



FINITE ELEMENT METHOD

AEF-A.2.10

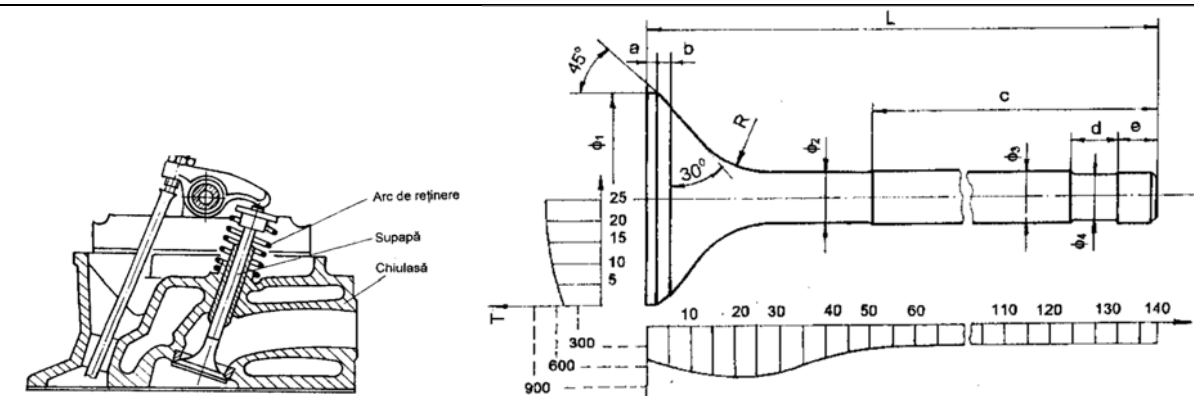
LINEAR STATIC ANALYSIS OF THERMALY LOADED STRUCTURES

INTRODUCTION

In many practical applications, designers of mechanical systems have to take into account, in addition to functional demands, the considerable effects of the existence of thermal fields. Thermal stress that occurs in a piece during operation or in a technological process may be superior to those caused by internal or external mechanical stresses, sometimes leading to damage to the workpiece. Thermal stress can occur frequently in internal combustion engine components, compressors, blowers, boilers, heat exchangers, or in thermal semi-finished products. The study of the thermal phenomena with implications on the mechanical fields in the structural elements of the mechanical systems with complex geometric shapes and various loads using performance analysis programs with finite elements is, in many practical situations, the only solution for determining the resulting fields.

DESCRIPTION OF THE APPLICATION

In the operation of internal combustion engines, simultaneously with mechanical stress, thermal stresses arise as a result of both the heat transfer from the gases to the parts adjacent to the combustion chamber and the friction of the moving parts in relative motion. From these parts, in this application, the valve is analyzed by a distribution mechanism located in the cylinder head. During operation, the valve plate is subjected to the pressure of the gases, the elastic force of the retaining spring in the closed position and the dynamic loads. Additionally, the valve operates in the presence of high temperature with variable distribution in the adjacent space. Mechanical and, in particular, thermal stresses cause elastic deformations of the roller and valve stem, leading to the loss of tightness in the conical contact area with the valve seat and the grip of the stem in the translation guide. For the valve construction, taking into account the special conditions imposed on the material (low coefficient of thermal expansion, good thermal conductivity and increased thermal resistance), the best behavior is the alloy steels with the main elements chrome and nickel.



PURPOSE OF THE APPLICATION

In this application it is intended to determine deformations and stresses in the valve structure generated by the operating temperature fields. The figure above shows the shape, dimensions and temperature variations in the axial and radial direction of the manifold valve shown above. The dimensions of the dimensional parameters are $L = 151,2$ mm, $a = 2,3$ mm, $b = 2$ mm, $c = 80$ mm, $d = 15$ mm, $e = 10$ mm, $R = 15$ mm, $\phi_1 = 50$ mm, $\phi_3 = 11$ mm, $\phi_4 = 10$ mm. The valve is made of alloy steel 34MoCrNi15.

Application FEA-A.2.10

LINEAR STATIC ANALYSIS OF THERMALLY LOADED STRUCTURES

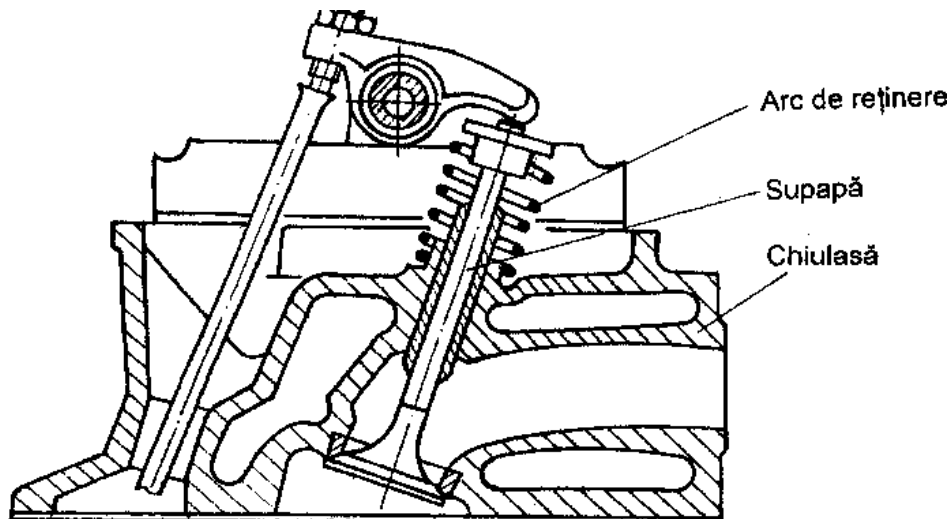
A. DEFINITION OF THE APPLICATION

A.1. Introduction

In many practical applications, designers of mechanical systems must take into account, in addition to functional demands, the considerable effects of the existence of thermal fields. Thermal stress that occurs in a part during operation or within a technological process can exceed those caused by internal or external mechanical stresses, sometimes leading to damage of the part. The occurrence of thermal stresses is frequently found in components of internal combustion engines, compressors, blowers, boilers, heat exchangers, or semi-finished products subjected to heat treatments. The study of thermal phenomena with implications for mechanical fields in structural elements of mechanical systems with complex geometric shapes and various loads, using high-performance finite element analysis programs, is, in many practical situations, the only solution for determining the resulting fields.

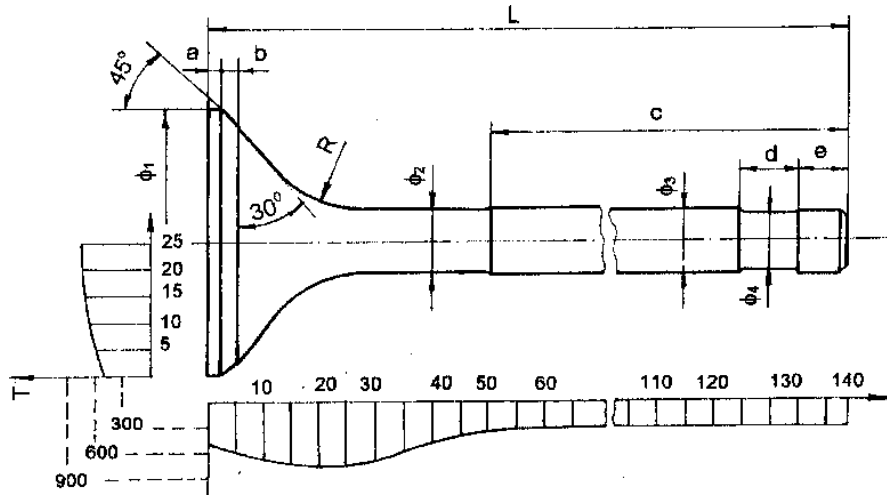
A.2. Description of the application

In the operation of internal combustion engines, simultaneously with mechanical stresses, thermal stresses arise as a result of both heat transfer from gases to parts adjacent to the combustion chamber and friction of moving parts in relative motion. Among these parts, this application analyzes the valve from a distribution mechanism located in the cylinder head. During operation, the valve head is subjected to gas pressure force, the elastic force of the retaining spring in the closed position, and dynamic loads. Additionally, the valve operates in the presence of high temperatures with variable distribution in the adjacent space. Mechanical and, especially, thermal stresses cause elastic deformations of the valve head and stem, leading to a loss of tightness in the conical contact area with the valve seat and the seizing of the stem in the translation guide [1]. For valve construction, taking into account the special conditions imposed on the material (low coefficient of thermal expansion, good thermal conductivity, and increased thermal resistance), alloy steels with chromium and nickel as main elements exhibit the best behavior.



A.3. Purpose of the application

The analysis in this application aims to determine the deformations and stresses in the valve structure generated by temperature fields during operation. The figure below presents the shape, dimensions, and temperature variations in the axial and radial directions of the distribution mechanism valve presented above. The values of the dimensional parameters are $L = 151,2$ mm, $a = 2,3$ mm, $b = 2$ mm, $c = 80$ mm, $d = 15$ mm, $e = 10$ mm, $R = 15$ mm, $\phi_1 = 50$ mm, $\phi_2 = 10$ mm, $\phi_3 = 11$ mm, $\phi_4 = 10$ mm. The valve is made of an alloy steel 34MoCrNi15.



B. PREPARATION OF THE ANALYSIS MODEL

B.1. Defining the analysis model

To prepare the finite element analysis model associated with the application above, the following must be identified:

- Geometric shape and dimensions.
- Restrictions induced by connections with adjacent elements.
- Isostatic restrictions.
- External loads of temperature type.
- Material characteristics.

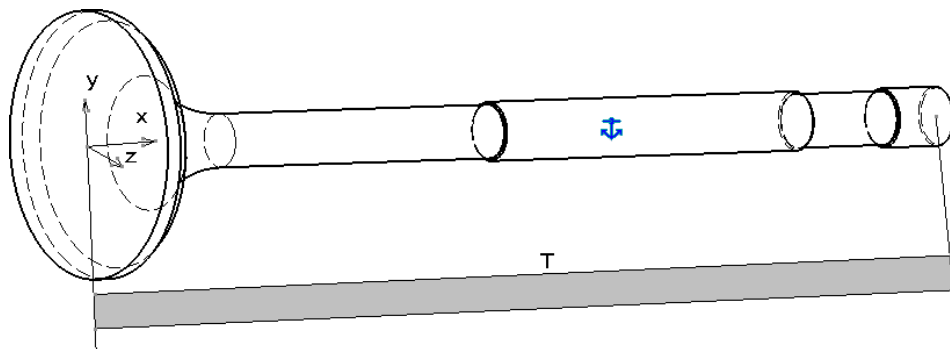
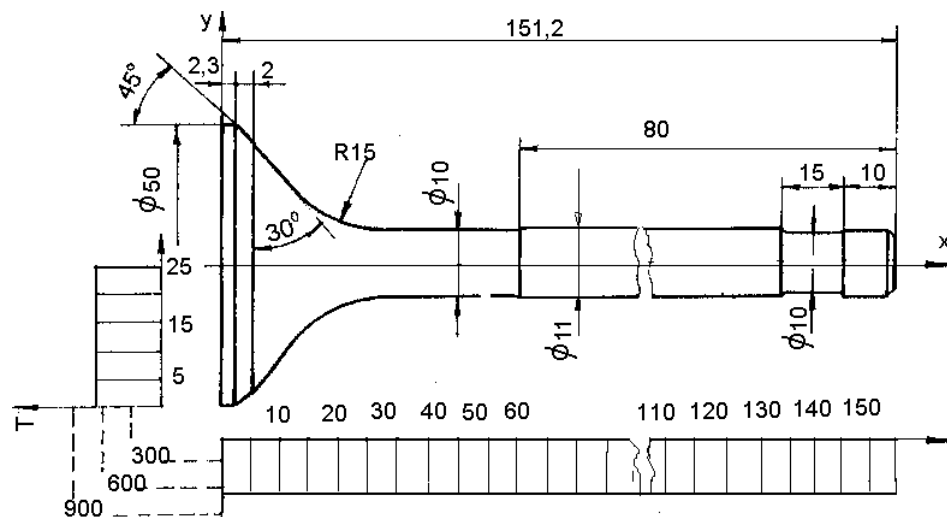
B.2. Description of the analysis model

The geometric shape and dimensions of the geometric model are identical to those of the valve structure and are presented in the figure below.

This application aims to highlight the displacement and stress fields solely as an effect of the thermal field developed during the operation of the internal combustion engine. In this sense, fields due to pressure in the combustion chamber and the retaining spring force are neglected.

To ensure the analysis model behaves like the real model, appropriate boundary conditions must be associated; in this case, displacements in the axial direction of the points on the frontal surface of the valve head are canceled. Additionally, to avoid free displacements of the analysis model, an isostatic restriction is associated with it.

During operation, the studied valve is under the influence of a temperature field with an unknown distribution inside the geometric domain. Determining the values of this temperature field will be done starting from the temperature distributions in the outer zones presented in the figure above. For simplification, the temperature distributions on the outer surface of the valve, for the purpose of introduction as loads, are approximated with a constant plateau corresponding to the temperature $T = 620$ degrees Kelvin.



B.3. Establishing material characteristics

For the finite element analysis, the physical characteristics of the material (34MoCrNi15 steel) are:

- Density $\rho = 7,8 \times 10^{-6} \text{ kg/mm}^3$,
- Thermal conductivity $c = 0,35964 \text{ w/mmK}$
- Average coefficient of thermal expansion $\alpha = 2,2 \times 10^{-5} \text{ K}^{-1}$,
- Specific heat $C_p = 5,1 \times 10^{-4} \text{ J/KgK}$,
- Longitudinal modulus of elasticity (Young's Modulus) $E = 2,15 \times 10^5 \text{ MPa}$,
- Transverse contraction coefficient (Poisson's ratio) $\nu = 0,3$.

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling

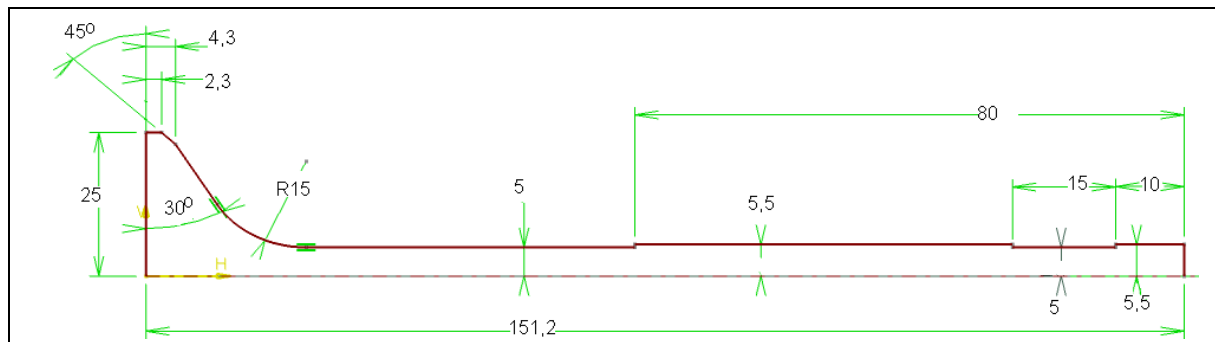
Activating the solid generation module and setting the unit of measurement for lengths

Start → **Mechanical Design** → **Part Design** → **New part**: New part name: Supapa.

Tools → **Options ...** → **Options**: Parameters and Measure; Units; Length, Milimeter (mm);
 ↵ OK.

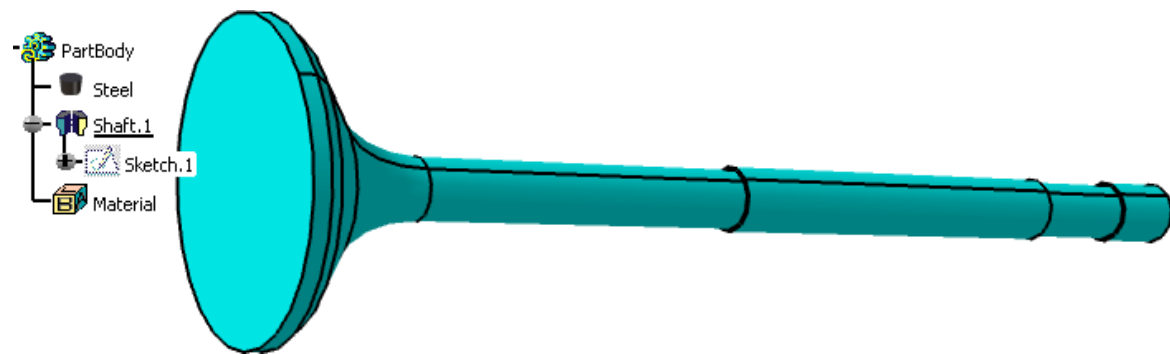
Generating the reference sketch

(Sketcher) → xy plane → (Profile) → [draw the closed contour of the axial semi-section] → (Constraint) [enter dimensions successively by selecting entities, using the Ctrl key for multiple selection] → (Exit Workbench).



Generating the solid model

→ **Shaft Definition:** First angle 360deg, Profile Selection [select Sketch.1],
 ↵OK.



Generating chamfers

[Select edges, using Ctrl command] → (Chamfer) → **Chamfer Definition:** Length: 0,25 mm; Angle: 45 deg; Object(s) to chamfer: 4 Edges, ↵OK.



C.2. Modeling material characteristics

[Select the PartBody specification in the tree structure] → (Apply Material) → **Library (ReadOnly):** Metal, Steel → Apply Material, ↵OK.

[Select Steel from the tree structure] → **Properties:** Feature Properties, Feature Name: OLC45; Analysis, Young Modulus 204000 N/mm², Poisson Ratio 0,3, Density: $\rho = 7,8 \times 10^{-6}$ kg/mm³, Thermal Expansion: $2,2 \cdot 10^{-5} \text{ K}^{-1}$ → Apply, ↵OK.

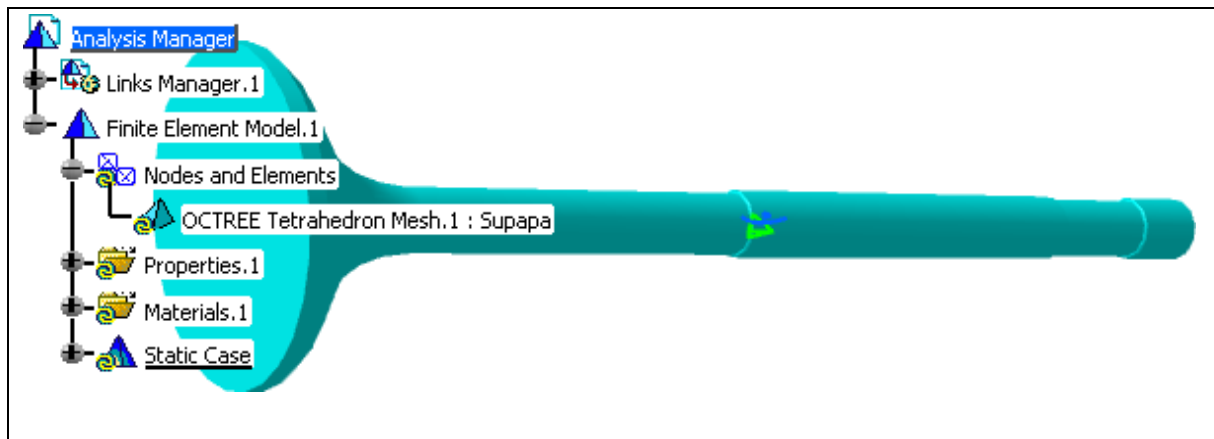


C.3. Finite element modeling

Activating the finite element analysis module

Start → Analysis & Simulation → Generative Structural Analysis → **New Analysis Case:** Static Analysis, ↵OK.

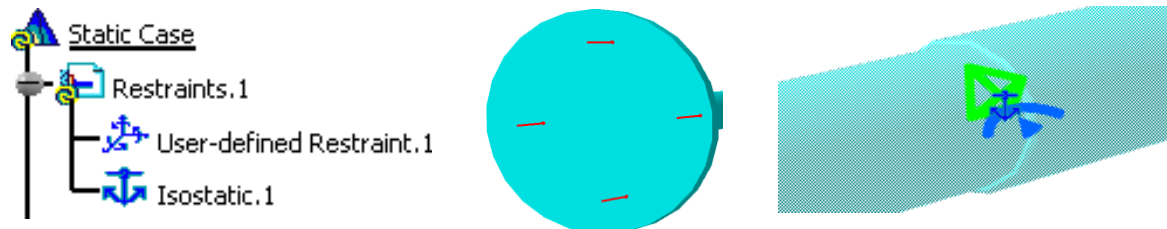
[Select with the mouse the OCTREE Tetrahedron Mesh specification in the tree structure, or the symbol on the solid model structure] → **OCTREE Tetrahedron Mesh:** Size: 4 mm, Absolute Sag: 1, Element type: ☒ Parabolic ↵OK.



C.4. Modeling geometric constraints

Introducing displacement and isostatic restrictions

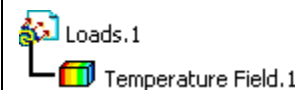
→ **User-defined Restraint**: Supports [select the frontal surface of the valve head], Activate ☐ Restrain Translation 1, ↵OK.
 (Isostatic) → **Isostatic Restraint**: Name: Isostatic.1, ↵OK.



C.5. Modeling loads

Introducing the temperature

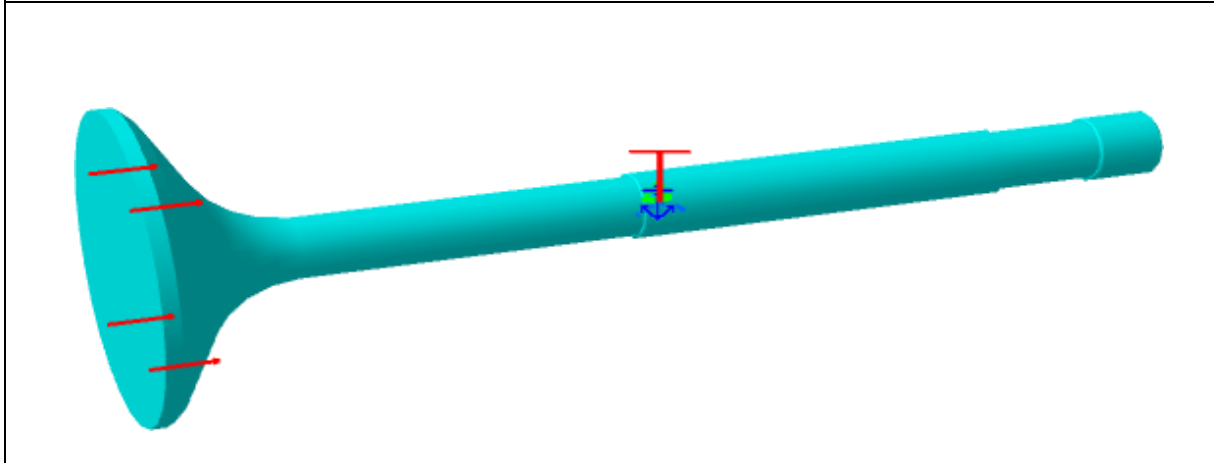
(Temperature Field) → **Temperature Field**: Support: [select the valve body], Temperature: 620Kdeg, ☐ Data Mapping, ☐ OK.



D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the calculation module

(Compute) → **Compute**: ↵ All, ↵OK → **Computation Resources Estimation**, ↵Yes
 → **Computation Status**.



E. POST-PROCESSING OF RESULTS

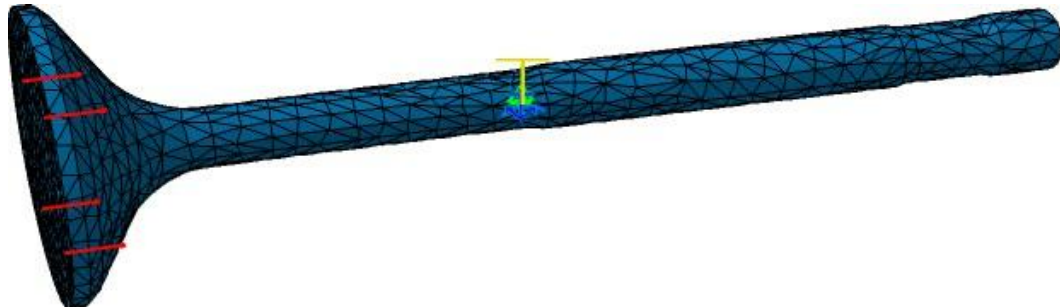
E.1. Visualizing the deformed state / Animation



(Deformation).



(Animate) → **Animate**: Steps number: 5 (other parameters can be modified), ↵Close.



Observation

To modify the amplitude of displacements during animation for better visualization, follow the sequence

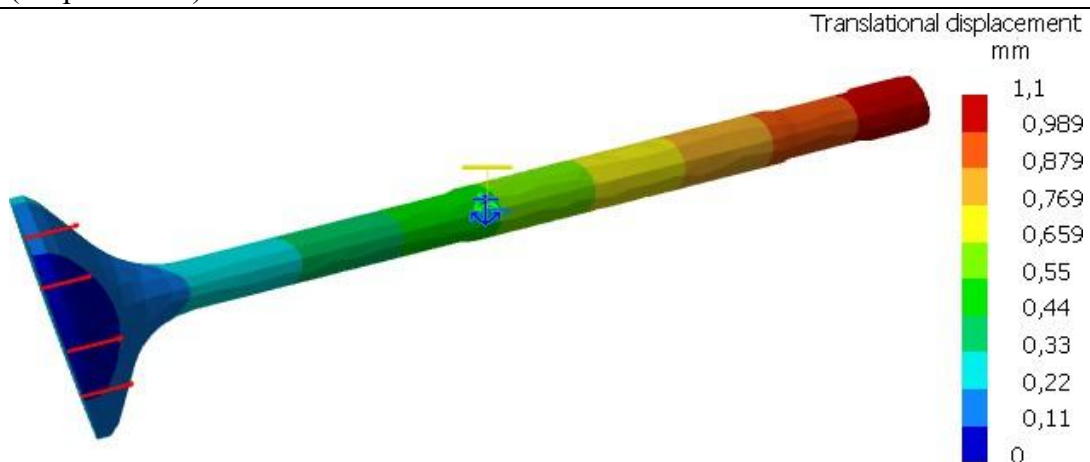


→ **Amplification magnitude**: Factor: 1 (other parameters can be set), ↵OK.

E.2. Visualizing the displacement field



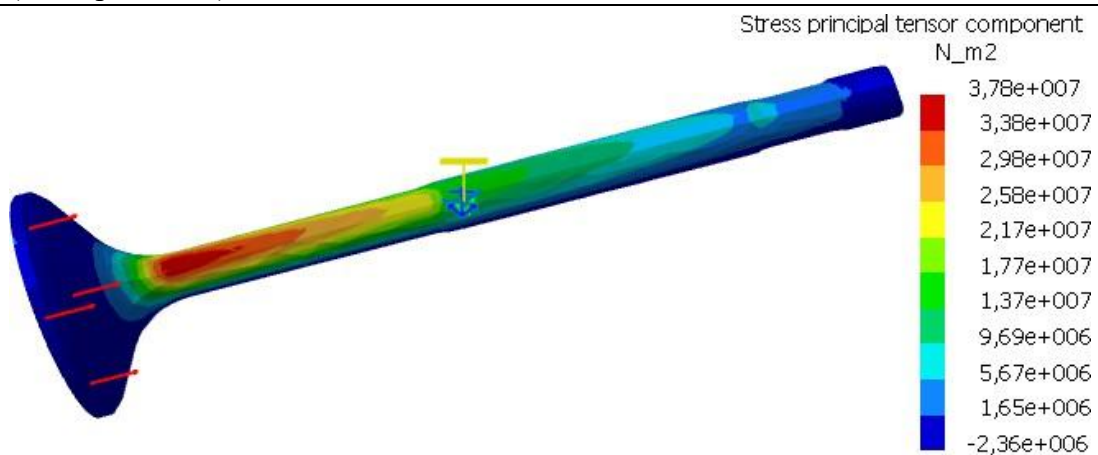
(Displacement).



E.3 Visualizing stress fields

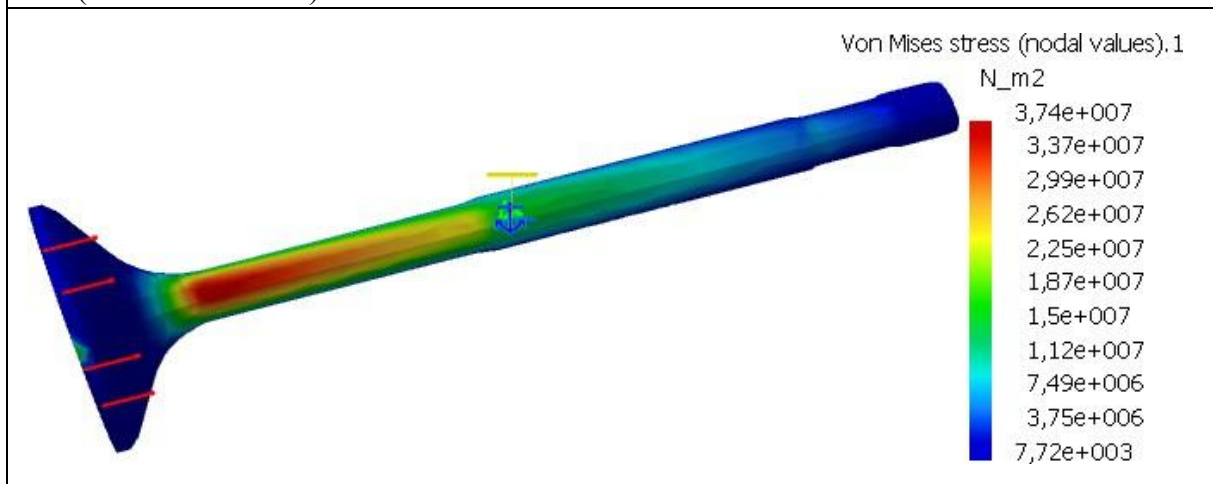


(Principal Stress).





(Von Mises Stress).



F. CONCLUSIONS

The study of thermal and mechanical fields in parts of mechanical systems operating at high and variable temperatures, using high-performance programs based on FEM, allows the designer to optimize their shape as well as accurately determine adjustment values.

In the case of the analyzed valve, the maximum displacement of 1.1 mm, in the area of the stem head, implies the value of the rocker arm adjustment during assembly. On the other hand, increased stresses are highlighted in the fillet area connecting the head to the valve stem, justified by the quasi-abrupt change in section.

From the analysis of displacements, the values of the stem expansion in the radial direction can be highlighted, an aspect that aids in the dimensional design of the valve guide bushings.

G. EXERCISE

Perform the finite element analysis of the shaft in the bearing assembly below, in order to determine the adjustment clearance j in the case of a temperature increase of 300 Kelvin.

The shaft is made of OLC 45 steel, improved to 260-280 HB, and has the following dimensions of the cylindrical sections (diameter, length), expressed in mm: (20, 50), (25, 11), (30, 10), (50, 350), (30, 10), (25, 14). The bearings used are single-row radial ball bearings, series 6005.

