

## The Finite Element Analysis & Improvement on a Single Cylinder Head of Spark Ignition Engine

<sup>1</sup>Pradeep Mani Tripathi, <sup>2</sup>Satya Prakash, <sup>3</sup>Rahul Singh, <sup>4</sup>Satish Kumar Dwivedi  
(<sup>1,2</sup>Student, <sup>3</sup>Lecturer & Guide, <sup>4</sup>HOD of Mechanical Department)  
(<sup>1,2,3,4</sup>Department of Mechanical Engineering-B.I.T GIDA, GORAKHPUR, UTTAR PRADESH TECHNICAL UNIVERSITY, India)

---

**ABSTRACT** -The finite element method is applied to find the stresses field from the parts of a single 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, which is known constructive characteristics and the thermal parameters. The paper deals with thermal & structural stress analysis and improvement of the cylinder head assembly of SI (Spark Ignition) engine. A detailed FE model was created for this purpose. The FE model consists of main parts of the cylinder head assembly and it includes a description of thermal and mechanical loads and contact interaction between its parts. The model considers a temperature dependency of a thermal conductivity and heat transfer coefficient on wall temperature. The paper presents a comparison of computed and measured temperature. The design was carried out using the SolidWorks v12 and analysis was carried out using the FE program ANSYS v14.

**Keywords** -Actual process, Cylinder Head, Deformation Spark Ignition Engine, Temperature, Finite element, Stresses, mesh.

---

### I. 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. Such as determining the stresses and strains in an internal combustion engine cylinder head.

This paper aims to study the stresses & temperature distribution in single cylinder head spark ignition engine. Cylinder head temperature has considerable influence on efficiency, emission, and performance of the SI (Spark Ignition) engine. Purpose of this investigation is measurement of cylinder head transient temperature at several points. On the cylinder head temperature changes with time and engine speed, from cold start to steady condition and comparison with results of finite element analysis. From analytical analysis of cylinder head, it is proposed to conduct stress analysis of a thick walled cylinder head near the radial hole on the surface.

The literature indicated that there will be a ductile fracture occurring in such cases. The radial holes cannot be avoided due to various attachments. Hence the stress analysis of cylinder head and its ultimate failure under internal pressure beyond elastic limit is an appropriate scenario. It is observed that there are several factors which influence stress intensity factors. The Finite element analysis is conducted using commercial solvers ANSYS & SolidWorks. Theoretical formulae based results are obtained from ANSYS programs. The single-cylinder petrol engine has the properties of small volume, high power, high torque at low speed, good start-up and acceleration performance, low oil wear and cheap, and it has been widely used. Adopting the aluminum alloy engine cylinder head can effectively reduce the total weight of the engine. But the elastic modulus of aluminum alloy is far less than that of cast iron, the deformation amount of the engine cylinder head and related parts sharply increase with the fuel gas maximum combustion pressure. Deformation of the cylinder head will cause gas leakage, oily smoke, power down, fuel consumption increase, increase the possibility of piston scraping; Deformation of cylinder top surface will increase the possibility of the cylinder gasket and decrease mechanical efficiency, moving parts can't work normally. Otherwise, because of the high thermal conductivity of aluminum alloy, the heat of combustion chamber can transfer to crankcase easily and make the oil temperature rise so that lubrication performance of engine get worse, engine power decrease and can't work normally in serious case. The metals are typically alloys of aluminum, magnesium, or titanium. The special benefits exhibited by metal matrix composites, such as lower density, higher specific strength and stiffness, high temperature performance limits, and improved wear-abrasion resistance on the properties of the matrix alloy dependent on the properties of the matrix alloy. In the combustion chamber, there are high peaks of combustion pressure and temperature in the order of 15 MPa and 2500K. The maximum temperature of the head material is much lower and the regions around the combustion chamber need to be safely cooled to prevent overheating

these facts are key factors to many compromises in design, which can be sources of failures in operation. To nullify the risk of operational failure is one of the targets of engine designers. The design of the engine head must be tested under these operational conditions. This procedure is necessary but expensive. FE modeling of the cylinder head assembly operation conditions is an appropriate complement to the operational testing. A detailed FE strength analysis provides valuable information about temperature distribution and mechanical stresses in the overall assembly of the cylinder head. This information is especially useful in regions where the experimental data is scarcely available to gather. Temperature and mechanical stresses is analyzed using temperature field, combustion pressure in the combustion chamber and mechanical loads, such as bolt pre-stress, molded seats and valve guides, etc. The resulting displacement/stress fields may be utilized for the evaluation of operational conditions, i.e. contact pressure between valves and valve parts uniformity as well as strength and failure resistance of the assembly. Such information contributes to a detailed evaluation of the thermal and mechanical processes in cylinder-head assembly under engine operation, which is a prerequisite for further optimization of engine design. Cylinder head Distortion means deviation from ideal circular shape (roundness), or cylinder head out of roundness. Initially it is understood that manufacturing tolerances can cause the cylinder head to deviate from ideal condition.

## **II. SOME CONCEPT ABOUT FINITE ELEMENT METHOD**

The finite element method is one of the most used methods that are available in our day's 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. 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. 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 model should efficiently synthesize all the available information about the analyzed piece, 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. 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 thermic 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. 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. 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. The mesh of an 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.

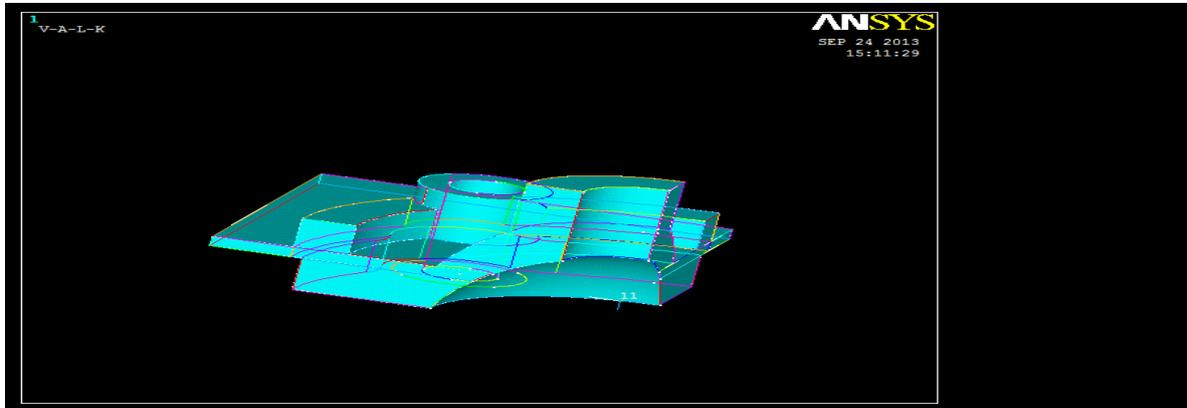
## **III. FEM APPROACH**

It is not always possible to obtain the exact analytical solution at any location in the body, especially for those elements having complex shapes or geometries. Always matters are the boundary conditions and material properties. In such cases, the analytical solution that satisfies the governing equation or gives extreme values for the governing functional is difficult to obtain. Hence for most of the practical problems, the engineers resort to numerical methods like the finite element method to obtain approximate but most probable solutions.

Finite element procedures are at present very widely used in engineering analysis. The procedures are employed extensively in the analysis of solids and structures and of heat transfer and fluids, and indeed, finite element methods are useful in virtually every field of engineering analysis.

### **III.I. Finite Element Model**

The petrol engine cylinder head assembly parts are shown as Fig.1.



### **III.II. Boundary Conditions**

Displacement constraints of the X, Y, Z three directions are imposed on the location of the four bolt holes of the lower body. In order to investigate the effect of side pressure of cylinder head, forced with the maximum side pressure of cylinder head.

### **III.III. Calculation Schemes**

**III.III.I. Schemes1:** The indexes of the stress, the flatness of the cylinder head end surface and the rate of light leakage of each cross section of the petrol engine cylinder head combined parts can be got by analyzing the cast-iron engine cylinder head according to finite element method. Those indexes are as evaluating indicator of the improved schemes.

**III.III.II. Scheme2:** Changing the cast-iron engine body into aluminum alloys cylinder head, and then analyzing the evaluations indexes of scheme1 and scheme2 by finite element method contrastively.

**III.III.III. Scheme3:** Increasing the depth of the counter bore of cylinder head bolt from 5mm to 12mm and thickness of the four stiffeners behind the cylinder head bolt from 6mm to 10mm, dredging the Partial metal near the two bolts on the upper body, increasing the area of the stiffener over the Body's head. Then analyze the evaluating indicator of scheme3 and scheme1 by finite element method contrastive.

**III.III.IV. Scheme4:** Increasing the depth of the counter bore of cylinder head bolt from 5mm to 15.2mm; other improvement measures are same as scheme3. Then analyze the evaluating indicator of scheme3 and scheme1 by finite element method contrastively.

## **IV. ANALYSIS**

### **IV.I. Thermal Analysis**

A thermal analysis calculates the temperature distribution and related thermal quantities in a system or component. Typical thermal quantities of interest are:

- The temperature distributions.
- The amount of heat lost or gained.
- Thermal gradients.
- Thermal fluxes.

Only the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Professional, and ANSYS FLOTRAN programs support thermal analyses. The basis for thermal analysis in ANSYS is a heat balance equation obtained from the principle of conservation of energy. The finite element solution perform via ANSYS calculates nodal temperatures, and then uses the nodal temperatures to obtain other thermal quantities. The ANSYS program handles all three primary modes of heat transfer: conduction, convection, and radiation.

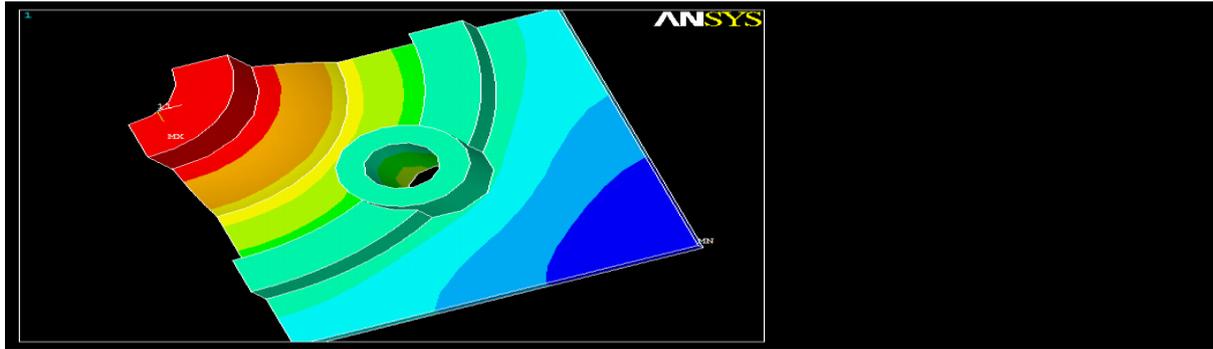


Fig-Temperature Distribution

#### **IV.I.I. Types of Thermal Analysis**

ANSYS supports two types of thermal analysis

-A **steady-state thermal analysis** determines the temperature distribution and other thermal quantities under steady-state loading conditions. A steady-state loading condition is a situation where heat storage effects varying over a period of time can be ignored.

-A **transient thermal analysis** determines the temperature distribution and other thermal quantities under conditions that vary over a period of time.

### **V. TRANSIENT THERMAL ANALYSIS**

Transient thermal analysis is the thermal analysis where in boundary conditions and properties Change with time. This is to say that the constraints such as ambient temperature, thermal coefficient and material properties etc. are time dependent. Transient thermal analysis is important in analyzing models that are subjected to boundary conditions and material properties that with time and temperature. Cylinder head used in SI engines are subjected to high temperature rise. Since the cylinder head is subjected to large temperature variation, the material properties such as specific heat, enthalpy and young's modulus undergo variations with time. In such conditions there is the probability of failure of the cylinder head if the cylinder head is not designed taking into consideration the transient effects. At the beginning of the cold start, the component is at uniform ambient temperature. When a cold metal component is exposed to a hot fluid medium, thermal gradients set in across the thickness and along the length. During the thermal transient, this temperature gradient changes with time till the metal reaches its steady state temperature distribution. Due to thermal energy of the metal, it takes more time to reach its steady state value by conduction than the actual duration of fluid temperature change during the transient.

### **VI. STRUCTURAL ANALYSIS**

Structural analysis is probably the most common application of the finite element method. The term structural implies not only civil engineering structures such as bridges and buildings, but also automobile, aeronautical and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical components such as pistons, machine parts, and tools.

#### **VI.I. Types of Structural Analysis**

The seven types of structural analyses available in the ANSYS family of products are explained below. The primary unknowns (nodal degrees of freedom) calculated in a structural analysis are displacements. Other quantities, such as strains, stresses, and reaction forces, are then derived from the nodal displacements. Structural analyses are available in the ANSYS Multiphysics, ANSYS Mechanical, ANSYS Structural, and ANSYS Professional programs only. You can perform the following types of structural analyses. Each of these analysis types are discussed in detail in this manual.

**VI.I.I. Static Analysis** is used to determine displacements, stresses, etc. under static loading conditions. Both linear and nonlinear static analysis. Nonlinearities can include plasticity, stress stiffening, and large deflection, large strain, hyper elasticity, contact surfaces, and creep.

**VI.I.II. Modal Analysis** is used to calculate the natural frequencies and mode shapes of a structure. Different mode extraction methods are available.

**VI.I.III. Harmonic Analysis** is used to determine the response of a structure to harmonically time varying loads.

**VII.IV. Transient Dynamic Analysis** is used to determine the response of a structure to arbitrarily time varying loads all nonlinearities mentioned under Static Analysis above are allowed.

In addition to the above analysis types, several special-purpose features are available for solving structural problems in the ANSYS family of products: the h- method and the p-method. The h-method can be used for any type of analysis, but the p-method can be used only for linear structural static analyses. Depending on the problem to be solved, the h-method usually requires a finer mesh than the p-method. The p-method provides an excellent way to solve a problem to a desired level of accuracy while using a coarse mesh. In general, the discussions in this manual focus on the procedures required for the h method of solution discusses procedures specific to the p-method.

## **VII. STRUCTURAL STATIC ANALYSIS**

A static analysis calculates the effects of steady loading conditions on a structure, while ignoring inertia and damping effects, such as those caused by time-varying loads. A static analysis can, however, include steady inertia loads (such as gravity and rotational velocity), and time-varying loads that can be approximated as static equivalent loads (such as the static equivalent wind and seismic loads commonly defined in many building codes). Static analysis determines the displacements, stresses, strains, and forces in structures or components caused by loads that do not induce significant inertia and damping effects. Steady loading and response conditions are assumed; that is, the loads and the structure's response are assumed to vary slowly with respect to time.

A static analysis can be either linear or nonlinear. All types of nonlinearities are allowed – large deformations, plasticity, creep, stress stiffening, contact (gap) elements, hyper elastic elements, and so on. This chapter focuses on linear static analyses, with brief references to nonlinearities. The thermal gradients produce thermal stresses, in transient and steady state conditions, in the component. Stress analysis is carried out on Finite element model of the component at the critical time of transient, when thermal gradient is high. Stresses due to fluid pressure are added to the thermal stresses. These total stresses vary as functions of space and time.

## **VIII. COUPLED THERMAL STRUCTURAL ANALYSIS**

A sequentially coupled physics analysis is the combination of analyses from different engineering disciplines which interact to solve a global engineering problem. When the input of one physics analysis depends on the results from another analysis, the analyses are coupled. Thus, each different physics environment must be constructed separately so they can be used to determine the coupled physics solution. However, it is important to note that a single set of nodes will exist for the entire model. By creating the geometry in the first physical environment, and using it with any following coupled environments, the geometry is kept constant. For our case, we will create the geometry in the Thermal Environment, where the thermal effects will be applied. Although the geometry must remain constant, the element types can change. For instance, thermal elements are required for a thermal analysis while structural elements are required to determine the stress in the rotor and casing. It is important to note, however that only certain combinations of elements can be used for a coupled physics analysis.

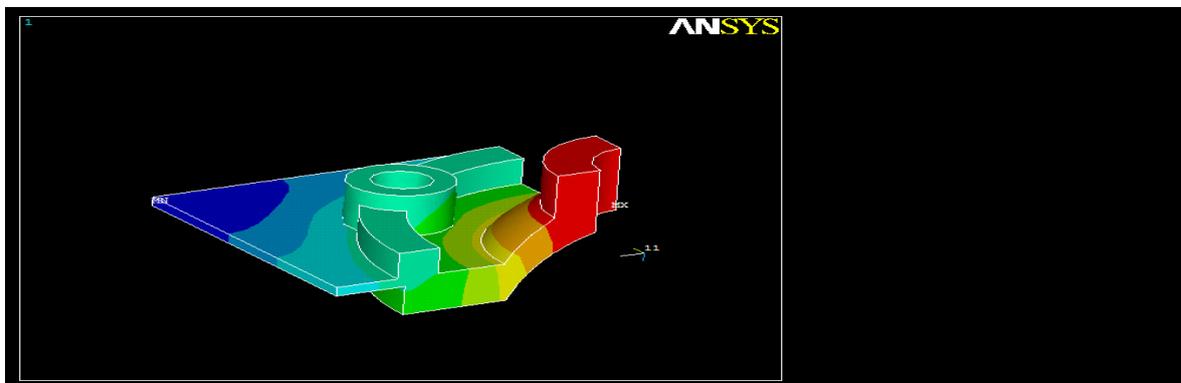


Fig-stress distribution

## **IX. RESULTS AND DISCUSSIONS**

### **IX.I. Transient Temperature Distribution and Thermal stress**

At the beginning of the cold start, the component is at uniform ambient temperature. When a cold metal component is exposed to a hot fluid medium, thermal gradients set in across the thickness and along the length.

During the thermal transient, this temperature gradient changes with time till the metal reaches its steady state temperature distribution. Due to thermal inertia of the metal, it takes more time to reach its steady state value by conduction than the actual duration of fluid temperature change during the transient.

**IX.II. Thermal Stress Analysis**

The thermal gradients produce thermal stresses, in transient and steady state conditions, in the component. Stress analysis is carried out on Finite element model of the component at the critical time of transient, when thermal gradient is high.

**IX.III. Static Analysis**

The fluid pressure produces static stresses, in steady state conditions, in the component. Stress analysis is carried out on Finite element model of the component at different pressure loads.

**IX.IV. Static Transient Thermal Analysis**

Stresses due to fluid pressure are added to the thermal stresses. These total stresses vary as functions of space and time. Pressure loads and Thermal gradients are applied at a time to find out the stresses.

Modeling was performed for the gas pressure  $P_g = 3 \text{ N/mm}^2$  and temperature  $T_g = 1900 \text{ K}$  were determined by diagrams of the engine indicated in the nominal regime. The material properties are: Young modulus ( $7 \cdot 10^{10} \text{ N/m}^2$ ), Poisson ratio (0.346), density ( $2710 \text{ kg/m}^3$ ), thermal expansion ( $2.36105 \text{ K}$ ) and the yield strength ( $9.5 \cdot 10^7 \text{ N/m}^2$ ). The solid model of the cylinder head is presented in fig. 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.

Mechanical components		Values	Reactions[N]
Load [N]	Fx	- 4150	4150
	Fy	968.91	- 968.91
	Fz	2662.1	- 2662.1
Moments [N-mm]	Mx	5.349	- 5.3496
	My	- 853.12	853.12
	Mz	310.51	- 310.51

Table 1. The value of forces and moment used in modeling 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  $p_g = 7.8 \text{ N/mm}^2$  and temperature is  $T_g = 2200 \text{ K}$  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. is presented cylinder head on the top, examined from the finite element mesh 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. 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.

Principal stresses		Values [N/mm <sup>2</sup> ]	Nodes	Nodes coordinates to the reference system chosen		
				X[mm]	Y[mm]	z[mm]
$\sigma_x$	Min	$3.24638 \times 10^7$	5619	324.63	-52.706	-21.287
	Max	$3.7036 \times 10^7$	11169	374.75	-12.787	-27.840
$\sigma_y$	Min	$3.2904 \times 10^7$	5619	324.63	-52.706	-21.287
	Max	$3.427 \times 10^7$	5671	298.76	-27.575	-9.4410
$\sigma_z$	Min	$4.1075 \times 10^7$	5618	325.98	-45.991	-21.329
	Max	$4.3755 \times 10^7$	959	281.07	-20.228	-21.783

Table 2. The maximum and minimum principal stresses

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 have 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. By analyzing the stresses fields, we are able to see the main critical area of the cylinder head. Von Mises 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. 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.

## X. CONCLUSION

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. It 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 area is the most requested where variable temperature fields create excitement that overlaid the mechanical generate significant demand. Modeling carried out allows comparison of tensions with the experimentally determined and analytically.

Conclusions are drawn from this analysis are

1. Weight of the Cylinder Head can be minimized by making Cylinder head with metal matrix composites instead of Cast iron.
2. Stress analysis is carried out through FE model.
3. Agreement between 2D and 3D is arrived.
4. Computational time is reduced.
5. This Analysis is handy in automotive applications.

## REFERENCES

- [1] Saniel, M. - Macek, J. - Diviš, M. - Tichánek, R. Steady State Heat Analysis of Engine Head, Abstracts of 20th Danubia- Adria Symposium on Experimental Methods in SolidMechanics. Győr: Széchenyi István University of AppliedSciences, 2003, s. 74-75. ISBN 963-9058-20-3.
- [2] Macek, J. – Vitek, O. – Vávra, J.: Kogenerační jednotka splynovým motorem o výkonu vitším než 3 MW –II. ÈVUTv Praze, Fakulta strojní, 2000. (in Czech)
- [3] Horák, F. – Macek, J.: Use of Predicted Fields in Main Parts of Supercharged Diesel Engine. *Proceedings ofXIX. Conference of International Centre of Mass and Heat Transfer*. Pergamon Press, New York, 1987.
- [4] Kreith, F. – Black, W.: *Basic Heat Transfer*. Harper and Row, New York, 1980.
- [5] Baehr, H.D. – Stephan, K.: *Heat and Mass Transfer*. Springer-Verlag, Berlin, 1998.