International Conference on Mechanical Engineering Research (ICMER2013), 1-3 July 2013
Bukit Gambang Resort City, Kuantan, Pahang, Malaysia
Organized by Faculty of Mechanical Engineering, Universiti Malaysia Pahang
Paper ID: P161

DESIGN AND SIMULATION OF A CYLINDER HEAD STRUCTURE FOR A COMPRESSED NATURAL GAS DIRECT INJECTION ENGINE

A. Shamsudeen[1*], S. Abdullah[1,2], A.K. Ariffin[1,2] and Y. Ali[3]

[1]Centre for Automotive Research, Faculty of Engineering and Built Environment
University Kebangsaan Malaysia, 43600 Bangi Selangor, Malaysia
Phone : +60389216454; Fax : +60389118314
*Email: azhari@eng.ukm.my
[2]Department of Mechanical and Materials Engineering, Faculty of Engineering and Built Environment, University Kebangsaan Malaysia,
43600 Bangi Selangor, Malaysia
[3]Faculty of Mechanical Engineering, University Malaysia Pahang
26600 Pekan, Pahang, Malaysia

ABSTRACT

In this work, the design and simulation of four strokes, spark ignition engines with compressed natural gas direct injection is studied based on finite element analysis. The design is performed using CATIA V5, and simulated using MSC.Patran. Based on simulation results, it shows that the optimum design is achieved. The maximum von Mises stress at 5400 rpm is about 39.84 MPa for single load and 56.16 MPa for assembly load. Overall, these maximum stresses are better than original model. For the displacement at 5400 rpm, the maximum value is 0.01296 mm (single load) and 0.00972 mm (assembly load) with a decrease for about 3.55% compared to the original model. Both maximum stresses and displacement are not exceeding the material strength in order to avoid any components failure during engine operation. The result can be used to assess the quality of the design i.e. no yielding and structural failure under firing load case, can be satisfied.

Keywords: Cylinder head; finite element; direct injection; compressed natural gas.

INTRODUCTION

Both the design and development of the automotive engine are complicated process. To acquire the best performance and optimize design of engine in any operating condition, many analytical tools and experimental works are used to find the optimum structure for engine design (Lee et al., 2005). In the automotive engine, the cylinder structure is the most important component affecting the performance of a car and involves many components. All these components need to careful attention especially during the assembly design in order to avoid any failed structure at engine operations (Liu, 2013). The cylinder head of direct injection engines is highly influenced by the geometry of combustion chamber injector location. In fact, a small change in geometry can lead to considerable changes in the mixture distribution and performance of the engine (Yadollahi, 2013).With the increasing emphasis to achieve improvements in automotive fuel economy, automotive engineers are motivated to develop new engines to enhance fuel consumption, power and reduce emission (Zhao et al., 1999; Yadollahi, 2013).

In addition, due to the increasingly aggressive legal emissions levels, the use of alternative fuels, including compressed natural gas (CNG), has increased along with the importance of developing alternative fuel technology. In the SI engine, direct injection (DI) technology significantly increases the engine volumetric efficiency and decreases the need of throttle valve for control purposes (Liu, 2013). All these technology were developed to overcome the problems of using CNG especially about the board storage problem and reduced performance compare to gasoline (Durell, 2000). However, by the design and development of compressed natural gas direct injection (CNGDI) engine, it will be managed to provide the improved of engine performance and as well as exhaust emissions.

In the engineering analysis, a theoretical and numerical model is the starting point for researchers to develop and design an engine component (Fadaei, 2011). In addition, the application of finite element analysis (FEA) tools in engine development can reduce the risk and the testing effort in producing the prototype (Baverstock, 2002; Chyuan, 2000). Other than that, the combination of computational fluid dynamic and FEA analysis can also be performed to predict with precise stress/temperature distribution and thermo-mechanical cycles that can affect internal combustion engines under actual operating conditions (Stefano, 2013).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 (Fadaei, 2011).

Therefore, the commercial software, MSC.Patran, is employed to perform numerical simulation on the structural analysis. This work is based on pressure loading in order to investigate the stress/deformation behaviour of cylinder head structure which is subjected to direct injected natural gas combustion pressure under various loading condition. The results of this research can be used to develop a design procedure for a new direct injection CNG cylinder head engine.

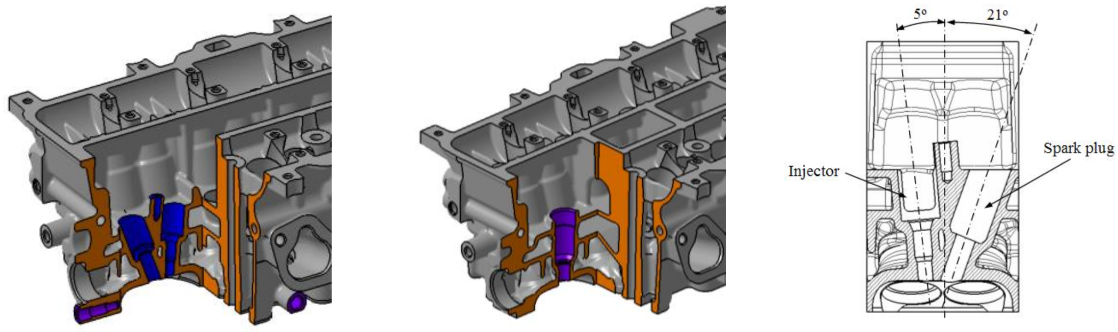
CYLINDER HEAD MODELLING

The original and CNGDI cylinder head design material used in this simulation are shown in Tables 1. In this work, the material use for the original cylinder is aluminium alloy types AC4B and for the new cylinder head is aluminium alloy types A356-T6. However, for the simulation, both designs will be used only material A356-T6.

Table 1. Specification for the cylinder head model.

Material Properties	**Value**	
	AC4B	**A356-T6**
Modulus of elasticity (MPa)	71×10^3	72×10^3
Compressive yield strength (MPa)	166	172
Tensile yield strength (MPa)	159	164
Poisson’s ratio	0.33	0.33
Density (kg/m^3)	2740	2713

The original and modified cylinder head model and cross section area of spark plug and injector are shown in Figure 1. The design specifications of the cylinder head were shown details in Table 2. This original model was modified to make retrofitting of gas injectors alongside the spark plug possible in the centre of combustion chamber, and was used for the 3D solid geometry modelling. Because of the complexity of geometrical design, only a quarter or one-cylinder model was considered for analysis. 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.



(a) (b) (c)

Figure 1. CAD model of (a) original cylinder head; (b) modified cylinder head; (c) cross section area of spark plug and injector.

Table 2. Specification for the cylinder head model.

Specification	Cylinder Head Design	
	Original	Modified
Cylinder height (mm)	134.7	136.7
Intake valve angle (deg)	21.5	20.95
Exhaust valve angle (deg)	20.5	20.05
Spark plug angle (deg)	90	21
Injector angle (deg)	34 (at intake manifold)	5 (at centre cylinder)

FINITE ELEMENT MODELING AND VALIDATION

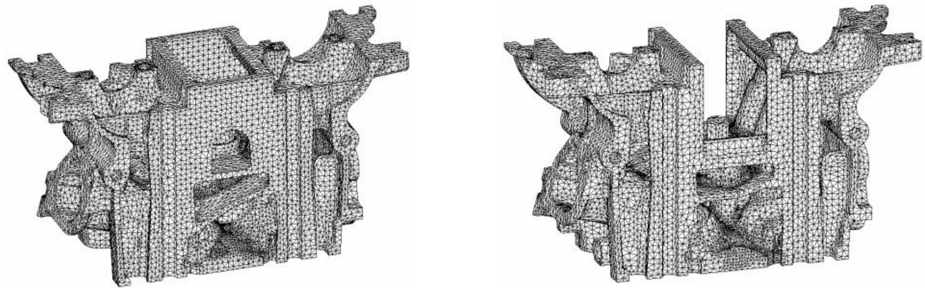
Finite Element Analysis

The FEA 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 (Harper, 2001). 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 for analysis to reduce simulation time.

Geometry Definition and Mesh Generation

For structural analysis, pressure from combustion process will be used as a load for the analysis. 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 distribution. Hence, a model of the cylinder head with 243,096 solid elements and 402,704 nodes was produced as shown in Figure 2.

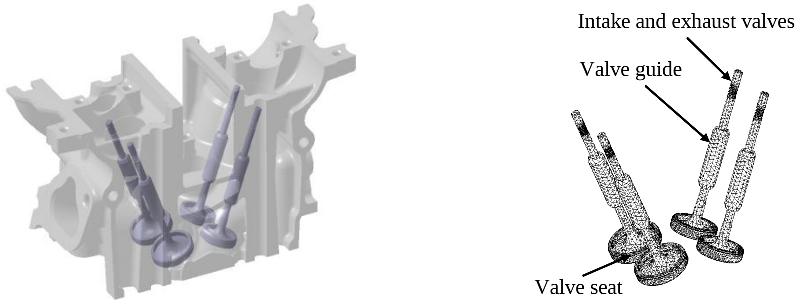
The FEA was executed on HP Workstation xw6200 and interface with a high performance computer; SGI Origin 300 (4 CPU). The CPU time required for a static run under as single load was about 2 hours and assembly load case was about 2.5 hours.



(a) (b)

Figure 2. A 3D solid element meshes (a) original model; (b) modified model.

Before the simulation began, several consumptions are made with regards to the modelling the cylinder head structure (Danielson et al., 1992; Chyuan, 2000) 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. Besides that, the assembly model (see Figure 3) also will be simulate in order the get a better result in term of stresses and displacement, and to ensure their value were not exceed the strength and deformation limit of the material strength.



(a) (b)

Figure 3. Assembly modeling (a) assembly of sub component to cylinder head; (b) 3D solid element mesh for valve guide, valve seat and valves.

Boundary Conditions for Stresses

In order to decrease the complexity of the boundary conditions, the interaction between the cylinder head gasket, cylinder head bolts and cylinder block was not modelled. Constraints were set on the bolt of the cylinder head used to fix the cylinder head to the cylinder block. The pressure load is applied on the combustion chamber deck. The maximum pressure load is 5.8 MPa at 2000 rpm which is close to the experimental results which based on single cylinder research engine (SCRE) as shown in Figure 4. The gas pressure is varied according to the engine speed, and base on the experimental, the pressure will increase with increasing the engine speed but for the very high engine speed around 6000-7000 rpm, the pressure will be decrease because of the insufficient fresh air through to the engine for the combustion process.

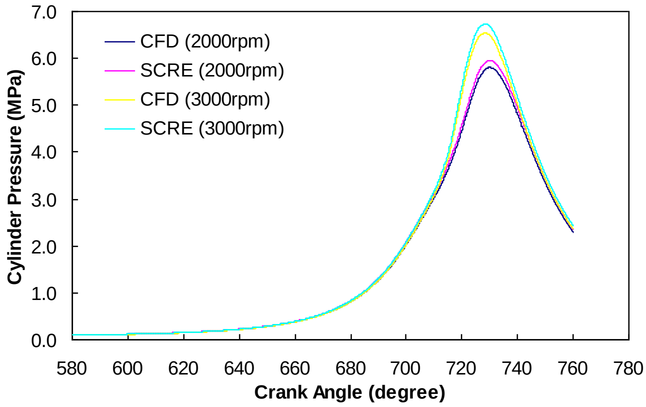
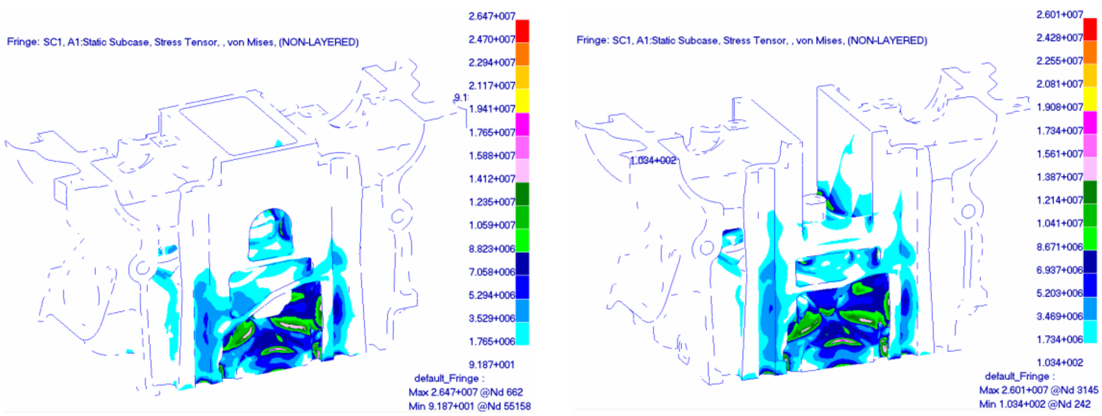


Figure 4. The gas pressure of CNGDI and SCRE engine at 2000 rpm and 3000 rpm.

RESULTS AND DISCUSSION

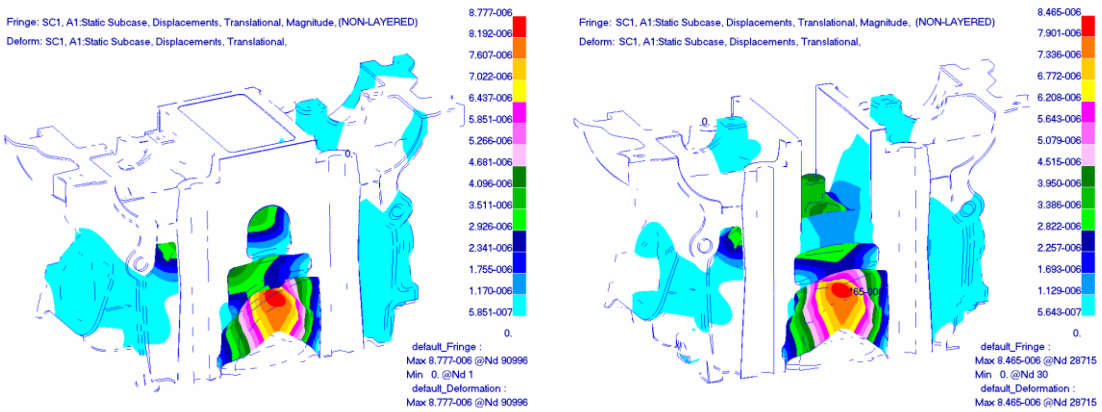
Stress and deformation analysis for single load case

FEA steady state predictions of stresses and deformation distributions in the cylinder head are shown in Figure 5 for an engine speeds 1000 rpm. The maximum stress for modified cylinder head varies from 26.01 MPa at 1000 rpm to 39.84 MPa at 5400 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 0.008465 mm at 1000 rpm to 0.01296 mm at 5400 rpm as shown in Figure 6. 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) (b)

Figure 5. Stress distribution at 1000 rpm (a) original design; (b) modified design.



(a) (b)

Figure 6. Deformation contours at 1000 rpm (a) original design; (b) modified design.

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 (Chyuan, 2000). 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.

In totally, the design of CNGDI cylinder head has be improved about 1.72% of their stresses and 3.56% of their displacement compare to the original design (see Figure 7). However, both designs still below under the compressive yield strength limit i.e. 172 MPa. Hence, the purpose of the base design which, the maximum von Mises stress for both designs are not exceed or above the material strength limit, the cylinder head design can be satisfied i.e. with no yielding and structural failure under firing load case. In term of safety factor, both of the cylinder head designs are safe and fulfilled the nominal safety factor for the base design especially for the engine component. For the CNGDI cylinder head, the safety factor is about 4.32 compare to the original design is about 4.09. These safety factors were determined base on the maximum von Mises stress at 5400 rpm and compressive yield strength by using Eq. (1):

$$N = \frac{S_y}{\sigma_v} \tag{1}$$

where N is safety factor, S_y is yield strength and σ_v is von Mises stress.

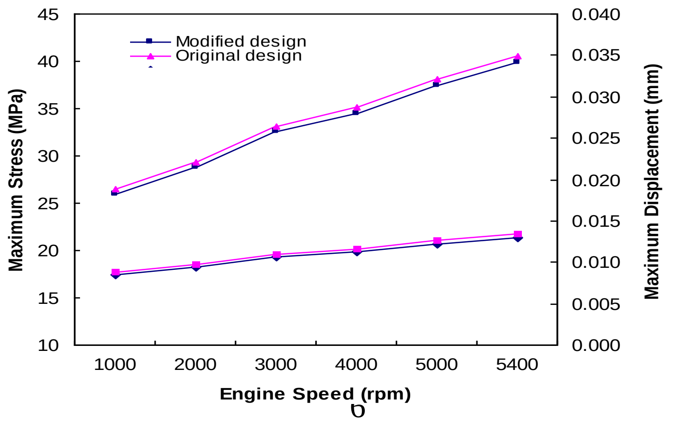
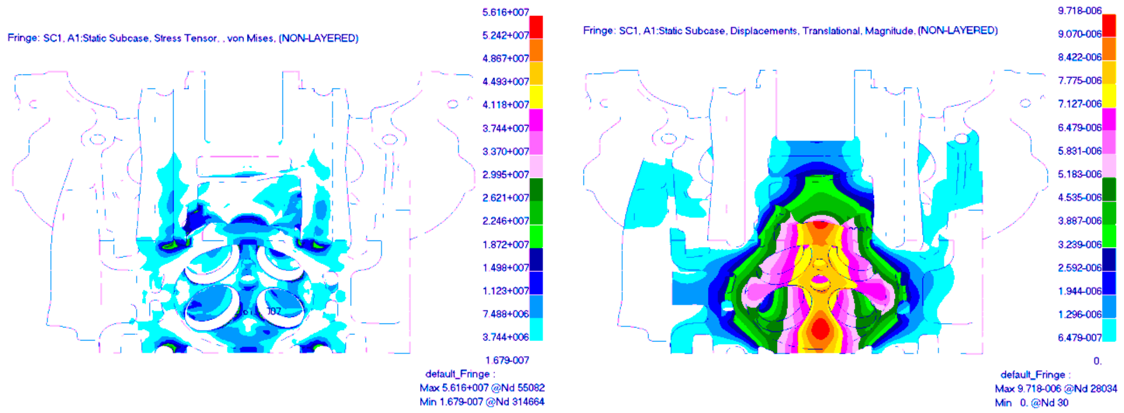


Figure 7. Maximum stress and displacement with several of engine speed.

Stress and deformation analysis for assembly load case

Under assembly loads on cylinder head, valve seats and valve guides, the maximum von Mises stress value is 56.16 MPa at 5400 rpm located at the interface area between valve seat and cylinder head deck because of local bending moment as shown in Figure 8. Meanwhile, the maximum displacement distribution of cylinder head is 0.00631 mm at 1000 rpm and increase to 0.00972 mm at 5400 rpm. 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. The result of the simulation shows that occur higher stresses but lower displacement distribution to the cylinder head design with the assembling of valve guides and valve seats. For engine speed at 5400 rpm, the different maximum stress can reach about 29.1% but the maximum displacement can reduce about 33.4% compare to the single load case. However, the both of the design are still below of the material yield strength and deformation limit (0.035 mm for valve guide, 0.045 mm for intake valve seat and 0.049 for exhaust valve seat) as shown in Figure 9.



(a) (b)

Figure 8. (a) Stress and (b) deformation contours at 5400 rpm for modified design

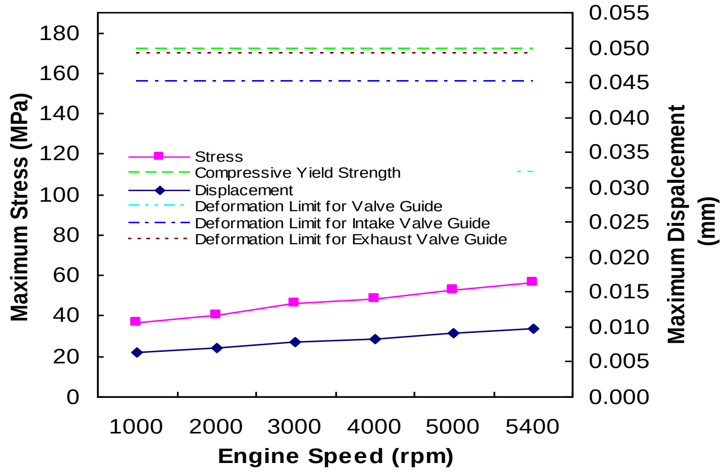


Figure 9. Maximum stress and deformation limit for cylinder head design.

CONCLUSION

In this paper, the structural of a cylinder head under gas firing loading for varies an engine speed at 1000-5400 rpm were carried out by using FEA. For the CNGDI cylinder head, the structural analysis highlighted several area of interest and the maximum stress in the new cylinder head is found not to exceed the allowable compressive yield strength of cylinder head design material, i.e. 172 MPa and below the deformation limit i.e. 0.035 mm. Thus, the basic design criteria, i.e. no yielding and structural failure under firing load case, can be satisfied. In addition, the strength and displacement of CNGDI cylinder head design was improved about 1.72% and 3.55% decrease compared to the existing design. All these analyses are able to be used to assist the automotive engineers to design of a more reliable cylinder head for a new engine.

ACKNOWLEDGEMENTS

The authors would like to thank the Malaysian Ministry of Science, Technology & Innovation for sponsoring this work under project IRPA 03-02-02-0057-PR0030/10-04.

REFERENCES

B. Yadollahi & M. Boroomand. 2013. The effect of combustion chamber geometry on injection and mixture preparation in a CNG direct injection SI engine. Fuel, 107 (2013) 52–62.

Baverstock, L. 2002. Investigation of diesel engine cylinder blocks structural performance using design of experiments. Technical Review, Ford Motor Company Limited, Altair Engineering Ltd, pp. 1-9.

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

Durell, E., Allen, J., Law. D. & Heath, J. 2000. Installation and development injection system for a bi-fuel gasoline and compressed natural gas engine. Proc. of 7th International Conference and Exhibition on Natural Gas Vehicles, p. 311-316.

Harper, C.A. 2001. Handbook of materials for product design. New York: McGraw-Hill Companies.

Lee, C.C., Chiang, K.N., Chen, W.K. & Chen, R.S. 2005. Design and analysis of gasket sealing of cylinder head under engine operation conditions. Finite Element in Analysis and Design, 41: 1160-1174.

M. Fadaei, H. Vafadar, & A. Noorpoor. 2011. New thermo-mechanical analysis of cylinder heads using a multi-field approach. Scientia Iranica B, 18 (1), 66–74,

Stefano Fontanesi & Matteo Giacopini. 2013. Multiphase CFDeCHT optimization of the cooling jacket and FEM analysis of the engine head of a V6 diesel engine. Applied Thermal Engineering, 52 (2013) 293-303.

Yu Liu, Jeongkuk Yeom, Seongsik Chung. 2013. A study of spray development and combustion propagation processes of spark-ignited direct injection (SIDI) compressed natural gas (CNG). Mathematical and Computer Modelling, 57 (2013) 228–244.

Zhao, F., Lai, M.C., & D.L. Harrington. 1999. Automotive spark-ignited direct-injection gasoline engines. Progress in Energy and Combustion Science, 25: 437–562.