Finite-element analysis of elastic sound-proof coupling thermal state

V G Tsyss, I M Strokov and M Yu Sergaeva
Omsk State Technical University, 11, Mira ave., Omsk, 644050, Russia
e-mail: mjsrgaeaw@mail.ru

Abstract. The aim is in calculated determining of the elastic rubber-metal element thermal state of soundproof coupling ship shafting under variable influence during loads in time. Thermal coupling calculation is performed with finite element method using NX Simens software with Nastran solver. As a result of studies, the following results were obtained: - a volumetric picture of the temperature distribution over the array of the deformed coupling body is obtained; - time to reach steady-state thermal coupling mode has been determined; - dependences of maximum temperature and time to reach state on the established operation mode on rotation frequency and ambient temperature are determined. The findings prove the conclusion that usage of finite element analysis modern software can significantly speed up problem solving. Key words – soundproof coupling, thermal state, torque, finite element model.

1. Introduction
Elastic soundproof couplings are used in ship shafting of diesel generator sets, or propeller drives on ships. During operation, the couplings are subjected to repeated dynamic loading and undergo significant dissipative heating in some cases resulting to their premature failure. In this connection, the practical task is the problem of thermal coupling calculation, its feature is that energy dissipation leads to self-heating of the coupling body rubber under dynamic loading and it must be considered when assessing its efficiency.

2. The problem setting
Elastic coupling in the process of operation except for a constant load in the form of torque is also subject to time-varying dynamic loads. Dynamic loads acting on the coupling can be divided into two types depending on the causes of their occurrence. The loads connected with angular and radial displacement of half-couplings are considered the first. The nature of their variation is harmonic, and the frequency is equal to the coupling rotation frequency. The shaft rotation unevenness is considered the second, the moment inconsistency of the coupling loaded is connected. They include:

Uneven torque on the shaft of the power plant.
Uneven torque on the shaft of the actuator.

The problem is to estimate the thermal state of a coupling loaded with a harmonic moment

\[ M(t) = M_1 + M_2 \sin(\omega t) \]  \hspace{1cm} (1)
where $M_1$ is permanent component of the moment; $M_2$ is periodic component of the moment; $\omega$ is angular frequency of the torque change.

The following initial load data was taken for calculation:

$$
M_1 = 0 \text{ N}\mu;
M_2 = 1 \text{ kN}\mu;
\omega = 30 \text{ } \chi^{-1}
$$

Thermal coupling calculation was performed with finite element method using the NX Siemens software with Nastran solver.

3. Theory

Heat generation in a unit volume of the rubber elastic element is proportional to the mean for the deformation cycle of the variable voltages power value [1]:

$$
Q_\psi = \frac{\omega}{2\pi} \int_0^{2\pi/\omega} (\sigma_x \dot{\varepsilon}_x + \sigma_y \dot{\varepsilon}_y + \sigma_z \dot{\varepsilon}_z + \tau_{xy} \dot{\gamma}_{xy} + \tau_{xz} \dot{\gamma}_{xz} + \tau_{yz} \dot{\gamma}_{yz}) \, dt,
$$

where $\omega$ is angular frequency; $\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{xz}, \tau_{yz}$ are axial and tangential stresses; $\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{xz}, \gamma_{yz}$ are axial and shear deformations.

In the case the rubber follows a linear viscoelastic deformation law described, for example, with the generalized Maxwell model, the function $Q_\psi$ can be expressed in terms of the energy dissipation coefficient $\psi$ [1]:

$$
Q_\psi = \frac{\omega \psi}{2\pi} G_0 \left( 2 \varepsilon_{ax}^2 + 2 \varepsilon_{ay}^2 + 2 \varepsilon_{az}^2 + \gamma_{axy}^2 + \gamma_{axz}^2 + \gamma_{ayz}^2 \right).
$$

where $\varepsilon_{ax}, \varepsilon_{ay}, \varepsilon_{az}, \gamma_{axy}, \gamma_{axz}, \gamma_{ayz}$ are amplitude values of axial and shear deformations.

Experimentally, the dissipation coefficient is determined with the decrement of free vibrations, the amplitude of the resonant oscillations or oscillatory process phase pattern observed on the oscilloscope screen (Fig. 1). In the latter case, the dissipation coefficient is numerically equal to the ratio of the hysteresis loop square area to the area of OAB triangle:

$$
\psi = \frac{A_D}{A_E}
$$

where $A_D$ is energy dissipated as heat for deformation cycle; $A_E$ is potential energy of deformation (work of elastic forces).
It follows from equation (4), that if the material dissipation coefficient $\psi$ is known and elastic strain energy $A_E$ has been determined then it is possible to determine the energy dissipated in the form of heat per deformation cycle $A_D$:

$$A_D = \psi A_E.$$  \hspace{1cm} (5)

Then, knowing the rotation frequency, the amount of heat generated in the material per unit of time can be calculated

$$Q = f A_D$$  \hspace{1cm} (6)

where $f$ is deformation frequency.

In the end, having made the heat balance equation, it is possible to determine the thermal state of the coupling [3]:

$$([L] + [C_h])T = Q$$  \hspace{1cm} (7)

where $[L]$ is thermal conductivity matrix of the structure; $[C_h]$ is heat exchange matrix; $\{Q\}$ is heat generation vector

4. The calculation results

In modern finite element analysis software it is possible to obtain a volumetric picture of deformation energy distribution along a deformed body. Some software such as NX (Siemens), allow extracting the numerical values of the nodes (elements) depending on the nodes (elements) coordinates.

The values obtained can be recalculated using formulas (5) and (6) in the amount of heat released per unit volume of the body per time unit.

The next step is to solve the non-stationary thermal problem, where obtained values are used as the load, like heat generation per volume unit.

I stage. Simulating

To solve the problem in NX it was created finite element coupling model (Fig. 2).
To create the model 8-node hexagonal elements CHEXA (8) were used. Since this model is used both in structural and thermal analysis, then both structural and thermal characteristics of materials must be specified for it. For steel elements the material "Steel" from NX materials database was taken, and all the required characteristics are set. For elastic rubber elements it is necessary to develop a material and mechanical and thermal characteristics were indicated for it. Since the static problem is solved in a linear formulation, it is sufficient to take the value of Young's modulus $E=3$ MPa and the Poisson's coefficient is $\nu=0.49$. Thermal characteristics for rubber are [2]:

- thermal expansion coefficient is $\alpha=15 \cdot 10^{-4}$ K$^{-1}$;
- thermal conductivity is $\Lambda=0.22$ W/(m K);
- specific heat is $C=1.67$ kJ/(kg K).

**Figure 2.** Finite element model of the rubber-metal coupling element

**II stage. The solution of the static problem**

To find the elastic deformation energy a structural analysis of the coupling model was performed. The solution type is “SOL 101 Linear statics”.

To use the obtained result for the thermal coupling calculation, it is necessary to obtain an array of deformation energy density numerical values with elements depending on its coordinates. So the option "Determine value" is used, where a grid of elastic coupling element is chosen, and the extracted results are stored in electronic Excel table format as shown in Fig. 3.

Till to start thermal calculation it is necessary to prepare the initial data in a spreadsheet, i.e. to recalculate the energy density of deformation having been obtained in the previous stage with formulas (5) and (6) into the amount of heat released per volume unit per time unit. To do this the file saved in the previous stage is opened.
To start with, unnecessary information is removed, because the desired file must contain only four data columns with headings - three columns of coordinates (x, y, z) and one column with values of heat generated amount. Therefore, all redundant rows and columns are deleted from the file - three columns are obtained with the coordinates (A, B, C) and column with values of the deformation energy density (E). The column E is recalculated into column (F) depending on

$$\Theta = \phi \cdot \psi \cdot A_E,$$

Where \( f = \omega 2 \pi = 30/6,24 = 4,8 \ \text{Гц} \) is frequency of deformation, \( \psi = 0.31 \) is coefficient of dissipation, \( A_E \) is energy of elastic deformation (from the Table). It is written in the Excel editor for the second cell:

$$F_2 = E \cdot 0.31 \cdot 4.8$$

and values are stretched for the entire column F. As a result, the column F will have values of the heat amount evolved per time unit. Values of column F should be moved into column D, and the columns E and F are removed. The resulting file is saved for later use.

III stage. The solution of the heat state problem

To determine the thermal coupling state a new solution based on the solver NX THERMAL/FLOW is developed, type of analysis is thermal.

Solver settings are as follows (unspecified settings remain by default):

Type of solution is transition process;

Transition process settings are:

The start time is 0 s;

The end is to perform until stationary mode;

The maximum number of intervals is 10;

To finish solution if the temperatures change between the time intervals is less than 1о C;

The units of the solution are left by default;
Ambient conditions are left by default;
Parameters of results are left by default, but if needed they can be added;
Initial conditions:
The initial temperature is uniform;
Specifying conditions for solving the problem are:
a) specification of the conditions between the contacting surfaces.

Conditions of contact between the rubber and the metal are set using simulation object "Thermal connection" (Fig. 4). As settings, the interacting surfaces of rubber and metal are indicated as "Primary area" and "Secondary area". Further the interaction type is indicated as "Heat transfer coefficient" and "Coefficient" 1150 W/(m²·°C). Two such objects are developed for each half coupling.

![Figure 4. Setting the thermal contact between rubber and metal](image)

To set the heat exchange conditions with the discarded parts, "Boundary resistance" simulation tool is used. The area of application will be the surface interacting with the discarded part of the structure - in this case with the mounting shaft flange (Fig. 5). Interaction type is "Heat transfer coefficient", equal to 2000 W/(m²·°C). The same condition is developed on the second connecting surface.
b) specification of boundary conditions.

As boundary conditions, convective heat exchange with the surrounding environment is specified. To do this "Convection to Environment" tool is used. As surfaces dissipating heat (only the outer surfaces of the metal and rubber) are indicated (Fig. 6). Convection coefficient is considered 22 W/(m²°C).

c) load specification.

The thermal load is set by means of "Thermal loads" tool. The distribution method is "Spatial." The load application will be the elastic coupling element. The load field is set using the table designer, and where the Table having been prepared in the previous calculation with values of heat amount released by the material volume unit is substituted.

After all the above settings, the task is started for calculation.

When completing the calculation, the solver gives a series of results, and the most interesting is the self-heating temperature of the rubber coupling array among them. The picture of thermal coupling state into the steady thermal mode is presented in Fig. 7 in a color chart form. The Figure shows that the array of coupling rubber during operation is heated to temperature of 31.1°C. The most heated is the central (internal) part of the rubber mass, and it is a physically adequate result. The temperature
field is shifted to the inner part of the coupling, since there is no convective heat exchange with the environment.

![Temperature Distribution](image)

**Figure 7.** The color chart of the temperature distribution

The dependence of the temperature change in the most heated point of the rubber coupling mass in the process of output to the steady thermal mode is given in Fig. 8.

The Figure shows that the elastic coupling reaches the established thermal mode in 17000 s, and it is equal to 4.7 hours of operation.

![Temperature Change](image)

**Figure 8.** Graph of temperature change during the time of reaching the steady-state thermal mode.

The effect of various operational parameters on the thermal state of the soundproof coupling is determined.

**Effect of the rotation speed**

An analysis of the thermal coupling state for different rotation speeds is completed and the dependence of the maximum temperature inside the rubber of the coupling on the rotation speed is developed. The graph of this dependence is given in Fig. 9.
The time change in reaching the steady-state thermal mode of the coupling operation is shown in Fig. 10.

**Impact of changes in external temperature.**

The thermal coupling state is defined at various ambient temperatures: from 20°C till 30°C. To do this, calculations under various environmental conditions are performed. According to the results, graphs of coupling self-heating temperature dependences are plotted (Fig. 11) and time to reach steady-state thermal mode (Fig. 12) on the ambient temperature.
5. Results and discussions

The studies proved that the rubber coupling body during operation is heated to temperature of 31.1°C. The most heated is central (inner) part of rubber and the temperature field is shifted to the inside part of the coupling as there is no convective heat transfer with environment from the inner side.

The picture of soundproof coupling thermal state in the steady state mode is given in Fig. 7 in a color chart form. The dependence of changes in temperature in the most heated point of the rubber coupling in the process of reaching steady state thermal mode is given in Fig. 8. The Figure shows that the soundproof coupling reaches the steady thermal mode for 17000 seconds, and it is equal to 4.7 hours of operation.

6. Conclusion

As a result of the carried out researches the thermal calculation of an elastic soundproof coupling made of rubber was made. Based on the results of the calculations, the following results were obtained:

Fields of temperature distribution over the coupling.

Time to reach the steady-state thermal mode of operation.

Dependences of maximum temperature and time of reaching to a mode on coupling rotation speed and on ambient temperature.
The obtained results allow drawing a conclusion that application of modern finite element analysis software allows to speed up the solution of the problems with a sufficient degree of accuracy.

References
[1] Mihailov Y K and Ivanov B S 1987 Couplings with nonmetallic elastic elements: theory and calculation (Leningrad: Mechanical Engineering) p 145
[2] Starastenko I N 2007 Calculation of heat in the pre-strained rubber cylinder with harmonic loading KubGAU//Scientific journal 27 p 3
[3] Goncharov P S et al 2012 NX Advanced Simulation. Engineering analysis (Moscow: DMK Press) p 504
[4] Mancuso Jon R 1999 Couplings and joints. Design, selection, and application (New York: Marcel Dekker) p 581
[5] Francis Alex B 2014 Experimental, Numerical and Analytical Characterization of Torsional Disk Coupling Systems (Theses and Dissertations) p 625
[6] Reddy J N 2004 An Introduction to Nonlinear Finite Element Analysis (USA: Oxford University Press)
[7] Logan D L 2012 A First Course in the Finite Element Method, 5th ed (USA: Cengage Learning)
[8] Tsyss V G, Strokov I M and Sergaeva M Y 2015 The issues of life extension of seismic isolation system of circular tanks for storage of liquefied petroleum gas Procedia Engineering vol 113 pp 395–401
[9] Tsyss V G, Strokov I M and Sergaeva M Y 2015 Dampener resource of seismic isolation absorber system of circular tanks for liquid hydrocarbons storage Procedia Engineering vol 113 pp 402–407.
[10] Burkardt J, Du Q, Gunzburger M and Lee H.-C 2003 Reduced order modeling of complex systems (NA03 Dundee) pp 29–38