Numerical Analysis of Thermal Expansion and Nonlinear Distortion Effects on Performance of Engine Seal under Compressive Loading

Rana Ali Hussein

Al Mussaib Technical College, Al Furat Al Awsat Technical University, Kufa, Iraq
ra621057@gmail.com

Abstract: In internal combustion engines, one of the most critical factors in engine efficiency is the sealing unit. All mating surfaces should be seal very well by using gasket assembly to keep the closed cycle system during operation in different environmental conditions and loads operation. The problems of inadequate and non-suitable gasket selection and also the leakage problems due to the non-uniform load distribution will diminish the engine pressure and minimize the engine performance and finally to damage the components. The main objectives of this research are to find out the most effecting factors on the gasket unit performance by analysis and simulation method. Finite element method under (Abaqus/CAE) code has different solutions for such types of nonlinear behaviour. Modelling of these complex nonlinear behaviour's like loading along contact paths, nonlinear elasticity and plastic distortion are available in Abaqus. It can simplify the components and in same time the essential elements properties will maintain the nonlinear response. In this research, the completed assembly model of gasket sealing unit is consist of upper aluminium manifold cover, two bolts, lower manifold block, paste gasket with silicone bead and base steel plate. These complex geometric parts with different engineering materials are subject to large amount of compound strains and due to the nature of materials used in engine gasket, all compressive response will be nonlinear. This nonlinearity will add more challenge on the modelling and analysis of the whole continuum model. As a results, it's found that the effects of thermal expansion and engine pressure will cause some distortion in the assembly unit and will rise the tighten bolts force to maximum values. Also, the deformation or non-uniform in sealing cover will leads to down the engine performance. It's concluded that the stress distribution will be not uniform due to loads and differences in distortion rate on each individual part. Also, the clamping forces distribution need to be increase without exceeding the strength of each part.

Key words: Abaqus, thermal, deformation, nonlinear, engine and modelling, distribution

INTRODUCTION

In internal combustion engine, the more effecting factors on engine performance are the sealing unit. The prober design of gasket and suitable force calculation of the mounting bolts is important to avoid any leakage during operation process. The main function of gasket unit is to keep a good sealing in the combustion chamber during the maximum pressure and temperature. This sealing will works against coolants leakage of engine oil, air and thermal overload during combustion (Finuma, 2010).

Materials used in gasket should have a good thermal and chemical resistance for many types of coolants, oils and chemicals used in engine. The distribution of inner stresses has important effects on the engine components stability. Some of the engine components are very important like cylinder block and the stability calculations of these components during the operation are critical

(Mirajkar et al., 2013). Finite element simulation is widely used as an accurate tools to calculate precisely the main influencing factors on consumption, emissions and efficiency of the engine. Abaqus/CAE is a precisely analysis tool which can relied upon to estimate any deformation and dimensional change in powertrain components of the engine (Eagleson, 2013).

The required output from any simulation will determine the category and type of finite elements for use in thermal process analysis and including gasket type, solid elements and continuum shell.

Finite element analysis is important to predict and validate the integrity, durability and service life of the engine under different types of thermal and pressure loads (Hassan, 2017).

Numerical simulation by (CFD) always used and adopted from many researchers as essential tools in estimation and analysis the effectiveness of combustion engine under different conditions of pressures and

temperatures. Many of combustion characteristics and combustion engine problems can predict by simulation process like wall conductivity, operating conditions and heat losses (Zhang *et al.*, 2015).

Some researchers are numerically investigated the characteristics of the turbulent flow inside the combustion chamber and also, they determine the effects of equivalence air-fuel ratio and inlet velocity on sealing assembly performance (Shih and Liu, 2014; Wan *et al.*, 2014).

Now a days, simulation are widely used as a powerful tools in troubleshooting and design process evaluation. Cost and time can reduce to the minimum values by running the analysis many times and using the same actual boundary conditions. In case of the engine sealing, the associated tribological components can easily analysis for each individual part or as a whole assembly unit (Cheng, 2014; Malagi, 2012).

There are many thermal losses during operation time in internal combustion engine and this will cause to inefficiencies of the whole system. The majority of these loses are due to high dynamic response during operation and sometimes for non-suitable sealing unit (Baker, 2014; Malagi *et al.*, 2009).

MATERIALS AND METHODS

Parts geometry details and method description: The collective of completed engine seal joint in this research work are consisting from a paste gasket with silicone pellet are compressed together inside the groove between the manifold cover of coolant manifold and the intake of lower engine manifold. Sealing joint assembly is very important to keep the circulation and distribution of coolant inside engine cylinders without any leakage. All parts of this seal joint are considered as one unit in terms of calculation of sealing performance. Figure 1 and 2 illustrate this assembly in 2D and 3D.

The main materials used in these components are steel (for engine manifold intake and the bolts) and aluminium material for manifold cover with the paper foam and silicon gasket. Table 1 list these material properties.

Numerical simulation and boundary conditions: Simulation process by (Abaqus/CAE) includes systematic procedure which covers all analysis stages starting from
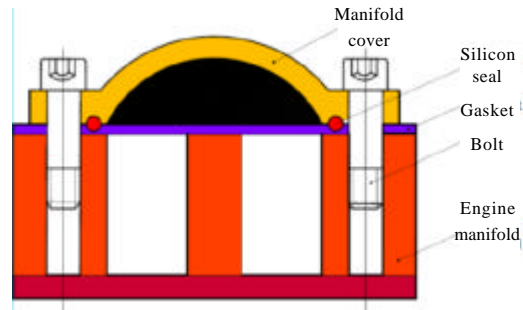

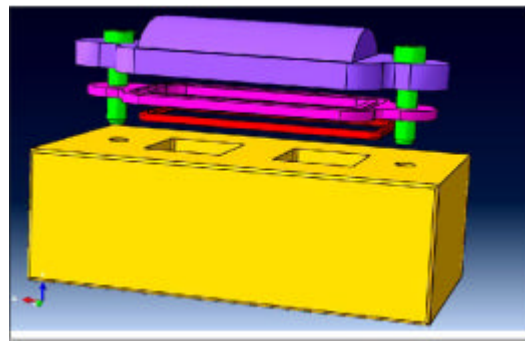Fig. 1: Gasket assembly unit in two dimensions

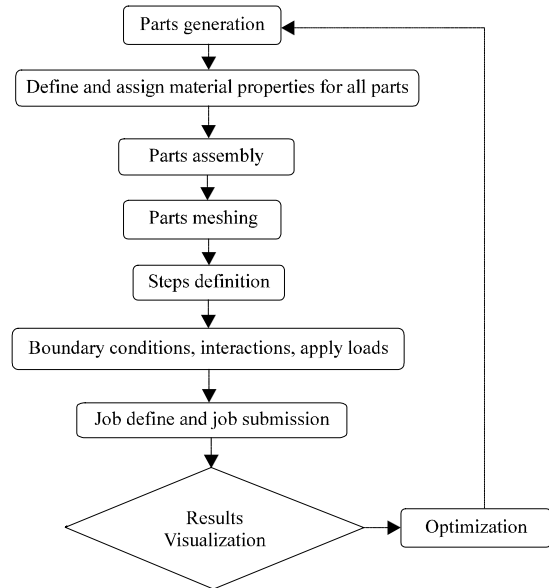Fig. 2: Gasket component in three dimensions

Fig. 3: Analysis procedure by Abaqus

parts definition till the job submission for the final analysis. Figure 3 illustrate the sequence of analysis procedure by Abaqus.

In the starting of simulation process, it's very important to define the interaction between all of
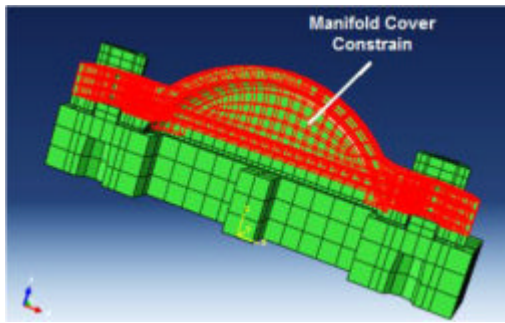
Table 1: Material properties used in gasket joint

Materials	Young's modulus (MPa)	Poisson's ratio	Thermal expansion (per°C)
Steel	2.0×10^5	0.28	1.6×10^{-5}
Aluminium	7.1×10^4	0.33	2.3×10^{-5}
Paper foam gasket			3.0×10^{-5}

Fig. 4: Constraints the nodes of manifold cover plane from any motion

Fig. 6: Constrain the base of engine manifold from any movement

Fig. 5: Constraints the gasket and bolts nodes along the symmetry axis
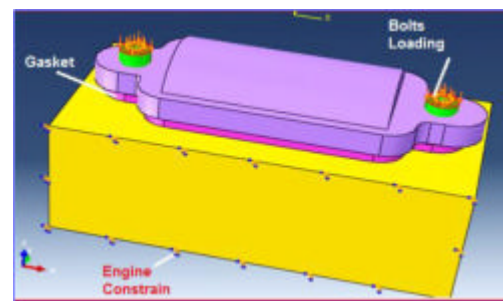
Fig. 7: Loading the bolts and constrain the engine base

components. In Abaqus, there are a lot of options to define the contacts and interactions between the mating parts and surfaces. The option (contact pair) is normally used in definition of paired surfaces which are mating together. Applying the condition of sliding contact between these deformable three dimensional parts lead to use (no separation) option between the mating surfaces of manifold and gasket.

The boundary conditions include constraints the entire nodes of gasket along the symmetry axis and planes. Even more and to secure the nodes along the plane of manifold cover from any motion in the normal direction (Z-direction), it's assumed that the manifold cover is bulky and stiff component. Figure 4 and 5 illustrate these two stages of boundary conditions.

To allow thermal expansion, the nodes in engine manifold are free in motion in horizontal direction (X direction) and the rigid body will eliminate from movement in the (Z-directions) as shown in Fig. 6.

There are many interactions between the contact parts, the bottom of the cover interacts with the top surface of gasket while the surface of top manifold will interact with gasket bottom. Also, the top surface cover flange has bearing interaction with bolt heads bottom. There are many sequence of analysis steps in this simulation including fasten and tight both the right and
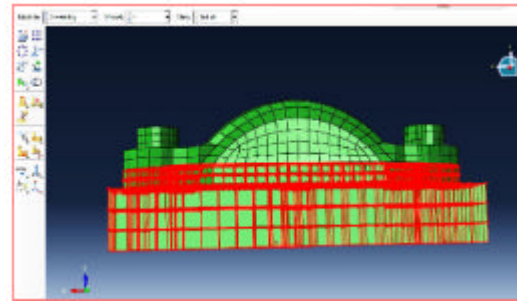
left bolt to load (6000 N) using the option (Pre-tension Section). Since, the simulation is symmetrical and only halve assembly part have been modelled, then the total load which carried by bolts will be (1200 N). Figure 7 explain this analysis step.

The following analysis step is a thermal cycle step which includes heating the entire joints uniformly to (165°C) which represent the maximum operation temperature of the engine. In same time the interior space will pressurize to (0.7 MPa) and replace the tension on the bolts by new boundary condition. In the final step, the interior pressure is maintained at same value (0.7 MPa), but the temperature of all the assembly unit will cool down to minimum temperature value (20°C). Also, it's including removing the interior pressure and returning gasket joint temperature to the ambient condition. Figure 8 show the creation of thermal boundary condition.

Elements mesh generation: Meshing the individual parts and assembly is very important in solution procedure. In Abaqus/cae there are many options can leads to the best results including mesh each part individually or merge many instances for meshing together. During parts meshing, the instances can be an orphan mesh or dependent mesh instances (original part is meshed
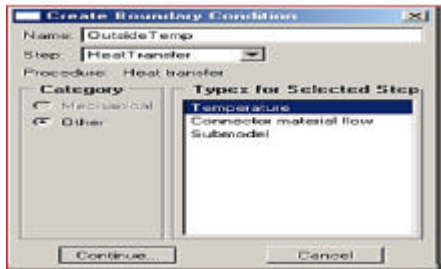
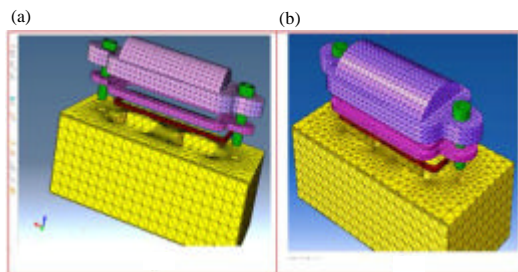Fig. 8: Generation heat transfer boundary condition

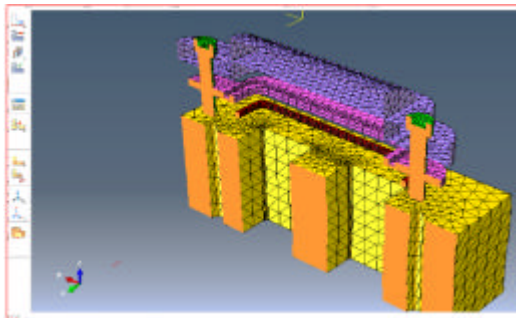Fig. 9: a, b) Assembly meshing from different angles

Fig. 10: Cross section in the mesh assembly

already) or independent mesh instances (part instance is meshed already in the main module) or a combination mesh from the three types above. Figure 9 and 10 show the assembly meshing from different angles and cross study in same assembly.

In abaqus/cae, if you have created meshed parts, then you can choose either you merge the parts geometry or merging their meshes. Its possible to create an orphan mesh and merge the original mesh with instance in one mesh. Mesh control is an important property in which the virtual topology of the parts and mesh accuracy can control. Also, the element type and element control for each part or components is very useful option in the common plot option to estimate the required values in each element exactly. Figure 11 and 12 explain the elements label for the whole model.
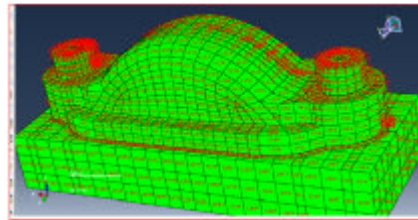

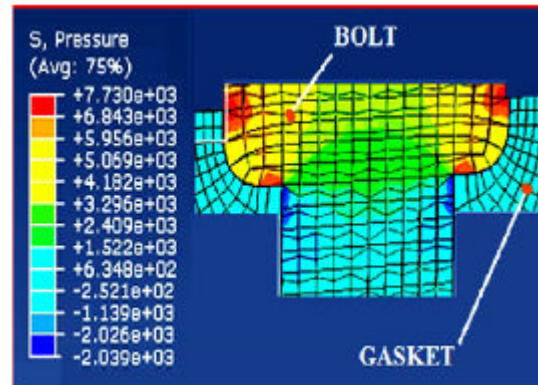Fig. 11: The elements label for the whole model

Fig. 12: Magnified contour of loading pressure between gasket and bolt
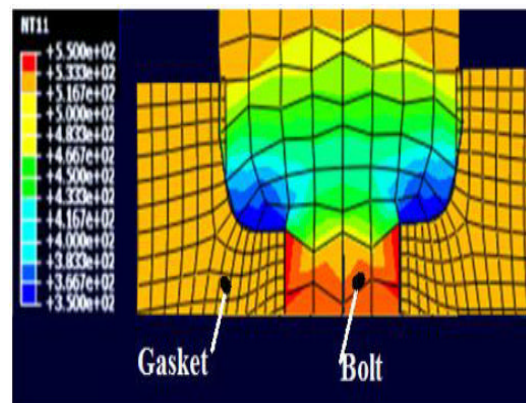

Fig. 13: Magnified contour of nodal temperature near combustion chamber

sealing surfaces due to the thermal load. To ensure and avoid the flanges bending, stress values of the metal engine components should be less than the yield values because any bending will give bad effects on the function of gasket sealing. Due to the effects of thermal expansion at high elevated temperatures, bolts load expected to be in a maximum values. The effects of engine pressure and thermal expansion in high temperatures will cause some distortion in the ssembly unit and will rise the tighten bolts force to maximum values and this will happened in third step exactly. Magnified contour of nodal temperature in Fig. 13 shows that temperature will be at maximum near
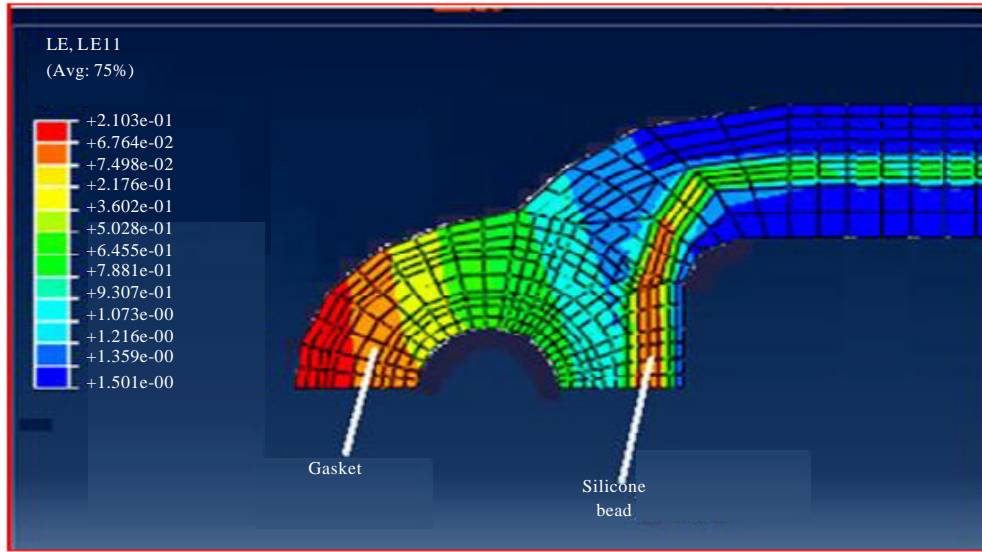
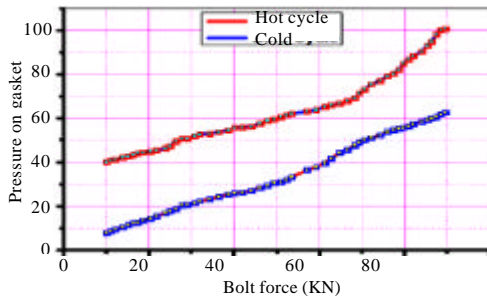Fig. 14: Contour of strain along the periphery of gasket

Fig. 15: Gasket pressure and bolt force in both of hot and cold operation

Fig. 16: Relation between thermal efficiency and operating pressure

combustion zone for both gasket and bolts. The pressure pattern of gasket sealing is highly affected by the flanges rigidity, so, it's very important to predict the values of applied load which the structure can resist before deformation. Any deformation or non-uniform conformity in sealing cover will leads to down the engine performance. Also, there is inelastic response will induced in gasket and inside silicon bead due to the engine pressure and thermal effects. Contour of logarithmic strains along the periphery of gasket are illustrated in Fig. 14.

Due to the frictional and thermal effects on the upper and lower surface of the gasket membrane, shearing or stretching will occur but the bolts are assumed to be fastening hardly with the base and there's no chance for surfaces to slip in any direction. Figure 15 illustrate that there's a proportional between gasket pressure and bolt force in both hot and cold operation cycle but in hot cycle will be at maximum value. Thermal efficiency of the whole
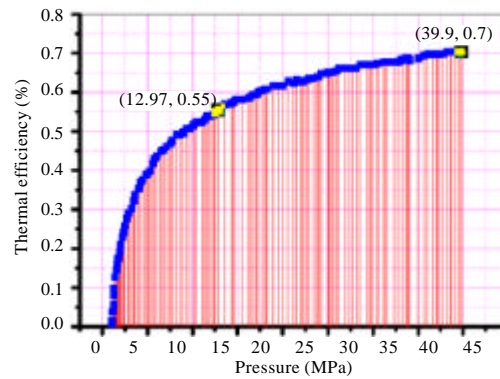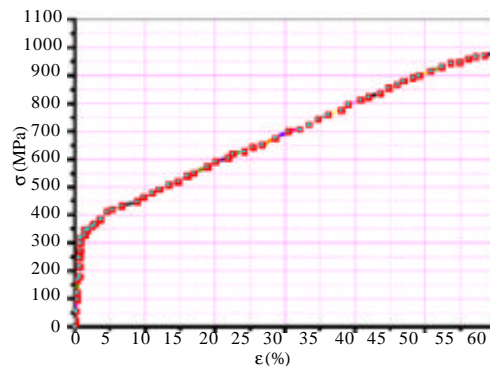

Fig. 17: Stress-strain relationship of gasket

join will increase gradually in the starting of thermal process and then increase proportionally with operating pressure as shown in Fig. 16 and 17.
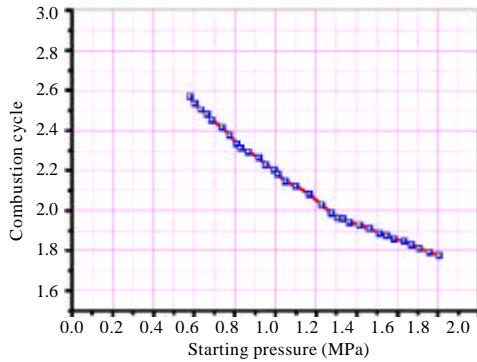
Fig. 18: Relation between starting operation pressure with combustion cycle
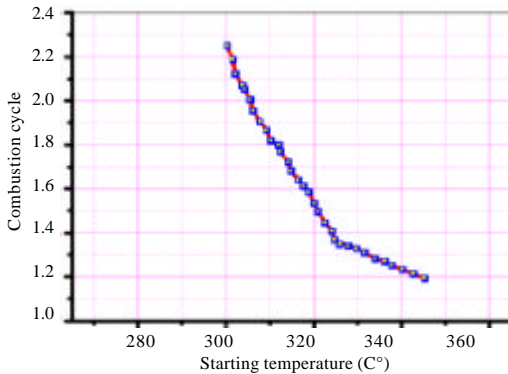

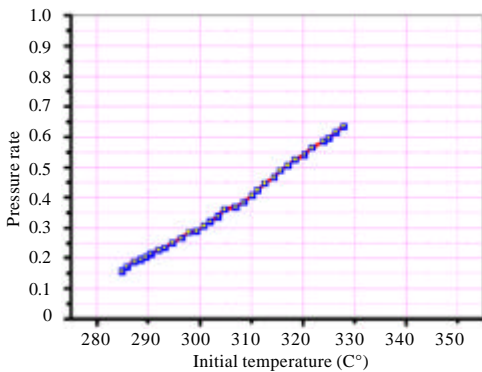Fig. 19: Relation between starting pressure with combustion cycle

Fig. 20: Proportional between temperature and pressure sealing surface

When the working temperature approaches to the maximum value and due to the effects of thermal expansion, the gasket stress and strain will reach the peak values. Figure 17 explain the gasket stress strain relationship. The starting operation pressure and temperature of gasket unit will record maximum values due to the high rotating torque as shown in Fig. 18 and 19.

This rapid increase will consider the main cause of gasket unit failure. In the starting of operation cycle there's a linear proportional between temperature and pressure along the gasket sealing surface. This can attributed to the high tension and pre-stressing of mating surfaces and all sealing components as shown in Fig. 20.

CONCLUSION

The following points are summed up as conclusions from this researcher numerical analysis by Abaqus can accomplish the variation values of gasket sealing in combustion engine under different boundary conditions. The effects of thermal strain and thermal stress on the gasket is consider the main weakness of the sealing joint and should be minimize by suitable load adjustment during assembly. Sealing unit capacity are highly depends on bolts pre-stressing because bolts loading is the main external load which withstand the thermal and pressure of the whole system. To avoid the thermal stresses and to improve the sealing performance, it's important to increase the whole clamping force of the assembly but without overtakes the strength of each part in the assembly. Mesh refinement is important for capturing the accurate results of temperature and pressure in any region. Distribution of resultant stresses can highly affected by the sequence followed the bolts tightened because any over tightening or poor tightening will leads to extensive distortion of flanges gasket and cause weak sealing performance.

REFERENCES

Baker, C.E., 2014. The effect of transient dynamics of the internal combustion compression ring upon its tribological performance. Ph.D Thesis, Loughborough University, Loughborough, England.

Cheng, C., 2014. Numerical modeling of the power cylinder system for internal combustion engine with an emphasis on ring pack design. Ph.D Thesis, Michigan State University, East Lansing, Michigan.

Eagleson, J.S., 2013. Head gasket finite element model correlation. Master's Thesis, Automotive and Materials Engineering, University of Windsor, Windsor, Ontario, Canada.

Finuma, G.D., 2010. Analysis of cylinder head gasket sealing under engine operation conditions. MSc Thesis, The Ontario Association of Certified Engineering Technicians and Technologists, Toronto, Canada.

Hassan, G., 2017. Design of research oriented cylinder head for a heavyduty engine. Master's Thesis, Aalto University, Finland.

Malagi, R.R., 2012. Estimation of blowby in multi-cylinder diesel engine using finite element approach. Master's Thesis, SAE Institute Adelaide, Adelaide, Australia.

Malagi, R.R., S.N. Kurbet and N. Gowrishenkar, 2009. Finite element study on piston assembly dynamics emphasis with lubrication (No. 2009-28-0064). Master's Thesis, Automotive Research Association of India, Pune, India.

Mirajkar, P.B., K.A. Kanase and N.K. Chhapkhane, 2013. Cylinder liner bore distortion estimation during assembly of diesel engine with FEA. Intl. J. Mech. Eng. Rob., 1: 2321-5747.

Shih, H.Y. and C.R. Liu, 2014. A computational study on the combustion of hydrogen/methane blended fuels for a micro gas turbines. Intl. J. Hydrogen Energy, 39: 15103-15115.

Wan, J., W. Yang, A. Fan, Y. Liu and H. Yao et al., 2014. A numerical investigation on combustion characteristics of H2/air mixture in a micro-combustor with wall cavities. Intl. J. Hydrogen Energy, 39: 8138-8146.

Zhang, Y., Z. Zuo and J. Liu, 2015. Numerical analysis on combustion characteristic of leaf spring rotary engine. Energies, 8: 8086-8109.