STUDY OF THERMAL STRESSES OCCURRING IN THE VALVE OF A DISTRIBUTION MECHANISM IN THE COMPONENT INTERNAL COMBUSTION ENGINES

Ioan GHIMBASEANU

Transilvania University of Braşov, Romania (ghimbasani@unitbv.ro)

DOI: 10.19062/1842-9238.2018.16.1.10

Abstract: *The aim of the paper is to create interactive programmes for monitoring the thermal stresses obtained after the analysis with finite elements for the valve of a distribution mechanism in the component internal combustion engines. The method can also be applied for monitoring the thermal stress nodal of aircrafts, rockets, ballistic missiles and gun barrels.*

Keywords: *thermal stress, combustion engines, valve, finite elements.*

1. INTRODUCTION

In the operation of internal combustion engines simultaneously with the mechanical stresses, thermal stresses occur as a result of the transfer of heat from the gases to the adjacent parts of the combustion chamber **.**

In this paper we study the thermal stress of the valve in the distribution mechanism. During operation, the valve plate is subjected to the pressure of the gas. The valve operates under high temperature conditions. Mechanical and thermal stress causes elastic deformations of the plate and valve stem, leading to loss of sealing in the tapered contact area and rod grid. Thermal stresses that occur in the work piece during operation may be superior to mechanical stress, leading to damage to the work piece.

In this application, the field of displacements and stresses is emphasized as the sole effect of the thermal field. It is neglected the fields due to the pressure in the combustion chamber and the force of the holding spring. The temperature distribution on the outside surface of the valve is approximated by a constant field corresponding to the temperature T = 620 degrees Kelvin. For the valve execution, taking into account the conditions imposed on the material (low thermal expansion coefficient, good thermal conductivity), the best behavior has the alloy steel with nickel and chromium. The occurrence of thermal stresses in internal combustion engine elements may frequently occur.

2. STATICAL ANALYSIS WITH FINITE ELEMENTS FOR MONITORING THE THERMAL STRESSES

2.1. Analysis model processing

In order to draw up the analysis model with finite elements associated with the application, it is necessary to identify: shape and geometric dimensions, restrictions imposed by adjacent connections, external temperature loads, material characteristics.

Geometrical modeling. The geometric shape and dimensions of the geometric model of the valve are shown in fig. 1 and fig. 2[1,2,6,7,8].

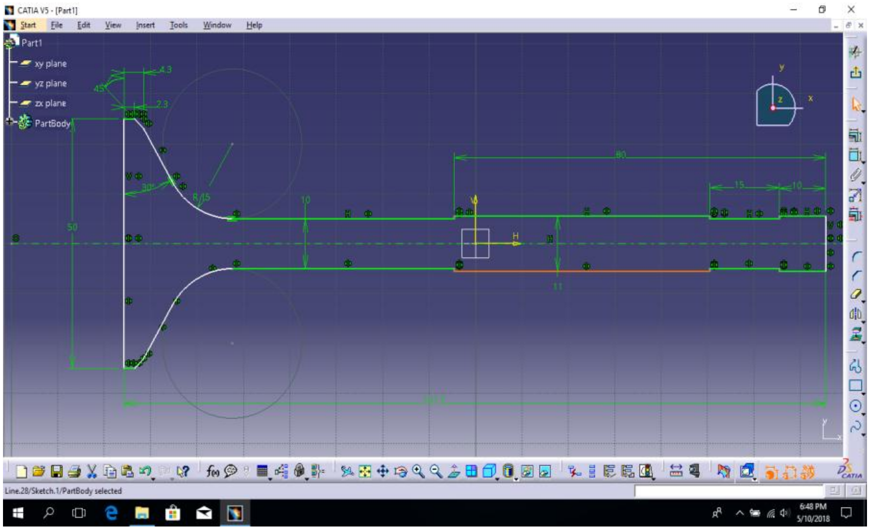

FIG. 1. The model 2d of valve

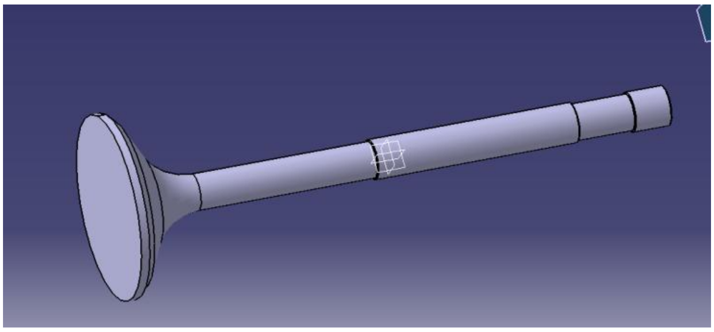

FIG. 2. The model 3d of valve

Modeling the characteristic of the material. The introduction of the values of the material characteristic necessary for the finite element analysis is made through using the CATIA programmer`s library of materials.

The steel material is selected[3,4,9].

2.2 Finite element modeling

CATIA Analysis & Simulation packed is launched for generating the finite element.

This packed makes the static analysis of the structure when some constraints are imposed and when some stress is independent-time.

Modeling of geometrical constrictions. Movement restrictions and isostaticity are introduced. These are shown in Fig. 3 [5].

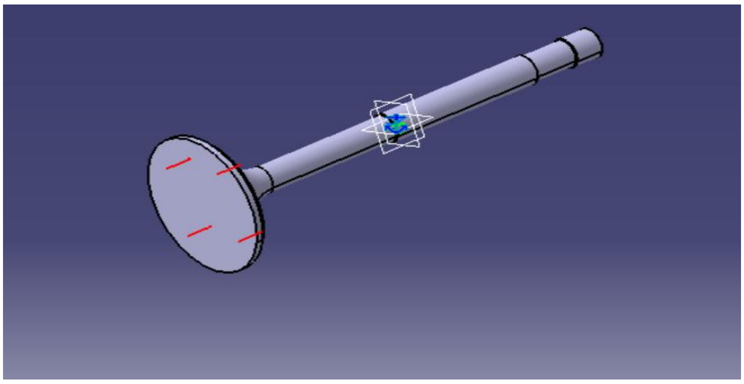

FIG. 3. User’s defined restraint and isostatic restraint

Load modeling. The temperature distribution on the outside surface of the valve is approximated by a constant field corresponding to the temperature T = 620 degrees Kelvin.

Solving the model and post processing the result. Then the calculation model is launched.

Fig. 4 shows the deformation, Fig. 5 shows the displacement and Fig. 6 shows the von Mises stress.

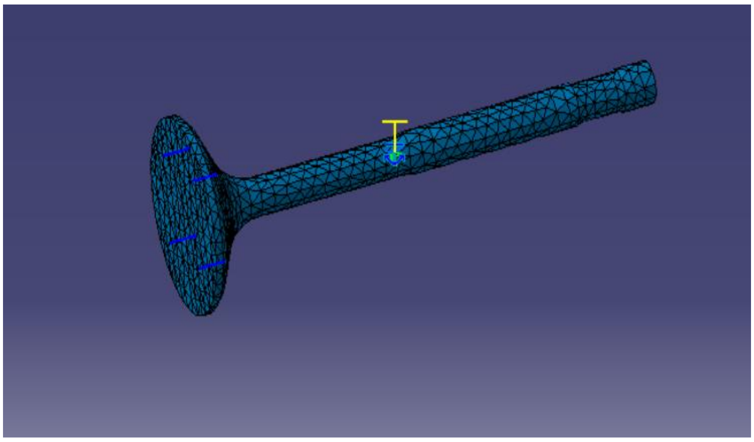

FIG. 4 Deformation of valve

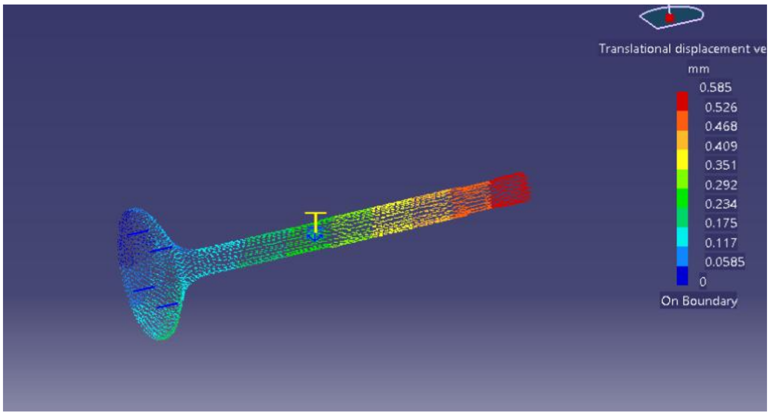

FIG. 5. Displacement of valve

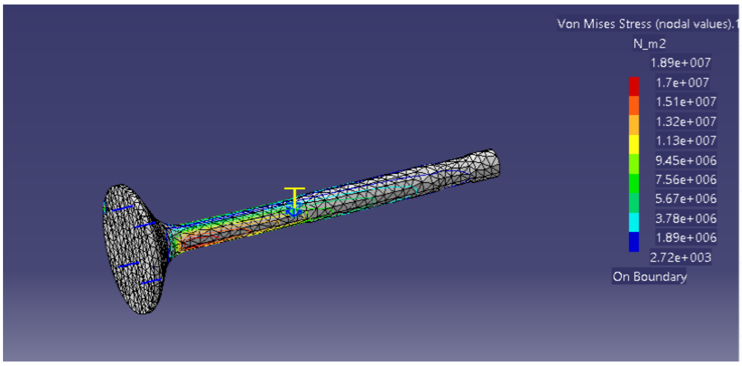

FIG. 6. Von Mises Stress

Some values of mechanical stresses in the valve are shown in Table 1.

Table 1. Value of Von Mises Stress

X[MM]	Y[MM]	Z[MM]	T[N/M2]
-52.41	4.38	2.39	18.89E+006
-52,41	-4.38	-2.39	18.6789E+006
-73.46	21.93	11.98	1.6689E+006
-75.76	21.93	11.98	17.489E+006
62.63	-4.38	-2.39	1.9589E+006
-48.32	4.38	2.39	17.8 E+006
-44.22	4.38	2.39	17.32 E+006
-52.41	-4.99	-0.05	18.1 E+006
-42.93	-4.38	-2.39	17.2 E+006
-38.93	-4.38	-2.39	16.7 E+006
-36.16	-4.38	-2.39	16.22 E+006
44.08	2.42	4.37	12.66 E+006
-36.08	2.43	4.36	11.8 E+006
-32.16	3	4	12.8 E+006
-27.98	2.34	4.41	10.77 E+006
-24.16	3	4	11.91 E+006
-20.07	2.42	4.37	10.13 E+006
-16.16	3	4	10.95 E+006

3. CONCLUSIONS

When performing programs based on MEF, one can solve a practical problem for the analysis of mechanical and thermal fields from solid media.

Performance programs using MEF allows the study of mechanical fields in high temperature parts. A good program based on MEF can also be appreciated by the types of materials that can be selected. Programs based on MEF have meshing possibilities based on a geometric model.

The efficiency of preprocessing programs is also due to the possibilities of introducing limit conditions and loads. In the case of the analyzed valve, the maximum displacement of 0.6mm requires adjustment to the installation.

Table 1 shows high values of mechanical stresses in the connection area of the plate with the valve stem due to the change of section.

The study of thermal phenomena in the mechanical systems structures using finite element analysis programs is the solution for determining the resultant fields.

REFERENCES

[1] Ghionea I., *Desing in CATIA v5,* Bren Publisching House,, Bucharest, 2007;

[2] Ghionea I., *CATIA v5 Aplicatii in ingineria mecanica,* Bren Publisching House,, Bucharest, 2009;

[3]Zecheru Gh., Drăghici Gh., *Elemente de ştiinţe şi ingineria materialelor*, vol. 1 şi 2, Editura Universităţii din Ploieşti, 2001*;*

[4] Lates T., *Metoda elementelor finite, Aplicatii,* Editura Universitatii Transilvania, Brasov, 2008;

[5] Mogan G., Butnariu S., *Finite element analysis in engineering,* Brasov, Transilvania University of Brasov Publishing, ISBN 978-973-4, 2007;

[6] Ghimbaseanu I., *Comparative method for determing the mechanical stress,* Review of the Air Force Academy, Vol.XIII, nr.1(28), 2015, pag 105-108;

[7] Ghimbaseanu I., Monitor the simulation of *mechanical stresses by computer,* Review of the Air Force Academy, Vol.XIV, nr.1(31), 2016, pag 105-110;

[8] Ghimbaseanu I., *Simulation of mechanical stresses of a assembly connecting rod,* Review of the Air Force Academy, Vol.XV, nr.1(33), 2017, pag 105-108;

[9] Metal testing, *tensil test,* STAS SR EN 10002-1:2002.