortion of the calculation process since ABAQUS cannot calculate an equilibrium in the first increment of the third step. Therefore, the simulation is repeated with an automatic adjustment of the size of the increments. Now the smallest increment size is 0.000125 s for the first increment. Since the overall number of increments is smaller than 60 this FE-model requires only 7 hours of computational time and produces an amount of data of 11 GB with 1 GB for the output data base.

The evaluation of the results of the simulation shows no significant difference in comparison with the FE-model neglecting friction effects. Once again the distribution of the contact pressure at the beginning of the revolution compared with the one at the end is slightly different. Once again an influence of the viscoelastic constitutive law is shown without predicting a gap between lip and shaft. Therefore, it can be said that the additional effort in order to take friction effects into account does not yield in better results of the simulation.

Axisymmetric FE-Model. It has to be pointed out that a dynamic eccentricity can actually be described by a 3D FE-model only. Therefore, in the axisymmetric FE-model a further simplification is done. Fig. 6 shows the axisymmetric FE-model in the initial state. The rotary shaft lip type seal is described with a 2D geometry. Once again, it is cut off close to the membrane. The shaft is modeled with an analytically rigid line. During assembly the shaft moves in negative 2-direction. Thereafter, it moves in positive and negative 1-direction with a sinusoidal function with a frequency of 2000 min$^{-1}$, i.e., the rotating shaft with its eccentric axis of revolution is now replaced by a round shaft that changes its diameter.

Fig. 6. Axisymmetric FE-model of a rotary shaft lip type seal
6 pav. RSŽ darbo ašinis simetrinis FE-modelis
Since ABAQUS does not offer a function in order to create an axisymmetric FE-model from an existing rotationally symmetrical 3D FE-model, the new model has to be started from scratch. Using the same space of 0.35 mm between the seeds and axisymmetric 4-node elements (CAX4H) does not yield in a solvable FE-model. Therefore, a slightly finer mesh with evenly distributed seeds of 0.2 mm is used. This results in a mesh with 414 nodes and 356 elements.

Since the numerical effort in order to solve an axisymmetric FE-model is comparable with the effort for a 2D FE-model it is expected that the computational time is much shorter and the amount of data much smaller. In fact, the computational time is within the range of minutes and the overall amount of data is 400 MB. The output database is only 100 MB large.

Fig. 7a shows the von Mises stress in the lip and Fig. 7b the von Mises stress in the membrane at the end of the assembly step. In both cases the smoothing function is deactivated. It is obvious that the FE-model predicts jumps in the stress distribution, i.e., it requires further optimization. However, since the 3D FE-models could not be optimized due to the computational power available this axisymmetric FE-model is evaluated despite its limitations for a direct comparison.

![Fig. 7. Von Mises stress at the end of assembly](image)

The distribution of contact stresses cannot be shown in an axisymmetric model since the cross section of the part is displayed only. Therefore, Fig. 8 shows the deformed plot during the time that simulates one revolution of the shaft or, to be more precise, in this case the increase and decrease of the diameter of the shaft.
All figures show the exact same position, i.e., one can see the movement of the shaft and the lip with respect to time.

The situation displayed here can be compared with a cut in the 3D FE-models at an angle of $\phi=90^\circ$, i.e., the maximum displacement of the lip is observed at $t=0.0075$ s (Fig. 8b) and the minimum displacement at $t=0.0225$ s (Fig. 8d). In this moment a gap between the lip and the shaft appears. A manual evaluation of the contact stress in the elements close to the lip shows zero values, thus implying that in principal this FE-model is suitable for describing a dynamic eccentricity.

The simulation is now repeated with a refined mesh. This mesh is based on seeds with an approximate distance of 0.1 mm and axisymmetric 8-node elements (CAX8H). This results in a mesh consisting of 4,061 nodes and 1,278 elements. Fig. 7c and 7d shows the calculated von Mises stress at the end of the assembly step. The smoothing function is deactivated. Fig. 7c shows that the stress distribution in the lip is worse with the refined mesh. This can be explained such that the contact between the shaft and the lip takes place in one node only and the distribution of elements around this node is not suitable for describing the subsequent stress distribution. Fig. 7d shows that the stress distribution in the membrane is now even, as expected. An evaluation of the deformed plot at different times shows an appearance of a gap between the lip and the shaft that is similar to the one shown in Fig. 8.

**Fig. 8.** Deformed plot at different times

**8 pav.** Deformuotas laukas skirtingu laiku

In principle, this FE-model is suitable for describing a dynamic eccentricity, too. However, the refined mesh does not improve the results for this model.
Conclusion and Outlook

1. Both 3D FE-models show an influence of the viscoelastic material behavior, however, they do not allow any quantitatively meaningful results, i.e., a gap between the rotary shaft lip type seal and the surface of the shaft is not observed at any time. The method of stress band plots shows that the mesh needs to be improved by refining it and/or by using 20-node brick elements (C3D20H). Since the rather coarse mesh already requires a significant amount of computational power these means are not practicable.

2. A promising alternative is the axisymmetric FE-model where the eccentrically rotating shaft is replaced with a shaft that changes its diameter with respect to time. This model does predict a gap between the lip and the shaft. Once again, the method of stress band plots shows that the mesh needs to be improved. Since the computational power required is much smaller it is rather easy to optimize the mesh. However, a first try with a finer mesh and axisymmetric 8-node elements (CAX8H) does not show the expected increase in the quality of the results.

3. Another conceivable approach in order to improve the outcome of the simulation that should be tested in future research, could be an assembly step that is modified such that contact is established on the whole lip, thus describing a more realistic interaction between the two parts.

4. If it seems necessary, the influence of other constitutive models on the axisymmetric FE-model can be analyzed rather easily. Perhaps, a focus should be laid on the Prony-series such that it is adjusted and optimized for the very small time step of one revolution. Finally, the influence of the spring has to be included in the FE-model. If the model then describes the behavior of the rotary shaft lip type seal properly it can be used for the verification of the new constitutive model.

Literature


Andreas Lucko, Stefan Meyer, Volkert Martin Wollesen

RADIALINIO SANDARINIMO ŽIEDO, DIRBANČIO ANT VELENO SU EKSCENTRICITETU, ANALIZĖ BAIGTINIŲ ELEMENTŲ METODU

Reziume


ABAQUS, modeliavimas, baigtinių elementų metodas, dinaminis ekscentricitetas, radialinis sandarinimo žiedas.
А. Луцко, С. Меер, Ф. М. Волнезен

АНАЛИЗ РАДИАЛЬНОГО УПЛОТНИТЕЛЬНОГО КОЛЬЦА, РАБОТАЮЩЕГО НА ВАЛУ С ЭКСЦЕНТРИСИТЕТОМ, МЕТОДОМ КОНЕЧНЫХ ЭЛЕМЕНТОВ

Резюме

В настоящей работе проведены исследования работы радиального уплотнительного кольца при радиальном биении вала. Работа уплотнительного кольца моделирована методом конечных элементов (КЭ) программой ABAQUS. Эти условия работы вала применяны для определения основного закона работы материала кольца. Анализированы два трёхмерные модели КЭ: первый – без учета влияния сил трения, второй – с учетом сил трения. Несмотря на принятые упрощения, было необходимо применить ПК высокой производительности. Оба модели КЭ подтвердили вязкоэластичное поведение материала, однако результат полностью предположения не подтвердил. Трение на результат практически не влияет. При существующем обеспечении ПК дальнейшие исследования не являются реальными. Другим перспективным решением может быть осевая – симметрическая модель КЭ, в которой эксцентрически вращающийся вал заменен валом переменного диаметра. В данном случае требуется меньшая производительность процессора. Предварительное моделирование показало хороший результат. Этот тип моделей КЭ имеет большую перспективу.

ABAQUS, моделирование, метод конечных элементов, динамический эксцентриситет, радиально уплотнительное кольцо.