View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by Universiti Teknologi Malaysia Institutional Repository

Finite Element Simulation of a Steady-State Stress Distribution in a Four Stroke Compressed Natural Gas-Direct Injection Engine Cylinder Head

A. Shamsudeen, S. Abdullah and A. K. Ariffin
Department of Mechanical and Materials Engineering,
Universiti Kebangsaan Malaysia, 43600 UKM Bangi,
Selangor, MALAYSIA
Tel: 03-89216013, Fax: 03-89259659
azhari@eng.ukm.my

Abstract: The main aim of this work is to predict the design performance based on the stress/strain and thermal stress behaviour of cylinder head under various operating conditions. The effects of engine operating conditions such as combustion gas temperature and maximum internal pressure, components initial temperature and engine speed on the stress and thermal stress behaviour of the cylinder head have been analyzed. The analysis was carried out using a finite element analysis (FEA) software package, MSC.NASTRAN which is use to simulate and predict the von-Mises stress and strain pattern and thermal distribution of the cylinder head structure during the combustion process in the engine and the geometry modelling was carried out using a popular computer-aided engineering tool, CATIA V5. The result can be used to determine the quality of the design as well as identify areas which require further improvement. In this investigation, structural analyses of the cylinder head highlight several areas of interest. The maximum stress is found not exceeding the material strength of cylinder head, and thus the basic design criteria, namely no yielding and no structural failure under firing load case, can be satisfied. In addition, the effect of thermal stress/strain provides a good indication on structural integrity and reliability of the cylinder head, which can be improved in the early stages of design. This steady-state finite element method (FEM) stress analysis can play a very effective role in the rapid prototyping of the cylinder head.
Keywords: Cylinder Head, Finite Element Analysis, Gas Pressure, von-Mises Stresses, Thermal

1. Introduction

Natural gas is increasingly seen as an effective alternative to gasoline and diesel fuel in many internal combustion (IC) engine applications. One of the major benefits of using natural gas as an engine fuel is that exhaust emissions can be reduced compared to the levels achievable with either gasoline or diesel fuel [1,2]. Natural gas has a high research octane number (RON>130), which allows combustion at higher compression ratios about 14:1; its higher hydrogen to carbon ratio offers much lower greenhouse gas emissions than those from the burning of other HCs [3]. Natural gas is a naturally occurring fuel found in oil fields, which primarily composed of about 90-95% methane (CH_4) and traces of additional compound including nitrogen, ethane and propane. Therefore it is attractive as an environmentally friendly alternative to petroleum. The combustion of methane involves only carbon-hydrogen bonds, and thus, is more likely to be more complete and produces less non-methane hydrocarbons. As comparison to gasoline, combustion of methane produces 22-25% lower CO_2 and particulate emission is very low relative to diesel fuel [4]. The most important factor in the conversion of a spark ignition (SI) engine from gasoline to natural gas operation as a monofuel fuel is the choice of compression ratio. Increase in the compression ratio (CR), increases pressure, temperature and mixture concentration of compressed gases. In addition, increasing the CR increases the surface to volume of combustion chamber at top dead centre (TDC) which should increase the heat transfer rates [3]. By these situations an appropriate engine design has to be examining in order to obtain a safe structure.

Finite element method (FEM) or finite element analysis (FEA) can be considered as one of the most powerful computer aided design tools for engineers. In the process of an engineering analysis a theoretical and numerical model is the starting point for researchers to develop and design an engineering system. In addition, the application of FEA tools in engine development can reduce the risk and the testing effort in producing the prototype [5,6,7]. Proper application of the FEM tools during the design phase has become essential nowadays in order to reduce lead time and cost of a new engine or vehicle model [8]. The ability to predict the design performance before it goes into production has become more important which relies on accurate computer models of all aspect of engine operation. In the design of an automotive engine, the cylinder head structure is among the most complicated part that characterises the type of the engine. Hence, modern tools and numerical techniques are required to simulate its physical behaviour and to evaluate its structural integrity, as well as to minimise production cost [7]. The cylinder head will have to withstand peak gas pressure and combustion temperature during the combustion process under firing load and this component needs careful attention during the design process due to avoid failure in its structure.

In addition, it is important to calculate the cylinder head temperature distribution in order to control thermal stresses and deformations within acceptable levels. The thermal analysis of cylinder head is important in many ways. First, the highest temperature of any point in cylinder head must not exceed more than 66% of the melting point temperature of aluminum alloy [9]. Aluminum alloys begin to melt at temperatures greater than 775 K [10] and the temperature distribution in cylinder head leads to thermal deformation and thermal stresses. In this design, both of thermal and mechanical stresses must be considered indicating the importance of cylinder head thermal analysis. With regards to thermal stress in cylinder heads, previous works includes investigations of thermal stresses development in engine component with isolative ceramic coatings by Bryzik et al. [11] and by Kamo and Bryzik [12]. Chyuan proposed a finite element model of a cylinder structure with a twin-cam 16-valve [7]. By using an effective calculating method via the MSC.NASTRAN software, he can predict the analytical thermal and stress and strain results at various loading conditions and operating environments. Liu et al. [13] made use of the components computer programme (HCC) to analyse the transition of heat transfer for an engine. Their results revealed that, compared to the FEM, the distribution of the temperature in the walls of the combustion chamber could be predicted accurately by the HCC method. The approach that was applied to the actual problems of heat transfer, such as the Isuzu and Caterpillar engines showed correct results in comparison with the experiments.

Therefore, the commercial FEM software, MSC.NASTRAN, is employed to perform numerical simulation on the structural analysis. This work is based on heat transfer theory and thermal stresses analysis in order to investigate the stress/strain behaviour of a 1.6-liter cylinder head engine model which is subjected to direct injected natural gas combustion pressure under various loading condition. The FEA will give results that can represent the design quality according to the von Mises criteria [13] and identify areas that need further attention. The results of this research can be used to develop a design procedure for a new direct injection natural gas engine.

2. Design Review and Materials

The original cylinder head model was obtained with courtesy of the manufacture of the base engine. The model was in the CATIA V4 format and was converted into a CATIA V5 format. This CAD model was then modified to make retrofitting of gas injectors along side the spark plug possible in the centre of combustion chamber, and was used for the 3D solid geometry modelling. The geometric modelling of this cylinder head was shown in Fig. 1 and 2. Because of the complexity of geometrical design, only a quarter- or one-cylinder model was considered for FEA analysis.

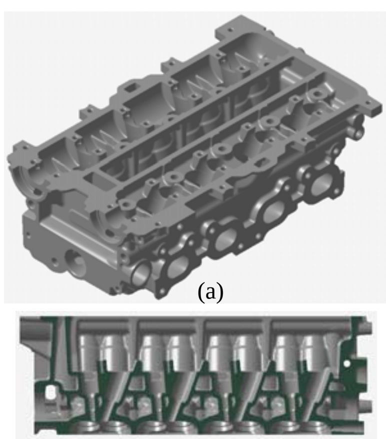


Fig. 1: (a) Full model and (b) section view of direct injection-natural gas cylinder head.

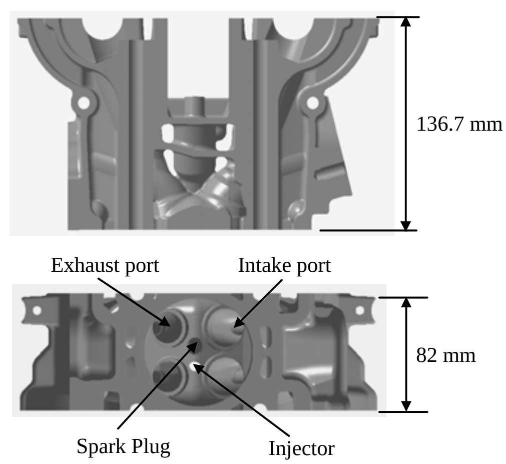


Fig. 2: Quarter model of the 1.6-liter 2nd cylinder head.

With this new design configuration, the combustion process inside the cylinder will be more efficient and completely burned in order to achieve the best performance and as well as to reduce the exhaust emissions. Nowadays, most of the automotive cylinder head engine is made by aluminium alloy because of their higher thermal conductivity, generally operate about 30-80% cooler than equivalent cast-iron as well as lighter than other materials. In this work, the material use for the cylinder head is aluminium alloy types A356-T6. Table 1 lists the material properties of cylinder heads used during the FEA simulation.

Table 1: Material properties of the cylinder head for structural analysis [14]

Mechanical properties	Value
Compressive yield strength	172 MPa
Endurance limit	59 MPa
Modulus of Elasticity $\times 10^3$	72 MPa
Poisson ratio	0.33
Melting range	560 – 610ºC
Density	2713kg/m^3
Coefficient of Thermal expansion per ºC at 20-100ºC	0.0000214
Thermal Conductivity cal/cm^2·K at 25ºC	0.36

3. Finite Element Modelling

1) Finite Element Method

The finite element method (FEM), which has been the core of a FEA software, is based on the idea of building a complicated object with a simple blocks or dividing a complicated object into a small and a manageable pieces, which is also known as element with nodes [14]. In this method of analysis, a complex region or structure is discretised into simple geometric shapes called finite elements. In addition, FEA meshing is the one of the finite element process, which requires special techniques in order to produce a good mesh element that will give an accurate result for the design. For convenience during the simulation of the stress and thermal behaviour, only a quarter of the cylinder head with four valves will be considered in this work. This approximation can lead to several advantages, including: (a) reduction of complications during specifying boundary conditions in the analysis and (b) economising on the element counts of the finite element analysis to reduce simulation time.

2) Numerical Modelling

The first stage of the process is to define the model geometry. This can be accomplished using engineering drawings and three-dimensional solid models. The geometric modelling was carried out using a computer-aided design (CAD) tool, CATIA V5. Furthermore, mechanical boundary conditions and model constraints are defined, approximated and calculated to represent the real scenario. Thermal and mechanical boundary conditions are applied to the finite element model using a computer simulation program [15,16]. Thermal and mechanical boundary conditions are applied to the finite element model using a computer simulation program. The finite element analysis was executed using a commercial finite element analysis software package, MSC.NASTRAN. The analysis is linear using a static gas pressure loading and in steady state condition, but in future this analysis will be recommend to the non-linear analysis and transient conditions. Following the finite element analysis, the results are post processed into a form suitable for engineering assessment that accesses the analysis code's binary database and extracts the appropriate results.

3) Geometry Definition and Mesh Generation

The CATIA V5 software is used for geometry modelling. For the thermal analysis, we need a heat and temperature parameters from the some area inside cylinder head that exposed to the high temperature and pressure during combustion process such as in the area of intake port, exhaust port, spark plug, injector and water jacket. All these parameters are requiring in analysis in order to simulate a real scenario of cylinder during loading condition. For a good quality of mesh and results, some of the critical areas in the cylinder head were defined with higher order curves and surfaces. The shape around the deck of the combustion chamber that provides the location of the intake and exhaust valves, the spark plug and the injector was an area of main concern and finer mesh was used in order to get accurate stress and thermal distribution. Hence, a quarter finite element model of the cylinder head structure with 243,096 solid elements and 402,704 nodes was produced as shown in Fig. 3. The regions which are not critical such as the near of the top of head, were meshed with coarse mesh to reduce the number of element and CPU time required to simulate the problem. The finite element analysis was executed on a high performance computer; SGI Origin 300 computer system with 4 CPU, and the CPU time required for a static run under, as single load case was about 55 minutes. Before the simulation began, several consumptions are made with regards to the modelling the cylinder head structure [7,17], i.e., (1) the 4 cylinder heads possess a structural symmetry in their entity, hence the 2nd cylinder head was removed from the complete model in order to reduce the calculated time and simpler boundary conditions can be enforced, and (2) the maximum gas pressure of the combustion process was used for the firing loading in steady state condition.

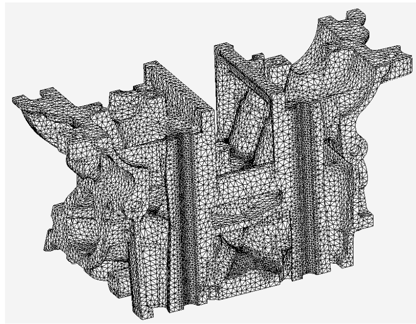


Fig. 3: A through model with 3D solid element mesh for the 2nd cylinder head of the 1.6 L engine.

4) Thermal and Pressure Boundary Condition

In thermal analysis, boundary conditions may vary significantly both in space and time particularly for combustion chamber components. The boundary conditions or stress analysis combine the results from thermal analysis and displacement boundary conditions suitable for the cylinder head. In this analysis only thermal loading and pressure loads from combustion chamber were considered. The gas pressure created as a result of the firing of the spark plug is imposed on the surface of the combustion chamber. For the investigation of cylinder head engine structure, the maximum loading conditions in the process of engine operation for the 2000-4000 rpm will be considered. In this case, Lee et al. [17] mentioned that the effect of temperature distribution on the main body of the cylinder head should be brought into the boundary conditions of heat transfer for simulating the operation situation. Therefore the heat transfer analysis must be performed prior to the structural analysis because of the demand for observation of the thermal stresses that are generated. Constraints also were set on the boltholes of the cylinder used to fix the cylinder head to the cylinder block during the gas pressure loading.

a) Boundary Conditions for Intake and Exhaust Ports

In an internal combustion engine, there are three hottest points are found they are around the spark plug, the exhaust port and valve, and the face of the piston. These places are exposed to the high-temperature combustions gases and they are difficult places to cool. The exhaust valve and port operate hot because they are located in the pseudo-steady flow of hot exhaust gases and create a difficulty in cooling. In this work, a temperature sensor such as thermocouple, with a time constant will give a pseudo-steady state temperature of the flow. This thermocouple temperature will be approximately an enthalpy average temperature and not necessarily a true time average:

$$T_{\text{thermocouple}} = \frac{\int \dot{m} c_p T \, dt}{\int \dot{m} c_p \, dt} \qquad (1)$$

where $\dot{m}$ is mass flow rate of exhaust, t is time, c_p is specific heat and T is temperature. In this work, the average temperature for the intake and exhaust port is 60^{o}C and 554^{o}C results to from the experimental of single research engine with engine speed of 2000 rpm, is considered as a boundary temperature. Exhaust temperature of an engine will increase with higher engine speed or load, with spark retardation, or with an increase in equivalence ratio.

b) Boundary Conditions for Spark Plug and Injector

During combustion occur the highest temperature is happen around the spark plug. In natural gas engine operation, its need higher sparks ignition energy to ignite the fuel because of their higher flash point about 905 K and higher compression ratio around 14:1, as compared to the similar gasoline engine which is about 572K with a compression ratio of 10:1. For the injector, the temperature is lower than gasoline because the fuel itself is stored in high pressure. For the normal quasi-steady state, the temperature of the spark plug electrodes in-between firings should be around 600^o to 700^{o}C [4,10]. For this simulation, the average temperature of spark plug and injector used is 600^{o}C and 20^{o}C.

c) Boundary Conditions for Water Jacket and Combustion Chamber

In term of water jacket and combustion chamber boundary conditions, the CFD simulation is used to determine the temperature distribution and the heat flux occurred during the combustion process. All of the engine parts are assumed at an initial temperature of 21^{o}C and the temperature of cooling water is not time-independent (steady). The boundary condition of combustion chamber is determined from CFD calculation in the forms of temperature and heat flux results.

d) Boundary Conditions for Stresses

In order to decrease the complexity of the boundary conditions, the interaction between the cylinder head, cylinder head gasket, cylinder head bolts and cylinder block was not modelled. Boundary condition of the bolts is very significant since there is an evidence of a pre-load and a pre-distance occurred in the actual modem, in which the cause of this pre-load is due to over-tightened bolts. Pre-load and constraint of other bolt and screw has not been modelled. Other boundary condition is pressure load is that at a field is applied on the combustion chamber deck. The maximum amount of pressure is 5.8 MPa (2000rpm), which is close to the experimental results (Fig. 4). The gas pressure is varied according to the engine speed, and base on the experimental, the pressure will decrease after 4000rpm.

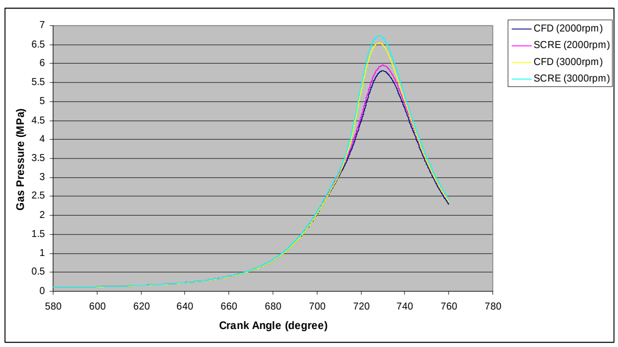
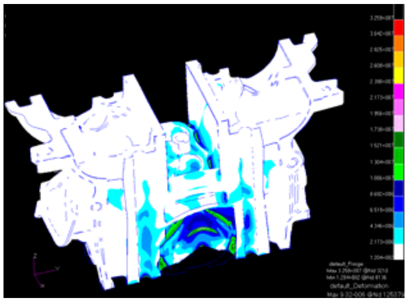


Fig. 4: The gas pressure of 1.6 L engine at 2000 and 3000 rpm compared to the experimental results.

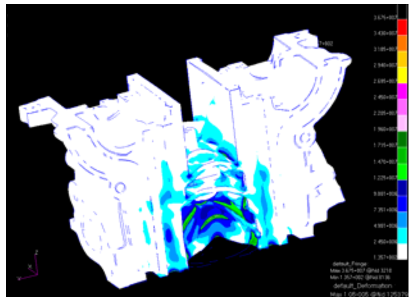
4. Result and Discussions

1) Stresses and Deformation Distributions

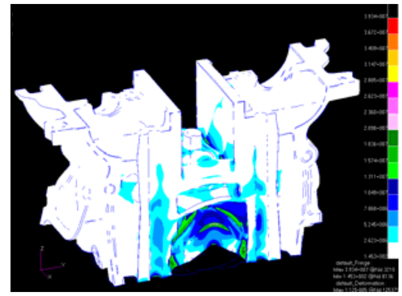
FEM steady state predictions of stresses and deformation distributions in the cylinder head are shown in Fig. 5 for a range of engine speeds. Fig. 5 indicates that the region of fire deck and the area around inlet and exhaust valve seat inserts experience considerably higher stresses and deformation than the rest of the fire deck which is contributed by local bending moment. The maximum stress varies from 32.59 MPa at 2000 rpm to 39.34 MPa at 4000 rpm as an increasing maximum gas pressure with increase engine speed. Meanwhile, the minimum von Mises stress is located at the areas between the cylinder head bolts and the top of cylinder head, which is due to local compressive loads. The combustion chamber deck shows the largest deformation under gas pressure load varies from 0.009324mm at 2000 rpm to 0.01125mm at 4000 rpm according to the highest stresses and strain. Majority of the high stresses are compressive and originated from the cylinder head bolt preload. Due to different sizes between intake and exhaust valves, the distribution of these compressing stresses is slightly different between the inlet and exhaust side of the cylinder head.



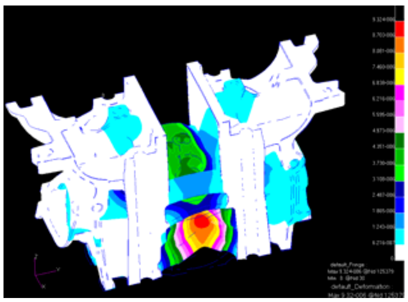
(a) Predicted von Mises stress at 2000 rpm; min = 120.4 Pa, max = 32.59 MPa



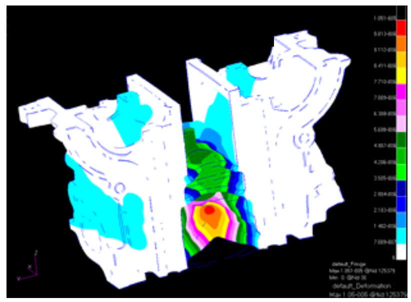
(b) Predicted von Mises stress at 3000 rpm; min = 135.7Pa, max = 36.75MPa



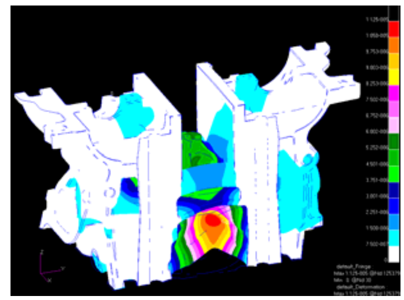
(c) Predicted von Mises stress at 4000 rpm; min = 145.3Pa, max = 39.34MPa



(d) Predicted deformation at 2000 rpm; max = 0.009324 mm



(e) Predicted deformation at 3000 rpm; max = 0.01051 mm



(e) Predicted deformation at 4000 rpm; max = 0.01125 mm

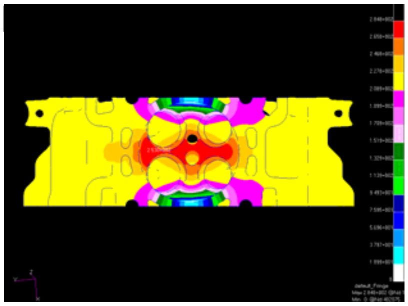
Fig. 5: Steady state von Mises stresses and deformation contours of the cylinder head.

The distribution of the stresses in the cylinder head is almost larger than the original cylinder head for the same engine prior to modification even though both used the same gas pressure. However, the design of the cylinder head is still above the safety factor specified in the design criteria, i.e. with no yielding and structural failure under firing load case. Another important result from this analysis is that the predicted von Mises stresses exceed the elastic limit for the cylinder head material. The red and orange coloured areas in Fig. 5 represent regions of stresses that are under the yield strength of the material. The pressure load in the cylinder head is equal but, in the area where thickness is maximum, the pressure rise is larger and so as the stress. This is the reason why high stress occurred in the valve bridge and near the valve seat.

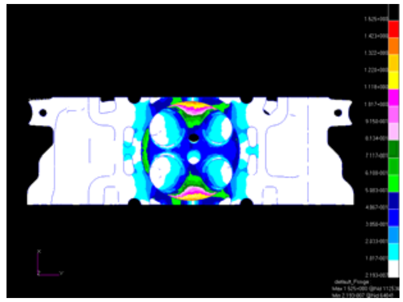
From the results, the middle deck around the valve guides had been pushed up during the firing gas loads. Engineering experience suggests that this region is prone to structural failure due to firing load if detailed design or material properties is not cautiously specified [7]. The narrow bridge of deck between the valve guide and the oil passage should be free of stress raisers on both sides by using generous fillets. Besides, the surface finish in this area should be carefully investigated during the quality assurance procedure to ensure that porosity or sand insertions are avoided. The cool water jacket in this area may be reduced to give some thickness to section at the deck or the upper surface may be corrugated to give the same effect.

2) Thermal Distribution

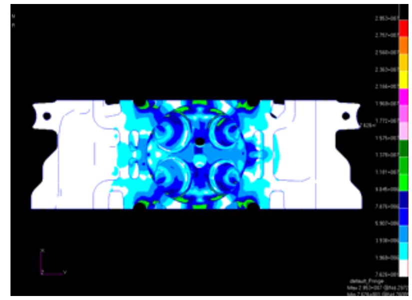
The steady state prediction of temperature, heat flux and thermal stress distributions in the cylinder head at 2000 rpm are shown in Fig. 6. From the result, it shows that the region of the fire deck and the area around valve seat considerably higher temperatures and thermal stresses about 284.8^{o}C and 29.55 MPa cause of a high temperature gradient in the surface of combustion chamber. The temperature and heat flux will be increase as a result of increasing gas temperature and pressure, and heat transfer with increased engine speed [16]. Also, it can be noticed that as speed increasing, the hot region propagates from the exhaust valve side to the intake valve side. It is important to examine the largest temperature gradients occur between the exhaust valve seat and the spark plug and injector hole as well as the valve bridge between two valves. At the lower engine speed, the spark plug and injector hole, and the valve bridge are surrounded by uniformly lower temperatures. However, the propagation of the hot front increases with the speed resulting in an asymmetric exposure of those critical regions to hot and cold temperatures. It is therefore anticipated that the highest thermal stresses should concentrated on either the spark plug and injector hole or the valve bridge region, as thermal stress linearly depends on the temperature gradient.



(a) Predicted temperature at 2000 rpm; max = 284.8°C



(b) Predicted heat flux at 2000 rpm; max = 1.525MW/m^2



(c) Predicted thermal stress at 2000 rpm; max = 29.55 MPa

Fig. 6: Steady state temperature and thermal stress distributions of the cylinder head.

5. Conclusions

In this paper, the structural and thermal analysis of a cylinder head under gas firing loading for varies an engine speed at 2000-4000 rpm were carried out by using FEA. A 3D solid model of cylinder head was used to obtain the numerical simulation. For the cylinder head, the structural analysis highlighted several area of interest and the maximum stress in the cylinder head is found not to exceed the allowable compressive yield strength of cylinder head design material, i.e. 172MPa and below the endurance limit for that material, and thus the basic design criteria, i.e. no yielding and structural failure under firing load case, can be satisfied. In addition, FEA simulation also was conducted to predict the temperatures and heat fluxes distribution under a series of steady state operating condition. The results show that the higher temperature is occurred around the spark plug and the injector hole, and near the valve bridge and the valve seat, which is due to material thickness in the valve bridge. Although the thickness is quite significant, it still under the thermal limit for this type material. All these analyses can be used to assist the automotive engineers to design of a more reliable cylinder head for a new engine.

6. Acknowledgements

The authors would like to thank the Ministry of Science, Technology & Innovation of Malaysia for sponsoring this work under the IRPA project no. 03-02-02-0057-PR0030/10-04.

7. References

[1] R. L. Evans & J. Blaszczyk, 1997. A comparative study of the performance and exhaust emissions of a spark ignition engine fuelled by natural gas and gasoline, *Proc. Instn Mech Engrs Vol 211* Part D, 39-47.

[2] M. Hiroshi, K. Keizi, N. Kenro, S. Hiroshi & H. Kawamura, 1996. Structure and performance of heat insulated natural gas engine, JSAE Review **18**: 377-384.

[3] A. Das & H. C. Watson, 1997. Development of a natural gas spark ignition engine for optimum performance, *Proc. Instn Mech Engrs Vol 211* Part D, 361-378.

[4] Ferguson, C.R. & Kirkpatrick, A.T., 2001. *Internal Combustion Engines: Applied Thermoscience 2nd Ed.*, John Wiley, New York, pp. 323-325.

[5] N. Mamiya, 1999. Application of FEM Analysis in Engine Development, MSC. Worldwide Automotive Conference, Nissan Motor Co. Ltd. Japan.

[6] L. Baverstock, 2002. Investigation of Diesel Engine Cylinder Block Structural Performance using Design of Experiments, Ford Motor Company Limited, Altair Engineering Ltd.

[7] S.W. Chyuan, 2000. Finite Element Simulation of a Twin-Cam 16-valve cylinder structure, *Finite Elements in Analysis and Design*, **35**: 199-212.

[8] H. Petrin & B. Wiesler, 1999. Application of FEM-Tools in the Engine Development Process", *1st Worldwide MSC Automotive Conference*, Munich, Germany.

[9] *Properties and Selection: Irons, Steels and High Performance Alloy*, ASM Handbook, vol. 1, ASM International, 1990.
[10] W.W. Pulkrabek, 2004. *Engineering Fundamentals of the Internal Combustion Engine*, 2nd Edition, Prentice-Hall, New Jersey.
[11] W. Bryzik, W. Woods, E. Schwarz & P. Glance, 1989. High Temperature Engine Component Exploratory Design Development, *SAE Technical Paper no. 890296*.
[12] R. Kamo & W. Bryzik, 1990. Performance and Durability of a Ceramic Coated Adiabatic Engine, *ETCC Symposium*, American Society of Mechanical Engineers, New Orleans.
[13] L. Yong & R.D. Reitz, 1998. Modelling of heat conduction within chamber walls for multi dimensional internal combustion engine simulation", *Int. J. Heat Mass Transfer*, **41**: 859-869.
[14] C. A. Harper, 2001. *Handbook of Materials for Product Design*, McGraw-Hill Companies, New York.
[15] T.R. Chandrupatla & A.D. Belegundu, 2002. *Introduction to Finite Elements in Engineering*, Prentice-Hall, New Jersey.
[16] M.H. Shojaefard, A.R. Noorpoor, S. Alizadehnia & M. Ghaffarpour, 2005. Numerical Analysis of an ICE Cylinder Head, *Proc. Int. Conf. Recent Advances in Mechanical & Materials Engineering*, Tehran, Iran.
[17] E. Danielson, J. Elwart, W. Bryzik & D. Turner, 1992. Thermo dynamical stress analysis in a low heat rejection cylinder head, *SAE* 929544.
[18] C.C. Lee, K.N. Chiang, W.K. Chen, & R.S. Chen, 2005. Design and Analysis of gasket sealing of cylinder head under engine operation conditions, *Finite Element in Analysis and Design*, **41**: 1160-1174.