ANNALS OF FACULTY ENGINEERING HUNEDOARA
- INTERNATIONAL JOURNAL OF ENGINEERING
Tome IX (Year 2011). Fascicule 3. (ISSN 1584 - 2673)

[1.] Camelia PINCA -BRETOTEAN

THE FINITE ELEMENT ANALYSIS OF A CYLINDER HEAD OF A SPARK IGNITION ENGINE

[1.] UNIVERSITY POLITEHNICA OF TIMISOARA, FACULTY OF ENGINEERING FROM HUNEDOARA, ROMANIA

ABSTRACT: The finite element method is applied to find the stresses field from the parts of a cylinder head of a spark ignition engine. There are thus identified the most stressed component elements and respective regions. This paper aims to study the stresses on an engine cylinder head spark ignition, Dacia 1,6L, which is known constructive characteristics and the thermal parameters. Values obtained by numerical calculation using the CATIA software will be compared with the analytical and the experimentally determined.
KEYWORDS: Cylinder head, ignition engine, internal, stresses, mesh

❖ INTRODUCTION

Design work of modern heat engines requires accurate determination of stresses and strains, in order to optimize the correlations between size, shape and properties of materials used for structural parts of a machine component, on the one hand, and the thermo-mechanical applications, on the other, [5]. Knowledge demands precision in stresses and strains are increasing continuously with increasing indices of power, economy and specific gravity, which involves raising the main components of motor vehicle claims, [2], [5].

Determination by calculation of the stresses and deformations present serious difficulties because of the complexity and constructive forms of load conditions. Classical methods of calculating the resistance of these machine parts are justified in most of cases a highly simplified representation of geometric shape and load and run only the benchmark against statistical data from the literature.

Recorded remarkable progress in recent years in methodology and technique, especially by introducing the finite element method, allowed solving problems of interest, such as determining the stresses and strains in an internal combustion engine cylinder head. Calculation accuracy can be ensured by this method is high, although it remains subject to the mode, and knowing that outline conditions, [2], [5].

This paper aims to study the stresses in an engine cylinder head spark ignition, Dacia1.6L, which is known constructive characteristics and the thermal parameters.

The necesity of stress field estimation is for optimum design imposed at a large scale level the numerical methods, [1]. These methods proved to be an extremly efficient but discrimination should be considered when using them. It is recomanded that the numerical results of a certain problem to be compared, when possible, to the analytical result when the last exist, or to the numerical model to a simplified problem which presents a known analylical solution, [1].

❖ SOME CONCEPT ABOUT FINITE ELEMENT METHOD

The finite element method is one of the most used methods that are available in our days for different calculations in the field of engineering. This method and the programs based on it become fundamental components in the computer aided design systems. They are indispensable in all engineering activities where high performance is required, [4], [5].

One of the major advantages in the finite element method is the simplicity of its basic concepts. It is very important that the finite element method user learn and correctly understand these concepts, because they include certain hypotheses, simplifications and generalizations, [4], [5].

To perform a finite element analysis, the user must develop a calculus model of the analyzed pieces. These models are only approximate mathematical models of the pieces. There are no algorithms and general methods for developing a unique model that approximate, with a known error, the real structure of the analyzed piece. The development of a model is based on the user's intuition, experience and imagination. The model should efficiently synthesize all the available information about the analyzed piece, [3].

A model consists of lines, planes or curved surface and volumes, created in a 3D CAD environment. In this stage of development, the model is continuous, with an infinite number of points, as the real pieces that is analyzed. The main goal of the finite element method is to obtain the finite element mesh, transforming the continuous structure into a discreet model, with a finite number of points, [4].

This operation is done using a mesh for the model, which is correct from an engineering point of view, the knowledge of stresses and displacements in a certain number of points inside the piece is normally enough to characterize the mechanical and thermical behavior of the piece.

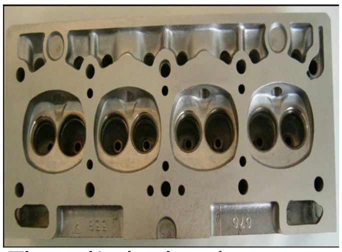
The finite element method defines these unknowns only in the nodes of the model and calculates their values in these points. That's why the meshing process must be performed in such a way as to have a number of nodes large enough in the areas of great interest in order to achieve a satisfactory approximation for the geometry of the piece and for the boundary and loading conditions. Thus, the mesh of the piece has a major importance in the finite element method analysis. The points defined in the mesh are called nodes. The primary unknowns of finite element method are defined in nodes, and their values are the analysis results. These unknowns identified can be displacements, displacement model or stresses, stress model, [4]. For the displacement model, it can be admitted that the deformed shape of the piece under a certain loading case, is defined by the displacements of all the nodes with respect to the initial node net. Each node may have a maximum of six components of the displacement, called nodal displacements in a coordinate system: three linear displacements and three rotations.

The meshing process divides the model into a certain number of finite elements. These elements are assembled together in common nodes. The finite element study as a single piece in interaction with the other elements only in nodes. Thus, the study of the real piece is replaced with the study of the ensemble of finite elements obtained by meshing, in an idealization of the real piece which is analyzed. For better results, the process should be adequate to the purpose of the analysis, implying the respect for some important rules regarding the meshing process and the elaboration of the model, and also to use adequate finite elements, [1], [4], [5]. The mesh of a analyzed piece can include elements defined for different types of analysis, as: linear elastic, nonlinear, heat transfer, fluid mechanics, electromagnetism, etc.

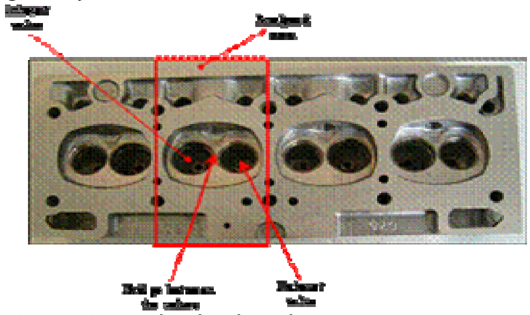
In the finite element method practice, the role of the material's characteristics is very important and in this case the material attached to the finite element can be homogeneous, isotropic or with a certain anisotropy. Each finite element is an ensemble of conditions and hypotheses and should be used with care only after a complete study of the environment where is functioning the real analysed piece: loadings, stress type, interaction with other elements, etc.

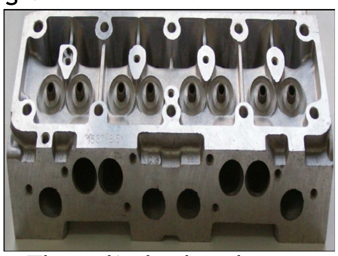
❖ RESULTS

Figure 1 presents the engine cylinder head Dacia will be modeled using Catia software.



The cylinder heads, on top





The cylinder heads, on bottom

Fig.1. 1,6L Dacia engine cylinder head

For spark-ignition engine, 1.6 L Dacia is known constructive characteristics and the thermal parameters. Modeling was performed for the gas pressure p_g = 3 N/mm^2 and temperature T_g = 1900 K were determined by diagrams of the engine indicated in the nominal regime. The material properties are: Young modulus ($7 \cdot 10^{10}$ N/m^2), Poisson ratio (0,346), density (2710 kg/m^3), thermal expansion ($2{,}36 \cdot 10^{-5}$ K) and the yield strength ($9{,}5\ 10^7$ N/m^2). The solid model of the cylinder head is presented in fig.2.

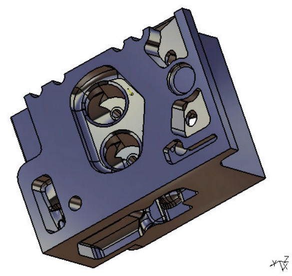


Fig.2. The solid model of a portion of the cylinder head spark ignition engine, Dacia 1.6 L, which includes an intake valve and an exhaust valve

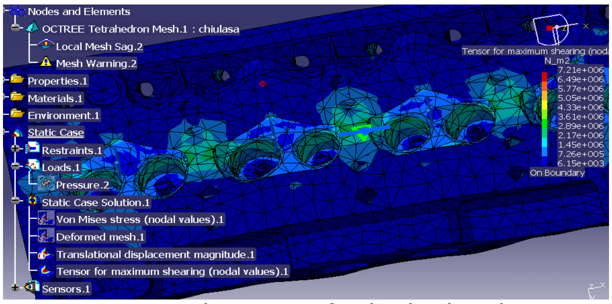
The state of stresses analysis of cylinder head will be taken into account both mechanical and thermal loads. They are valued as follows: distributed mechanical load (internal or external pressure, own weight); mechanical loads focused on small areas (load from the mass of an element over another) and loads due to differential thermal expansion caused by temperature variation from one point to another on the same piece (the thickness, length or the diameter). Mechanical loads are used in modeling are the forces and moments whose values are presented in table 1.

Table 1. The values of forces and moments used in modeling

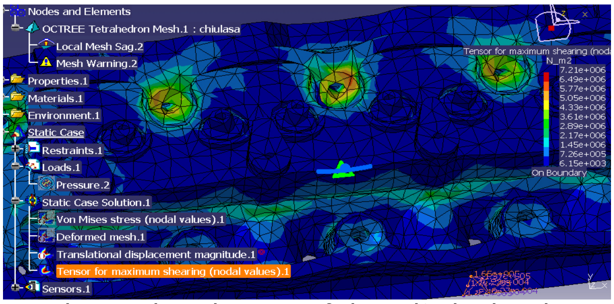
Mechanical components		Values	Reactions [N]
Loads [N]	F_x	- 4150	4150
	F_y	968,91	- 968,91
	F_z	2662,1	- 2662,1
Moments [N·mm]	M_x	5,349	- 5,3496
	M_y	- 853,12	853,12
	M_z	310,51	- 310,51

Note that in modeling or considered maximum values of mechanical loads and thermal loads in regard themselves as middle values. In this case the maximum pressure is p_g = 7,8 N/mm^2 and temperature is T_g = 2200 K

In the stage of the meshing of the cylinder head have been used a number of 55539 finite elements with a number of 13208 nodes. In fig.3 is presented cylinder head on the top, examined from the finite element mesh, fig.3a and on bottom fig.3b. In fig.4 shows how to impose constraints across the cylinder heads examined. In Fig.5 is presented modeling thermal field due to engine ignition operation.



a - mesh on top of cylinder head



b - mesh on bottom of the cylinder head

Fig.3 The finite element mesh across the cylinder heads

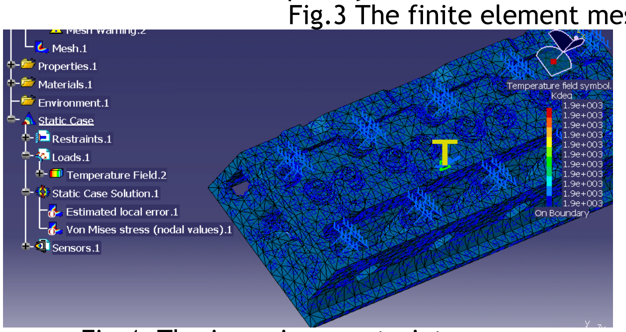


Fig.4. The imposing constraints across the cylinder heads

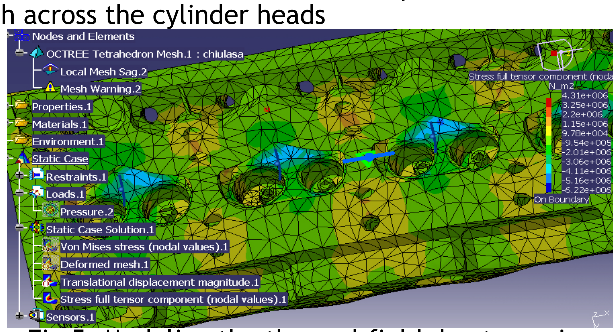


Fig.5. Modeling the thermal field due to engine ignition operation across the cylinder heads

Complete stress and deformation tensors, displacements and contact pressure are available as results of structural analysis. All these results give us new knowledge about loads on parts results of structural analysis. Their interpretation is general and very complex, mainly due to the uncertain influence of model simplification and approximations, [3].

Table 2 shows the maximum and minimum principal stresses obtained from modeling cylinder head, the nodes are recorded these values and the position of those nodes to the chosen reference system modeling.

Table 2. The maximum and minimum principal stresses

Principal stresses		Values [N/mm^2]	Node	Node coordinates to the reference system chosen		
				x [mm]	y[mm]	z [mm]
σ_x	Min.	3,2463 ·10^7	5619	324,63	-52,706	-21,287
	Max.	3,7036·10^7	11169	374,75	-12,787	-27,840
σ_y	Min.	3,2904·10^7	5619	324,63	-52,706	-21,287
	Max.	3,427·10^7	5671	298,76	-27,575	-9,4410
σ_z	Min.	4,1075 ·10^7	5618	325,98	-45,991	-21,329
	Max.	4,3755 ·10^7	959	281,07	20,228	-21,783

Table 2 shows that the same minimum value is in node 5619 and the node 5618 has a minimum value, located next to the previous node. With regard to the maximum, they are obtained in the corresponding area of the bridge nodes associated valves. All of the three components has the highest value. The analysis of stresses is more effective if we use the theory of the specific form modifying energy (stated by von Mises) as a determining factor for reaching the limit stages, [2].

By analysing the stresses fields, we are able to see the main critical area of the cylinder head. Von Misses stress state represents the values of a scalar field energy density obtained from the volume used to measure strain and stresses created in the model. In fig.6 presents the results embodied in the von Mises stress as a consequence of thermal and mechanical loads on the top of the cylinder head examined.

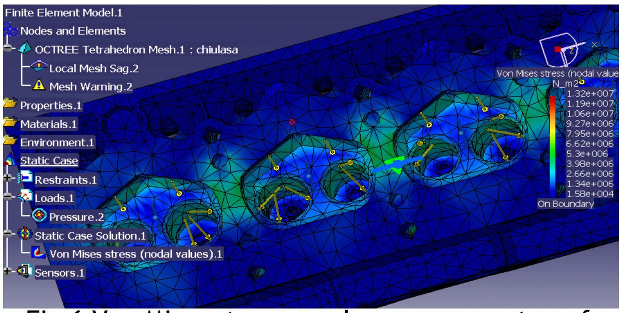


Fig.6 Von Mises stresses values across on top of the cylinder heads

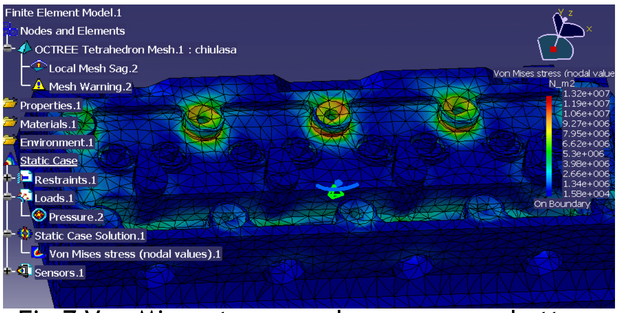


Fig.7 Von Mises stresses values across on bottom of the cylinder heads

From fig.6 we see that the most requested areas are appropriate areas bridge between the intake valve and exhaust valve, with a maximum value of $7{,}95 \cdot 10^7$ N/mm^2. The existence of maximum in this area confirms the theoretical literature and therefore the calculation model.

The map of values, shown in fig.7 for von Mises stresses is observed to obtain a maximum is reached at the cylinder head mounting bolts, with values of $1{,}32\ 10^7$ N/mm^2, and lower values its close around their area adiacente. It is envisaged that the thermal loads were not considered in modeling the maximum, but this does not change the existing critical areas in the cylinder head in terms of thermal and mechanical stresses

❖ CONCLUSIONS

Model calculation from the cylinder head design solution followed, this situation of exploitation to satisfy compatibility and equilibrium conditions inside the finite elements and the entire cylinder head forming, so simplification accepted modeling analysis does not change the results. In developing the model calculation to take into account the condition of symmetry of the cylinder head

This phenomenon expresses faithfully studied the cylinder head of their work cycle engine, because the conditions imposed meet the actual conditions of the outline application and also take account of finite element properties. is noted that the maximum application of stress concentrators arise in areas of stresses in the deck of the intake valve and exhaust valve which confirms the validity of the modeling because they are areas defined by the theoretical literature. This areas is the most requested where variable temperature fields creates excitement that overlaid the mechanical generate significant demand. Modeling carried out allows comparison of tensions with the experimentally determined and analytically.

❖ REFERENCES

[1.] Alaci S., Ciornei F.,L., Numerical versus analytical method in finding stress state in mechanisms elements, Fiability & Durability No.2 (6), 2010

[2.] Botean A.I., Studiul solicitărilor termomecanice în motoarele cu ardere internă utilizând metode moderne de cercetare , Raport final grant de cercetare, 2006

[3.] Gardyński L., Weroński A., Examination of diesel engine pistons' cracking reasons, Proc. 10thInternational Scientific Conference "Communications on the edge of the milleniums". śilina,SK, pp. 69-72, 1998.

[4.] Ghionea I., Consideration about the methodoly and results for the finite element analysis of a mechanical assembly, Proccedings of the 16th International Conference on Manufacturing System ICMaS, ISSN 1842-3183, 2006

[5.] Ghionea I., Proiectarea asistată de calculator în CATIA V5. Elemente teoretice şi aplicații, Ed. BREN, ISBN 978-973-648-654-8, Bucureşti, 2007

ANNALS OF FACULTY ENGINEERING HUNEDOARA
– INTERNATIONAL JOURNAL OF ENGINEERING

Faculty of Engineering Hunedoara,
5, Revolutiei, 331128, Hunedoara,
ROMANIA
http://annals.fih.upt.ro