CALCULATION OF THE HEAT-STRESSED STATE OF THE DISK USING FREE SOFTWARE CODE_ASTER

Cherednichenko A.V.1, Maksimova E.A.2, Savelyeva I.Yu.3

1 Candidate of Engineering Sciences, Associate Professor. Bauman MSTU.
2 student. Bauman MSTU.
3 Candidate of Physico-Mathematical Sciences, Associate Professor. Bauman MSTU.
elizaveta.alekseevna.m@gmail.com

Annotation: A numerical calculation of the axisymmetric heat-stress state of the turbine disk was performed using the free Code_Aster software. Distributions of temperature, radial displacements, and also radial and axial stresses in the cross section of the disk are presented.

Key words: heat-stressed state, free software, Code_Aster, disk, finite elements method.

Introduction
An important stage in the design of transport and stationary power plants and many other machines in various industries is the calculation of temperature and stress fields in structural elements operating at high temperatures [1]. For example, special requirements are imposed on disks of turbines of engines. Moreover, in stationary modes, the temperature and stress fields remain at a constant, but rather high level, which leads to the accumulation of damage [1, 2].

All over the world, the finite element method (FEM) has been successfully used for calculating the heat-stressed state of structural elements. One of the options for calculating mechanical engineering components, using the FEM, is the usage of ready-made libraries of finite element programs written in compiled high-level programming languages, for example deal.II. This approach involves writing your own code to implement a specific version of the FEM in the language of C++ using ready-made tested subroutines with its further compilation and launch.

An alternative to the previous approach is the usage of software finite element systems. There are a large number of commercial systems: ANSYS, Abaqus, MSC.Nastran, LS-Dyna, and free software (Code_Aster, CalculiX, Elmer, etc.). The usage of free software for calculating the heat-stressed state of structural elements is relevant, because, firstly, dependency on owners of commercial licenses is eliminated, secondly, software costs are reduced and, thirdly, the possibilities of adapting the code for solving specific problems are opened. However, the choice of a free software package requires a large amount of validation test calculations.

This paper presents the construction of the temperature and stress fields for the test problem of a thick-walled pipe, as well as for a turbine disk using the free software package Code_Aster (acronym for Analysis of Structures and Thermomechanics for Studies and Research solving strength and thermomechanics problems for scientific purposes and education) [3, 4].
1. Temperature stresses in a thick-walled pipe

The classical problem of determining thermal stresses in a thick-walled pipe in a state of stationary uneven heating is considered. The pipe inner radius is \( a = 0.1 \) m, and outer radius is \( b = 0.25 \) m. The temperature on the inner surface is \( T_1 = 543 \) K, on the outside \( T_2 = 393 \) K. Pipe material is a non-hardened steel, the following parameter values were taken in the calculations: \( \alpha(T) = 1.76 \cdot 10^{-5} \) K\(^{-1} \), \( E = 1.96 \cdot 10^{11} \) Pa, \( \nu = 0.3 \).

The formulation of the associated stationary thermomechanical problem in the region \( V \), corresponding to a homogeneous body of isotropic material has the form [2]:

\[
\Delta T(M) = 0, \quad \nabla \cdot \sigma(M) = 0, \quad M \in V,
\]

\[
\sigma = \frac{E}{1 + \nu} \left( \dot{\varepsilon} + \frac{\nu}{1 - 2\nu} \varepsilon I - \alpha(T) (T(M) - T_0) I \right), \quad \dot{\varepsilon} = \left( \nabla u + (\nabla u)^T \right)/2,
\]

where \( T(M) \) is a temperature; \( \sigma \) and \( \dot{\varepsilon} \) are stress and strain tensors, respectively, \( u \) is a displacement vector; \( \varepsilon = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} \); \( I \) is a unit tensor of the second rank; \( E \) is a Young’s modulus, \( \nu \) is a Poisson’s ratio, and \( \alpha(T) \) is a coefficient of thermal deformation.

The boundary conditions for such a problem have the form:

\[
T|_{S_1} = f_1(x), \quad q|_{S_2} = f_2(x), \quad \sigma|_{S_3} = p(x), \quad u|_{S_4} = g(x),
\]

where \( S_1 \cup S_2 = \partial V, S_3 \cup S_4 = \partial V, S_1 \cap S_2 = \emptyset, S_3 \cap S_4 = \emptyset, q \) is a heat flux density vector; \( \sigma \) is a stress vector; \( f_1(x), f_2(x), p(x), g(x) \) are known function and vector functions.

The geometric model of the pipe (Fig. 1) was built in the Geometry module of the Salome-Meca. The mesh was created in the MESH module, using quadrangular quadratic elements with a side of 0.01 m.

![Figure 1](image1.png)  
**Figure 1.** Geometric model, section of a pipe.  

![Figure 2](image2.png)  
**Figure 2.** Temperature field.  

![Figure 3](image3.png)  
**Figure 3.** Radial displacement field.

Modelling of the problem is carried out in the AsterStudy module. To calculate the temperature field, the previously constructed grid is read \( \text{LIRE_MAILLAGE} \), and by using the \( \text{AFFE_MODELE} \) command the modelling type is set. The thermal conductivity problem is defined by the keyword \text{THERMIQUE}. For the problem, which is symmetric with respect to the axis, the \text{MODELISATION} parameter takes the value \text{AXIS}. The concept of the material is determined by setting the thermal conductivity coefficient \( \lambda \) and the product \( \rho C_p \):

\[
\text{DEFI_MATERIAU} > \text{LAMBDA} = 27.0, \text{RHO}_C \text{P} = 5250000.0.
\]

The indicated material properties are applied everywhere. The boundary conditions are defined by the command \( \text{AFFE_CHAR_THER} \),
on the high and low bounds a zero flux (FLUX_REP) is set, and on the left and right borders of pipe — the temperatures (TEMP_IMPO).

At the second stage of modeling the grid is re-read to calculate the mechanics problem, which is defined on the AFFE_MODELE tab: PHENOMENE > MECANIQUE, MODELISATION > AXIS. Then the temperature field is projected. This option requires setting the following parameters: RESULTAT is the concept of the result created at the first stage, MODELE_1 is the model of the projected temperature field, MODELE_2 is the model on which the projection is performed.

The third stage is to model the problem of mechanics. The mechanical properties of the material DEFIMATERIAU > ALPHA, E, NU, RHO, are determined everywhere, the control temperature value AFFE_MATERIAU > AFFE_VARC > VALE_REF = T_{ref}, at which there are no deformations caused by temperature, is set (in this case, the value T_{ref} = 446 K). The DDL_IMPO function, responsible for the number of degrees of freedom, fixes the movement of the high and low bounds along the Oy axis. Further, taking into account previously created material concepts and boundary conditions, the analysis type is determined — MECA_STATIQUE. Using the CALC_CHAM function, the temperature stresses in the mesh nodes SIGM_NOEU are calculated.

Figure 4 shows the radial, axial and hoop stresses in the cross section of the pipe.

![Figure 4](image-url)
The symmetry of a rigid body with respect to the axis \( O_x \) makes it possible to construct only the upper half of the section. The result in .iges format is imported into the Salome-Meca. The MESH module allows to build a mesh using two-dimensional triangular quadratic elements (Fig. 8). The recommended element size for this geometry is 2-3 mm [6].

The geometric model of the disc is built using AutoCAD’s computer-aided design and drafting system. The radial section of the disk is shown in Fig. 7 [6].

### Table 1. Values of stresses obtained analytically and by using Code_Aster and ANSYS.

| Coordinates | Analytical solution | Solution using Code_Aster | Solution using ANSYS |
|-------------|---------------------|---------------------------|----------------------|
|             | SIXX                | SIYY | SIXX | SIYY | SIXX | SIYY |
| (0.1, 0)   | 0                   | 4.769 \cdot 10^8 | 1.886 \cdot 10^6 | 4.776 \cdot 10^8 | 1.586 \cdot 10^6 | 4.773 \cdot 10^8 |
| (0.12, 0)  | -6.004 \cdot 10^7  | -3.298 \cdot 10^8 | -6.192 \cdot 10^7 | -3.304 \cdot 10^8 | -6.178 \cdot 10^7 | -3.303 \cdot 10^8 |
| (0.14, 0)  | -7.984 \cdot 10^7  | -2.053 \cdot 10^8 | -8.034 \cdot 10^7 | -2.06 \cdot 10^8 | -8.025 \cdot 10^7 | -2.059 \cdot 10^8 |
| (0.16, 0)  | -7.859 \cdot 10^7  | -9.753 \cdot 10^7 | -7.886 \cdot 10^7 | -9.829 \cdot 10^7 | -7.880 \cdot 10^7 | -9.824 \cdot 10^7 |
| (0.18, 0)  | -6.715 \cdot 10^7  | -2.45 \cdot 10^8 | -6.73 \cdot 10^7 | -3.254 \cdot 10^8 | -6.726 \cdot 10^7 | -3.210 \cdot 10^8 |
| (0.2, 0)   | -5.044 \cdot 10^7  | 8.261 \cdot 10^7  | -5.053 \cdot 10^7 | 8.178 \cdot 10^7  | -5.049 \cdot 10^7 | 8.181 \cdot 10^7  |
| (0.22, 0)  | -3.108 \cdot 10^7  | 1.596 \cdot 10^8  | -3.113 \cdot 10^7 | 1.587 \cdot 10^8  | -3.110 \cdot 10^8 | 1.587 \cdot 10^8  |
| (0.24, 0)  | -1.048 \cdot 10^7  | 2.298 \cdot 10^8  | -1.051 \cdot 10^7 | 2.289 \cdot 10^8  | -1.049 \cdot 10^7 | 2.289 \cdot 10^8  |
| (0.25, 0)  | 0                   | 2.627 \cdot 10^8  | -22555.3 | 2.619 \cdot 10^8 | -5002.9 | 2.619 \cdot 10^8 |

2. Calculation of the heat-stressed state of the disk

The steel disk of a turbine (Fig. 6) [5] located in a stationary temperature field is considered. The radial section of the disk is shown in Fig. 7 [6].

![Figure 6](image6.png)  
**Figure 6.** Turbine stage diagram: 1 — nozzle diaphragm, 2 — turbine rotor blade, 3 — shaft, 4 — disk.

![Figure 7](image7.png)  
**Figure 7.** Radial section of a disk (the sizes are given in millimeters).

The following values were taken as characteristics of the disk material [6]: \( \lambda = 20 \text{W/m} \cdot \text{K}, \rho = 7800 \text{kg/m}^3, C_p = 462 \text{J/kg} \cdot \text{K}, \nu = 0.3, \alpha(T) = 1.1 \cdot 10^{-5} \text{K}^{-1} \). According to the Table 2 the
corresponding values of temperature and Young’s modulus are set. The dependence is shown in Fig. 9.

| Temperature $T$, K | Young’s modulus $E(T)$, P$\cdot$D$\cdot$\degree |
|--------------------|-----------------|
| 473                | $2 \cdot 10^{11}$ |
| 673                | $1.8 \cdot 10^{11}$ |
| 773                | $1.65 \cdot 10^{11}$ |

The problem is axisymmetric, therefore, it can be considered as two-dimensional. The simulation type AXIS corresponds to this problem both in the case of the thermal conductivity problem and during the calculation of the heat-stressed state [4]. The simulation is carried out in the Code_Aster module in three stages: calculation of the temperature field in the disk, projection of the temperature field and calculation of the heat-stressed state of the disk. In the calculation, six-node quadratic finite elements were used [7, 8].

To calculate the stationary temperature field, the parameters $\lambda$ and $\rho C_p$ are set, and the boundary conditions are imposed: on the right boundary of the geometric model $T_1 = 773$ K, on the left $T_2 = 573$ K. Previously, the first stage of modelling is generated, the mesh is read and the type of model is determined. Due to the symmetry of the disk with respect to the axis Ox, a zero flux $q = 0$ is set at the lower boundary of the model.

In addition to the thermal loads on the disk there are also power loads: inertial — rotation around the axis Oy with an angular velocity $\omega = \pi n / 30$, where $n$ is the number of disk revolutions per minute (in the calculations $n = 11200$ rpm), as well as tensile stresses equal to $\sigma = 200$ MPa [6].

The simulation of the problem of determining the stress-strain state begins with reading the mesh constructed earlier. The problem of mechanics is directly solved at the third stage of modelling. Using constant functions, the characteristics of the material $\rho$, $\nu$, $\alpha(T)$ and the boundary conditions $\omega$, $\sigma$ are specified; the dependence of Young’s modulus on temperature is determined. The concept of the result of projecting the temperature field is added to the definition of material: AFFE_VARC > EVOL > NOM_VARC = TEMP. The boundary conditions are taken into account using the function AFFE_CHAR_MECA. The ROTATION command sets inertial boundary conditions and the pressure PRES_REP > PRES = $\sigma$ is applied to the right side. The DDL.IMPO function fixes the movements of the lower bound along the Oy axis, then the type of analysis is determined and the parameters for calculating the temperature stresses in the grid nodes are set.

Figures 10-12 below show the distribution of temperature, radial displacements, and also radial and axial stresses in the cross section of the disk.
Figure 10. Temperature distribution over the cross section of the disk.

Figure 11. Field of radial displacements in the cross section of the disk.

Figure 12. Radial (a) and axial (b) stresses in the cross section of the disk.

Figure 13. Von Mises stresses in the cross section of the disk.

Figure 14. Von Mises stress versus disk radius (section $y = 0.003$).
Conclusion
Using free software, the stationary problem of thermoelasticity was solved, and the heat-stressed state of the turbine disk was calculated. The distributions of temperature, radial displacements, and also radial and axial stresses in the cross section of the disk are given.

Acknowledgements
The research was supported by the Russian Foundation for Basic Research, project No. 18-38-20108.

References
[1] Termoprochnost’ detalей mashin [Heat resistance of machine parts]. Ed. I.A. Birger, B.F. Shor. Moscow, Mashinostroyeniye Publ., 1975. 456 p.
[2] Raschet teplonapryazennykh konstruktsiy [Calculation of heat-stressed structures]. Zarubin V. S., Stankevich I. V. Moscow, Mashinostroyeniye Publ., 2005. 352 p.
[3] Code_Aster. Analyse des structures et Thermo-Mecanique pour des Etudes et des Recherches. Available at: http://code-aster.org , accessed 24.09.2019.
[4] Aubry J.-P. Beginning with Code_Aster: A practical introduction to finite element method using Code_Aster, Gmsih and Salome. P.: Framasoft, 2013. 357 p.
[5] Vvedenie v teploenergetiku: Ucheb. posobiye [Introduction to Thermal Engineering: A Training Manual]. Polishchuk I.Z. Ufa, Ufa State Aviation Technical University Publ., 2003. – 105 p.
[6] Resheniye praktikiykh zadach termomekhaniki s primeneniyem programnogo kompleksa ANSYS: Metod. ukazaniya k vypolneniyu laboratornykh rabot [Solving applied problems of thermomechanics using the ANSYS software package: Guidelines for laboratory work]. Krasnovskiy E.E. Ed. Zarubin V. S. Moscow, Bauman MSTU Publ., 2008. 88 p.
[7] Zienkiewicz O.C., Taylor R.L. The finite element method. The basis. Butterworth Heinemann. 2000. Vol 1. 708 p.
[8] K.J. Bathe Finite Element Procedures. Prentice Hall, Pearson Education, 2014. 1065 p.