Journal of Mechanical Engineering Research and Developments
ISSN: 1024-1752
CODEN: JERDFO
Vol. 43, No. 3, pp. 321-331
Published Year 2020

Modelling And Finite Element Analysis For The Engine Cylinder Head Under Nonlinear Dynamic Thermal Mechanical Loading

Tawfeeq. N. Hussein*, Minah. M. Fareed, Hussein Al-Gburi

Al-Mussaib Technical Institute, Al-Furat Al-Awsat Technical University, 51009 Babylon, Iraq.

*Email: com.h.tawfeeq@atu.edu.iq,com.fm.Mena@atu.edu.iq, com.t.husain@atu.edu.iq

ABSTRACT: Internal combustion engine is considered as a highly stressed structure due to high entire temperature and presser. For the engine structures and their entire components like cylinder head; it's no arguing that this structure can consider a highly loaded, because it's subjected to high fluctuations in temperature, pressure and clamping loads. The main objectives for this research work are to evaluate the materials response distortion, stress distribution and their effects on the engine performance. For these considerations, finite elements analysis methods by using (ABAQUS / CAE) software have been adopted for modelling and analysis purpose. The model with two layers (elastoplastic –viscoelastic) is considered as a best model which suitable to modelling the materials response of aluminum cylinder head with plasticity as well as considerable time-dependent at room and elevated temperatures. The results shown that the distortion in the cylinder head, valve ports and nearby regions is caused by the thermal stresses concentration and Von Mises stress distribution. The normal assessing of product performance is depending on fatigue and failure prediction in the overall structure. Also the direct cyclic analysis procedure is normally depending on the design considerations and this normally used for calculations of fatigue life.

KEYWORDS: Cylinder Head, Stresses, Plastic Deformation, Simulation.

INTRODUCTION

One of the most complicated part in the internal combustion engine which directly subjected to the effects of burning gasses and peak temperatures is cylinder head, and to prevent overheating inside the cylinder engines, cooling must be supplied. there are also several loads affecting on this parts like force vibration, inertia force, thermal loads and improper stress distribution which leads to high fluctuating and variation in speed, and gases explosions [1]. The important solutions for reducing the effects of high temperature is to applying the insulation coating on heat pipe cooling can used to reduce stresses and thermal deformations. In working condition, the piston will be heavy loaded with high rotational speed, therefore cylinder head will subject to fluctuation in temperature and pressure with high frequency and this may results damage of cylinder bore [2].

Thermomechanical fatigue analysis is consider the best way for estimation the reliable lifetime of cylinder head. By this analysis, it's possible to specify the reasons of failure, and identify the critical locations for any existing components to find out some design changes for better lifetime [3]. However, increasing the performance and power of engine will leads to generate more thermal stresses with high thermal loads which acting on the engine piston head. Finite element analysis is the powerful tools which used to find out the weak points to avoid the stress concentration problems and then maintain a suitable functioning and good condition of piston head [4].

It's recommended to modify, geometry, thickness, and material type of some crucial items in cylinder heads to avoid the problems of fatigue and cracking. Also it's advised that thinner parts and regions should not overload to prevent fatigue faller and increase life of cylinder heads [5]. The working temperature of any specific device is considering the crucial factor in evaluation the potential life of this device. Investigate the engine temperature analytically is so helpful in design and improvement the entire components of the engine [6]. Cylinder head is the most severing parts to different loading conditions like pressure and high temperature, and the thermal load is one of the considered factors that affected on cylinder head performance, and should be consider in design [7,8].

Piston head design and material specifications should have the capability to withstand the cyclic stresses caused by high pressure and high amount of heat with temperature fluctuations. Consequently, Von-Mises stress generation which normally leads to high fatigue rate should be avoid to increases the cylinder head life time. In heat distribution and stress analysis; computer simulation by applying FEA will leads to obtaining the optimum thermal distribution and minimize the production costs. Moreover; analysis and simulation of this thermo-mechanical system will emphasis the understanding of predicting the main reasons which cause to engine faller [9].

Finite element simulation is widely used to improve the design of components and forecasting the problems before it's happened. This type of analysis is widely used for correlation between theoretical calculations and empirical findings. ABAQUS software normally used as F.E.M tools to estimate the interaction between the working components of internal combustion engine and to locate the weakness and strengths of the structure [10,11].

The advantages of F.E.A are beyond than creating the virtual parts and then modelling it, but it's also offering many options for designer and manufacturer and enable them to avoids the high expense associated with actual fabrication. These analysis tools can eliminate many production problems and determine the sensitivity parameters associated with product quality. In order to control the deformations and the effects of thermal stresses on the piston head within a suitable levels; the temperature distribution should be determine. Aluminium alloys are widely used as the basic material for engine pistons rather than cast iron due to excellent thermal expansion coefficient comparing with iron, so the behaviour analysis of piston thermal is very crucial factor in designing good performance engines [12,13]. Finite element analysis is normally used to simulate the thermal stresses of cylinder head. These stresses can calculate and simulate under similar working conditions. The characteristics of temperature distribution at actual conditions have a little difference from that found by analysis due to some differences between the simulation boundary conditions and real working conditions [14,15].

MAIN OBJECTIVES

The main objectives for this research work are

1- To evaluate the materials response distortion, stress distribution and their effects on the engine performance.

2- To investigate the stress and temperature distribution in engine cylinder heads by using a viscoelasticity model by F.E approach.

METHODOLOGY

Design and Modelling Approach

Cylinder head is consider as a main component in combustion engine. Also the engine is consist of two valve guides which are tightly fit into valve ports. These three valve seats have press-fit tolerance with the port of cylinder head valve. Figure 1 shows these components in three dimensions.

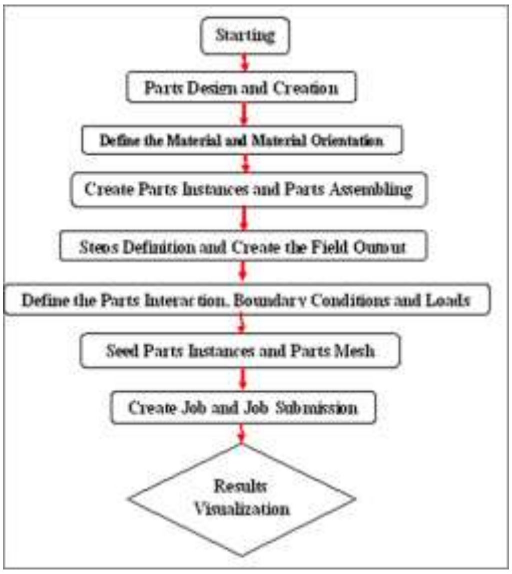

Figure 1. Cylinder head Assembly.

Cylinder head body is made of aluminum alloys with the following mechanical properties illustrated in Table 1 below. Also there are three valve seats and two valve guides. The material of these components is steel with poison's ratio of (0.35) and 200 GPa Young's modulus and considers to behave elastically.

Table 1. Mechanical properties of aluminum used in cylinder head.

No.	Property	Values
1	Young's Modulus (Gpa)	70
2	Yield Stress (Mpa)	62
3	Poisson's Ratio	0.33
4	Coefficient of Thermal Expansion	22.6 × 10–6 per °C at room temperature

Numerical Analysis and Simulation Process

All the components of this structure (cylinder head, valve seats and the valve guides) are modelled as three-dimensional continuum elements, and the procedure of static analysis is used for this purpose. This model is consisting from an elastic-viscous network parallel with an elastic-plastic network. Strain hardening with power-law creep model is normally used in elastic-viscous network, while kinematic hardening with Mises metal plasticity model is used with elastic-plastic network. However, aluminum elastic-viscoelastic response is greatly varies with temperatures, so the properties of (TEMPERATURE-DEPENDENT MATERIAL) should be specified.

Preceding the analysis procedure are requiring different boundary conditions like define the solution steps, applying loads, and find out the interaction behavior between the contact surfaces and bodies. Figure 2 is a flow chart which illustrates this procedure. Specify and define the boundary conditions, as well as the interactions type between the contact surfaces will help for locating the parts position according to the reference body or surfaces and eliminate the movement of constrain surfaces. One of the main boundary condition is to constrain and prevent cylinder head from any movement in all direction, as shown in Figure 3.

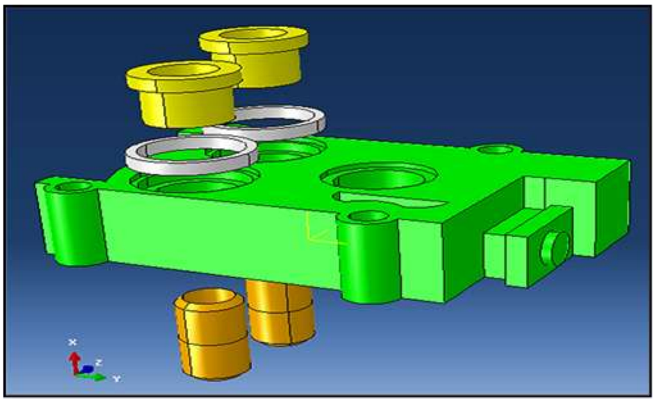

Figure 2. Analysis procedure flow chart.

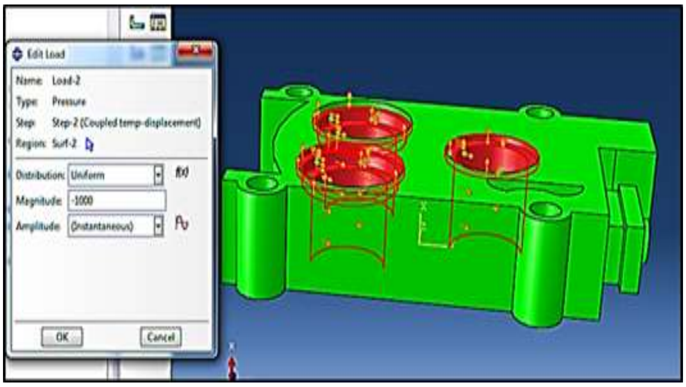

Figure 3. Constrain the cylinder head.

During operation, the system will subject to different severe loading like pressure load and thermal loading. Figure 4 illustrate the loaded cylinder head.

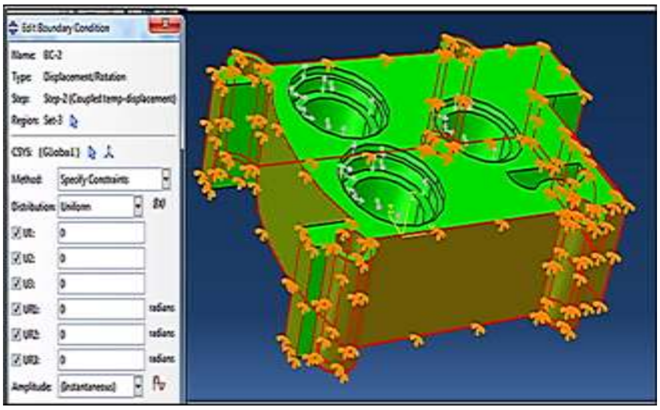

Figure 4. Applying different loads on the cylinder head.

Seeding parts according to global seed and control the approximate sizing by using curvature control with maximum deviation to estimate the average number of elements per cycle are used to facilitate the meshing process. Matched meshes are used to model interface between each of the two components to share the nodes along the interface. Hex- dominated element shape with default algorithm are using in mesh control. Element type of standard (3D) -Stress and quadratic geometry factor are used in mesh these parts. Figure 5 illustrate all parts after meshing process.

This will carried out by specify the equations of the radial constraint between the nodes on the valve port surface and the nodes on the valve seat surface, if we know that the radial displacement on the valve seat is the radial displacement on the valve port with same reference node. In the first analysis steps a specific displacement will apply to the reference node and this will results in developing normal pressures between the components.

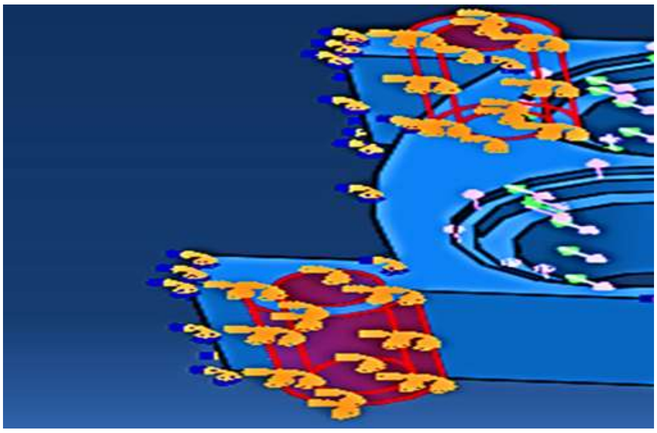

Figure 5. Meshing all parts.

This model is consist of (1550) first order elements and (18500) brick elements with about (73,000) totally degrees of freedom. In such complex geometry, the use of (C3D6) elements is considering the suitable choice in such case of analysis.

To enhance the common plot option; model labels is the prosperity which can used to count, determine, and locate all body and surface element and nodes according to their position. By using this statistical option, it's possible to estimate the values of loads that acting at any nodes or element. Figure 6 show the elements labels of cylinder head.

This model is consist of (1550) first order elements and (18500) brick elements with about (73,000) totally degrees of freedom. In such complex geometry, the use of (C3D6) elements is considering the suitable choice in such case of analysis.

To enhance the common plot option; model labels is the prosperity which can used to count, determine, and locate all body and surface element and nodes according to their position. By using this statistical option, it's possible to estimate the values of loads that acting at any nodes or element. Figure 6 show the elements labels of cylinder head.

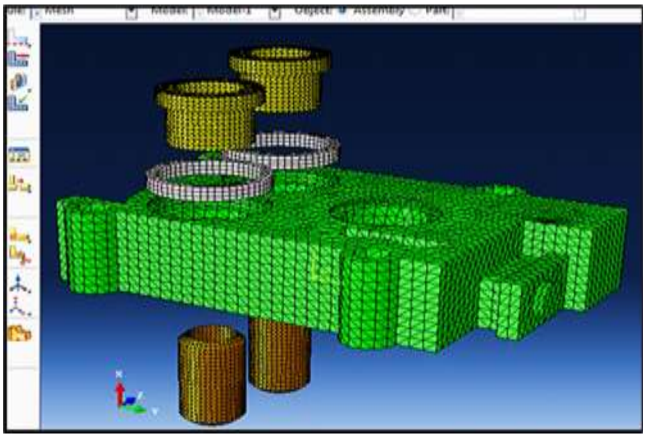

Figure 6. Elements labels of cylinder head.

Two analysis steps are used in applying load to the assembly. By using the prescribed displacement loadings and linear multi-point equation constraints will use in the first step to press-fit the seat of three valves into the corresponding cylinder head valve port. In second analysis step, cyclic thermal loads will apply. In this step, it is assumed that the four bolt holes are securely fixed the cylinder head to the engine block. That's means all the nodes along the four bolt holes base will be also secured in all directions as shown in Figure 7.

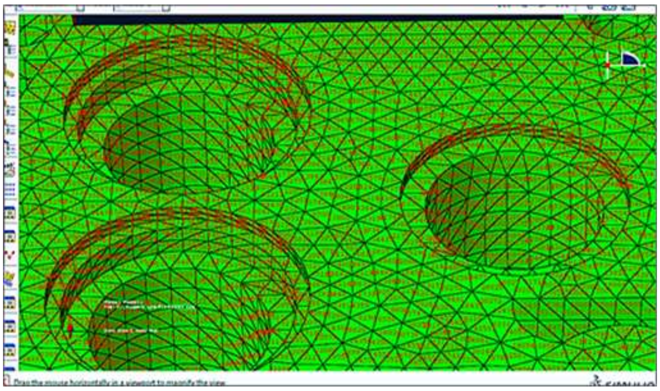

Figure 7. First analysis step.

To obtain a steady-state thermal cycle; a three thermal cycles are carried out in this analysis, and through performing the analysis of independent thermal, the cyclic thermal loads can be obtained. Each of this thermal cycle is consisting of two steps: the first one is by using the option (FILM and CFLUX), to heat the cylinder head until reaching the maximum operating temperature and then cooling the cylinder head to the minimum operating temperature. In thermal-mechanical analysis of the second step, the nodal temperatures will assume to be steady-state. This step is a coupled temperature displacement under the option of (Surface Heat Flux) as shown in Figure 8 which illustrate this steady-state cycle as a function of time at the node (49550).

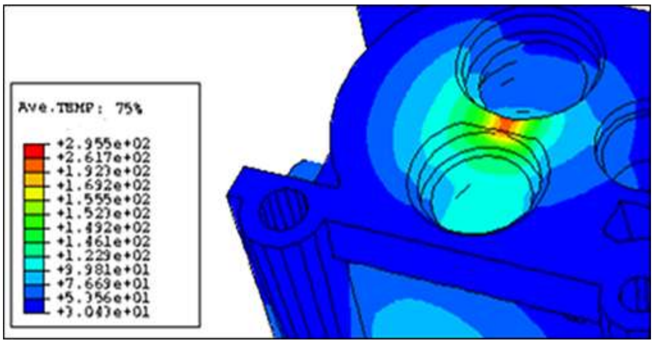

Figure 8. Second analysis step.

In the second step of mechanical analysis and from the prior heat transfer analysis, the option of generated nodal temperatures will apply. In this second step, the (35) load cycle period with (0.30) fixed time incrimination is specified. The maximum iterations number is (50) and (90) respectively, and one iteration is resulting of (140) total number increments. Same model is also analysed for comparison purposes by using the classical transient analysis. Many repetitive steps are requires for this this comparative analysis till this solution is settled. In each step, a (35) load cycle period and cyclic temperature loading with (0.3) time incrimination are applied.

RESULTS AND DISCUSSION

To obtain the response for such structure; the normal approach is that; periodic loading will apply repetitively to the structure parts until obtained a stabilized state or occurring the plastic ratcheting. To obtain the steady response for this approach, many cyclic loading was applied, and for this transient analysis, it's important to avoid the numerical expense and directly calculate the cyclic response of the structure (DIRECT CYCLIC ANALYSIS) procedure will use for this purpose.

Distortion in the valve ports and nearby regions with stress distribution is one of the design considerations of a cylinder head. Figure 9 is a contour which shows the distribution of the Mises stress in the end of a loading cycle of cylinder head (iteration85, increment (130) in the cyclic analysis).

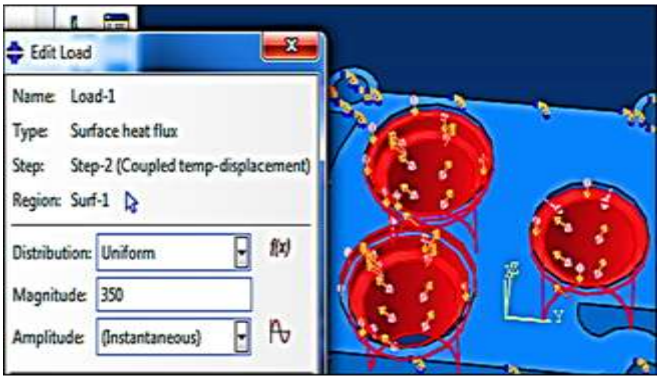

Figure 9. Contour of Mises stress distribution.

The critical regions in this design are those regions which they very close to the valve ports, because the stress and deformation are most severe. However, the average values of the viscous strain over a cycle, and the average values of the plastic strain during iteration is continue to increase from iteration to another, which indicate that the peak value of plastic will occur near by the valve ports. A comparison of plastic strain versus stress evolution in the direct analysis during the first round iteration with that evaluated at the end round of the transient cycle is shown in Figure 10.

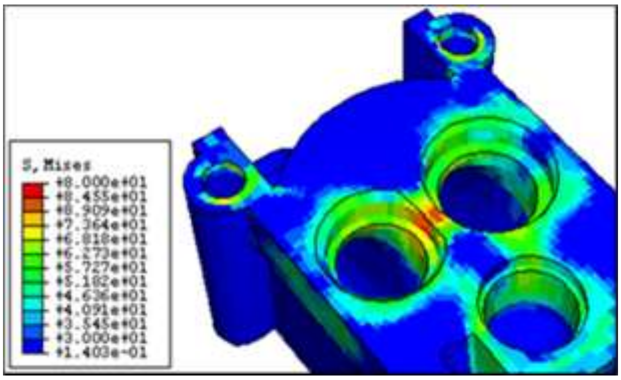

Figure 10. Contour of plastic strain.

The regions of the valve ports and nearby are subjected to the coverage of hot exhaust gases and fluctuations in cyclic temperature ranging from (35°C) as minimum value to (320°C) maximum. Creep deformation as well as plastic deformation is normally observed in such operating conditions. Figure 11 show the contour of average temperature distribution.

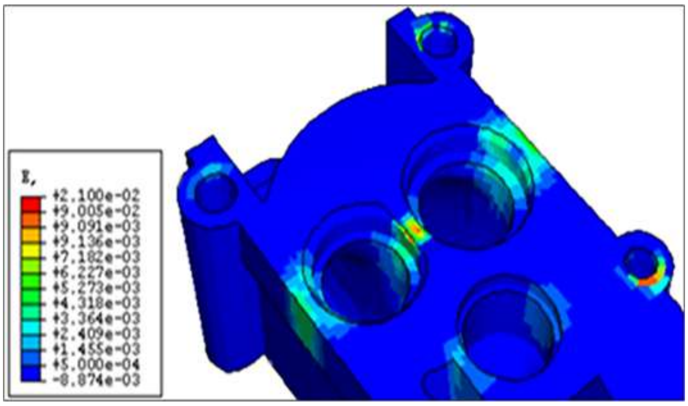

Figure 11. Contour of average temperature distribution.

In the valve ports and nearby this zone, the temperature will be in the maximum value due to the coverage of hot exhaust gases. Heat flux and temperature dissipation due to repetitive thermal cycles and over loading cycles is shown in Figure 12.

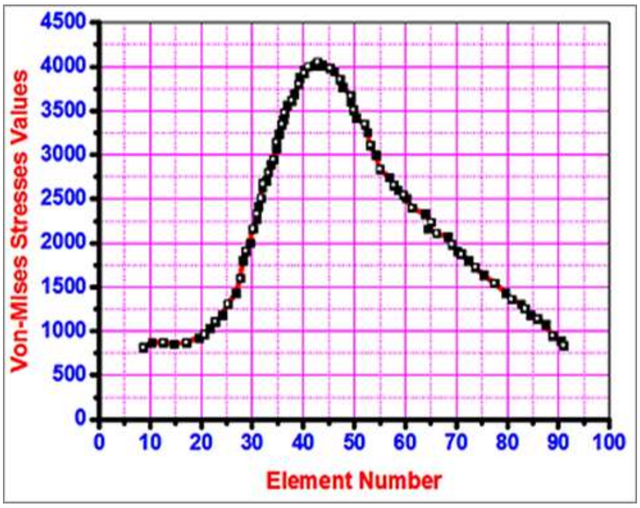

Figure 12. Heat flux and temperature dissipation.

The evolution of identical comparison between viscous strains versus stress is shown in Figure 13, and they have similar shapes of strain -stress curves.

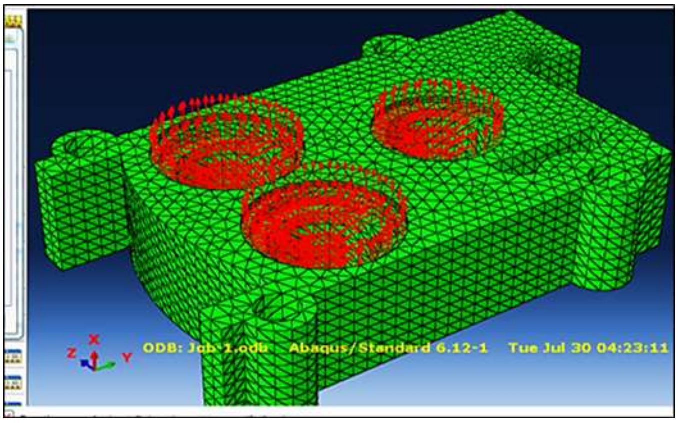

Figure 16. Whole system temperature versus total time.

During the peak loading cycle of cylinder head the Von-Mises stress values of any element in the structure will raise up or down according to their distance from the port. Figure 14 illustrate this relationship.

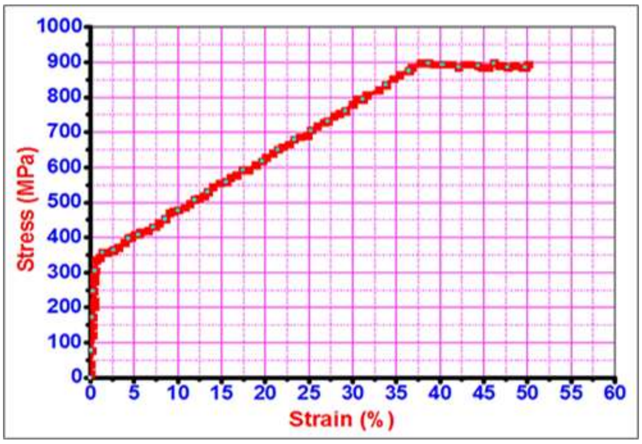

Figure 14. Von-Mises stress values with element number.

The computational time in the direct cyclic analysis of the first plastic occurrence is about (65%) of the total time spent in this analysis, and this consider as a significant saving as the problem data is very big. In this work, the time for the first plastic occurrence was approximately (60) % of computational time of the whole analysis time (80 iterations). Furthermore, the computational time of this analysis can reduced if the incrimination time in each step is specified to be (0.5) rather than (0.3) without any effects on results accuracy. During this time reduction; nodal temperature will increase rapidly until reach the steady state condition as shown in Figure 15.

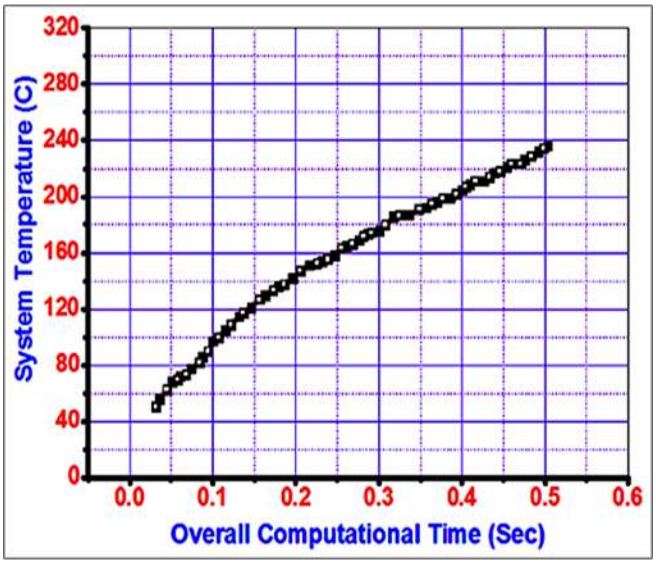

Figure 15. Relationship between nodal temperature and time.

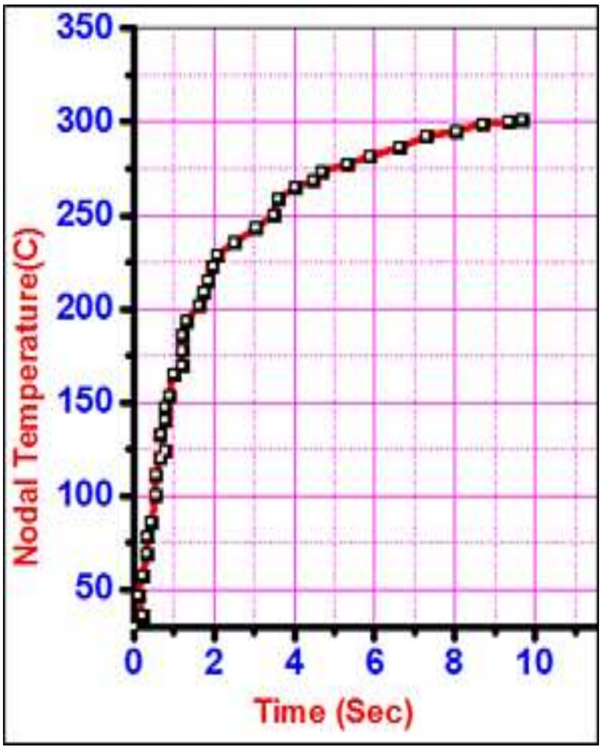

Figure 16. Whole system temperature versus total time.

Direct cyclic approach is normally used instead of classical approach for many advantages to predict automatically the plastic increase or to stabilize the cyclic response, and also for comparing the residual coefficients with displacement and others controlling variables.

CONCLUSIONS

Regarding to this analysis; thermal loading and others dynamic loading that effect on cylinder head has been carried out. As a conclusions, it has been proven that the thermomechanical fatigue analysis is a suitable method for estimation the reliable lifetime of cylinder head. Also it is possible to identify and estimate the critical locations which leads to main reasons for failures. It's concluded that, the critical regions in this design are those regions which they very close to the valve ports, because the stress and deformation are most severe. This analysis revealed that, the average values of the viscous strain over a cycle, and the average values of the plastic strain during iteration is continue to increase from iteration to another, which indicate that the peak value of plastic will occur near by the valve ports. The final examination step makes it so clear that residual coefficients and displacement value is not stabilize, and for this reason the plastic increasing will occur, and the last dangerous case is happen when plastic ratcheting, where in this case the design will fail. However, selecting the suitable material type for each part of cylinder head is one of the important choices to enhance the efficiency and performance of the engine. Finally, it's important to say that achieving an accurate results through using this type of analysis will enhance the lifetime of cylinder head.

REFERENCES

[1] S.K. Nagrik, M.S. Merchant, M.S. Rajpurohit, and S.S. Pachpore. "Design, analysis and comparing for a different cylinder head of C.I. Engine", *International Journal of Advance Research, Ideas and Innovations in Technology*, 3 (2), 2017. ISSN:2454-132X.

[2] M. Abid, M. Bannikov, and J.C.H. Ahmad. "3-D Finite Element Analysis of a Diesel Engine Piston". https:// www.research .net/publication/262005286, 2014.

[3] A. Leppa¨nen, A. Kumpula, J. Vaara, M. Cattarinussi, J. Ko¨nno¨ and T. Frondelius. ''Thermomechanical fatigue analysis of cylinder head''. *Journal of Structural Mechanics*, 50 (3), 2017, pp. 182 – 185, 2017. https://rakenteidenmekaniikka.journal.fi/index.

[4] G.V.N. Kaushik. (2019). ''Thermal and Static Structural Analysis on Piston''. *International Journal of Innovative Technology and Exploring Engineering (IJITEE)*, 8 (7), 2278-3075, 2019.

[5] H. Ashouri. 2015. "Finite element analysis of thermo mechanical stresses in diesel engines cylinder heads using a two layer viscoelasticity model", *International Journal of Automotive Engineering*, 5 (4).

[6] B. Sudarshan, and S. Roy, "Finite Element Analysis for Engine Cylinder head in Ansys", *International Journal of Scientific & Engineering Research*, 10 (6), 2229-5518, 2019.

[7] C.H. Cheng, "Numerical Modeling of the Power Cylinder System for Internal Combustion Engine with an Emphasis on Ring Pack Design", Doctoral Dissertation. Michigan State University. Mechanical Engineering, 2014.

[8] M. Fadaei, H. Vafadar, and A. Noorpoor, 2011. "New thermo-mechanical analysis of cylinder heads using a multi-field approach", *Scientia Iranica (B), Mechanical Engineering*, 18(1), 66-74, 2011.

[9] H. Ashouri. ''Thermo-Mechanical Analysis of a Coated Cylinder Head''. *Journal of Simulation & Analysis of Novel Technologies in Mechanical Engineering*, 2008-4927, 2017. HTTP://JSME.IAUKHSH.AC.IR.

[10] J.E. Scott, "Head Gasket Finite Element Model Correlation", *Master Thesis. University of Windsor. Department of Mechanical, Automotive and Materials Engineering*, 2013. https://scholar.uwindsor.ca/etd.

[11] W. Zhang, and X. Cao. "Structure Analysis and Improvement of Diesel Engine Block". *2nd International Conference on Electronic & Mechanical Engineering and Information Technology (EMEIT-2012) China*, 2013.

[12] M.X. Calbureanu, R. Malci, D. Tutunea, A. Ionescu, and M. Lungu. 2013. "The finite element analysis of the thermal stress distribution of a piston head", *International journal of*

mechanics, 4 (7), 2013.

[13] A.M. Schalch. "Structure Analysis of the Aluminum Cylinder Head for a High- Speed Diesel", *Society of Automotive Engineers,Inc. Conference*, 2007. Dol: 10.4271/2007-01-2562.

[14] L. Lei, and C.H. Bing-Wei. "Finite element analysis of thermal load on cylinder head of small-sized piston engine", *U.P.B. Sci. Bull., Series D*, 81 (2), 1454-2358, 2019.

[15] J. Hong, B. Li, Y. Chen, and H. Peng. "Study on the optimal design of engine cylinder head by parametric structure characterization with weight distribution criterion", *Journal of Mechanical Science and Technology*, 25 (10), 2607-2614, 2013.