Engine Combustion simulation using openFoam

#1K.S.Kolambe, #2S.L.Borse

#1P.G. Student, Department of Mechaniacal Engineering,S.P.University,Pune,R.S.C.O.E Pune,Maharashtra,India-411033
#2Department of Mechaniacal Engineering, S.P.University,Pune,R.S.C.O.E Pune,Maharashtra,India-411033

ABSTRACT— CFD models have been extensively validated and used to predict the performance and emissions of spark ignition (SI) and compression ignition (CI) engines.Opposite to the experimental methods, numerical methods are often less expensive and faster.The present work is the numerical investigation of Spark Ignition (SI) engines using an open source Computational Fluid Dynamics (CFD) tool.Investigations on the usage of OpenFOAM, CFD tool has been carried out for the simulation of SI engines using engineFoamsolver.The standardk—ε turbulence model will be used along with the Reynolds Averaged Navier Stokes (RANS) equationsfor simulating the flow field in the further work. Cylinder and cylinder head geometries needed for combustion chamber were created in OpenFOAM itself by using blockMesh. Meshing was done using snappyHexMesh. Comparison of average cylinder pressurefor different Crank Angles (CA) from −180 to 60 is to be done with the existing kivaTest.The temperature contours can also plottedon a vertical plane inside the cylinder to indicate the rise in temperature due to combustion.

Keywords— Spark ignition(SI),OpenFOAM, kivaTest, Combustion, blockMesh, snappyHexMesh

I. INTRODUCTION

The Spark Ignition (SI) engines are mostly used in motorbikes, cars and also in small trainer aircrafts. There are various requirements for the engines based on the regulatory authorities for automobiles and aircrafts. CFD models have been extensively validated and used to predict the performance and emissions of spark ignition (SI) and compression ignition (CI) engines. Opposite to the experimental methods, numerical methods are often less expensive and faster. . Beside these advantages, simulations give much further information (mixture formation, combustion process, flow field, etc.) CFD codes such as STAR-CD, ANSYS Fluent, *KIVA* etc. are able to solve this kind of problem with their numerical contents and models. Recently, the fluid dynamic analysis and optimization of SI type of Internal Combustion (IC) engines are being carried out using Computational Fluid Dynamics (CFD) tools. OpenFOAM®, an open source CFD tool has gained researchers attention for CFD analysis in various fields. The above said CFD tool has not been used like other commercial tools that are available in the market. Various developments are being carried out for engine simulation related solvers especially for solving diesel engine processes. Numerical simulations of SI Ignition (CI) engines due to lesser physical processes. The simulation of SI engines using has not been explored much using *engineFoam* solver of the OpenFOAM. The spark ignition engine combustion processes is an extremely complicated combustion of phenomena.

This paper mainly concentrate on, the simulation of combustion chamber geometry. Interest was to create different geometry by creating cylinder (as piston and liner) as blockMesh in OpenFOAM and cylinder head as a stereolithogrphy (STL) file in CATIA software. The evaluation of in-cylinder temperature and pressure for all crank angle(-180 to 60) can be done using OpenFOAM.

II. LITERATURE SURVEY

Kannan, B.T. compared for Cold flow compression and combustion simulations in terms of temperature and pressure for various CA. The temperature contours were plotted on a vertical plane inside the cylinder indicates the rise in temperature due to combustion.

A computer simulation was performed by Ender HEPKAYA, Salih KARAASLAN to visualize fluid flow and combustion characteristics of a single cylinder spark ignition engine. The complete engine cycle process (inlet, compression, expansion and exhaust strokes) in gasoline engine model was investigated using RANS (Reynolds Averaged Navier-Stokes) and CFM (Coherent Flame Model) approaches offered by Star-CD/es-ice. Courant and Mach numbers were investigated in details at the exhaust stroke. Also global engine parameters such as in cylinder pressure, temperature and heat release rates were plotted.

The peak pressure and temperatures were higher than experimental values and the combustion process was observed very fast, it has ended in 25-30 CA.

Cho *et al.* (2010) have studied on a four-valve, pent roof, direct-injection spark-ignition (DISI) engine, with the fuel injector located between the two intake valves. They measured and simulated wall film behavior of formation, transport, and vaporization on the surface.

Fontanesi (2009) presented the results of numerical simulation of contemporary gasoline direct injection engine using STAR-CD/es-ice software. They investigated the spray behavior on the walls of combustion chamber. They used the experimentally validated model for spray conditions. So, they concluded choosing the proper models, the injection process and effects of the strategy and location of injection on the wall/fuel interactions could be well demonstrated.

Koten *et al.* (2010) studied injection effects on the HCCI engine combustion to get nearly full combustion. In other words, nearly zero emissions. They used a model of one cylinder of a six-cylinder diesel engine with nine liters displacement. The comparative results of engine performance depending on compression ratio, injection timing, cone angle and bowl geometry are in good agreement with the results of similar numerical and experimental works.

D'errico (2008) developed a multi-zone combustion model for the prediction of the performance and exhaust emissions of an engine.

Yucel (2010) modeled the combustion in SI engine cylinder. The effects of air/fuel ratio on the combustion were investigated numerically. They offered ignition timing advance with respect engine speed. They used k- ε turbulence modeling and Arrhenius and EBU models in their study.

Lecocq *et al.* (2011) modeled abnormal combustion processes in reciprocating engines. They showed that retarding the spark timing delaying the occurrence of knock. In addition to this result, their model had the ability of determining the flame front due to a hot point before spark ignition, which is a factor for knocking.

Methodology

The computational tool used in this paper is the open-source computational fluid dynamics (CFD) software OpenFOAM released by ESI-Open CFD, which has been extended to perform large eddy simulation (LES) with topologically changing grids. The OpenFOAM (Open Field Operation and Manipulation) CFD toolbox is used to analyse the in-cylinder temperature and pressure for the combustion chamber geometries.

OpenFOAM has extensive range of features to solve anything complex fluid flow involving chemical reactions, turbulence, and heat transfer to solid dynamics and electromagnetics. The cylinder geometry having different external parameters is created in OpenFOAM itself by using blockMesh. Cylinder head geometry done in CATIA software and then converted into the stereolithogrphy (STL) file. STL file is then linked to snappyHexMesh for generation of mesh. Further both (cylinder and cylinder head)geometries are merged in OpenFOAM. KivaTest in engineFoam can be used to generate combustion flow.

By giving initial boundary conditions calculations can be done by using engineFoam and finally post processing will be done by using ParaFoam. The in-cylinder temperature and pressure distribution over the geometries are to be visualized and discussed. Then accordingly after studying it, the in-cylinder temperature and pressure distribution will be calculated.

Computational set up

Different cases were considered in order to achieve the given goals. However, the simulation setup is the same for all the cases and hence the procedure is same for all cases. The first step required to begin a new simulation is to input the actual mesh into the file structure of OpenFOAM .But in our project we are creating the geometry of cylinder in OpenFOAM itself by using blockMesh and cylinder head in CATIA software and converting the geometry in stereolithogrphy (STL) file.

Mesh

Mesh is created using snappyhexmesh, OpenFOAM's native meshing tool. Utility snappyHexMesh is used to create high quality hex dominant meshes based on arbitrary geometry. It is controlled by parameters in the snappyHexMeshDict. It can be executed in parallel. It preserves the feature edges, addition of wall layers.

Here we started with creating the blockMesh itself in OpenFOAM for the base cylinder (i.e. piston and liner) and then after we created the Mesh of cylinder head using snappyHexMesh by importing a STL file of cylinder Head created in CATIA software. The details of the geometry are as shown in the following Figures.

Step 1: blockMesh of cylinder

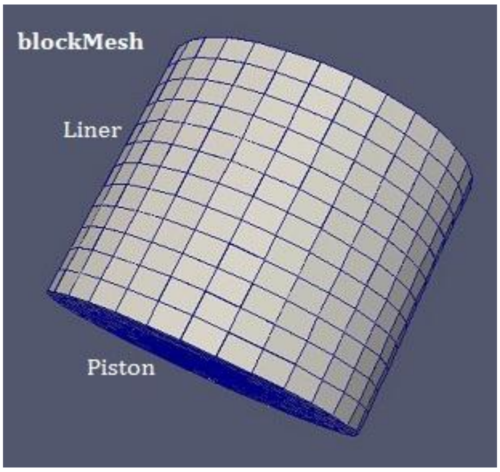
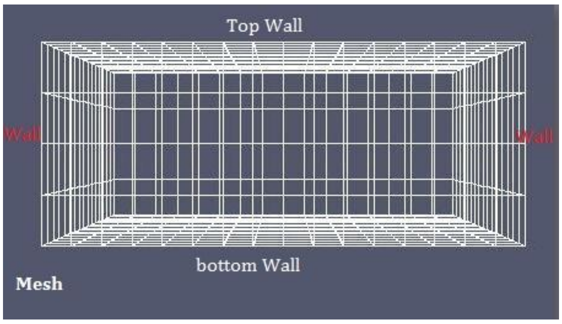


Fig. 1blockMesh of cylinder (piston & liner)

Step 2: snappyHexMesh of cylinder head



Step 2: snappyHexMesh of cylinder head

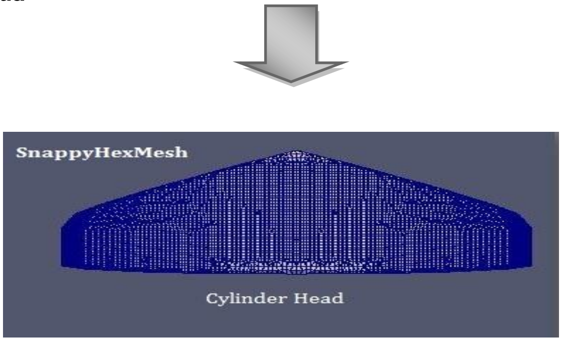


Fig. 2snappyHexMesh of cylinder head

Step 3: Merging of Meshes

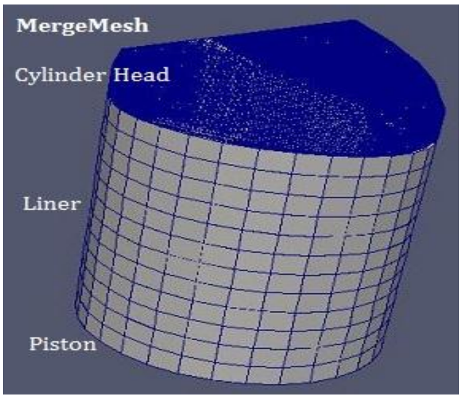


Fig. 3Merged Meshes of cylinder and cylinder head

4.2 Engine details

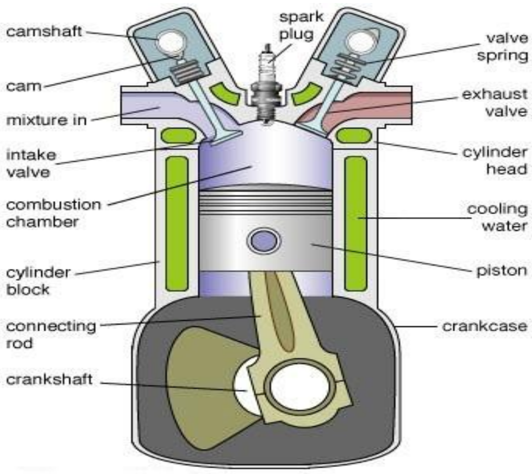


Fig 4 Engine with its different components

The engine selected for the present work is a pent-roof type with four valves closed same as of kiva Test. The various geometric details of the engine are given in Table 1. These geometric details will be used by the solver.

Table 1 Engine details for the simulation.

S. No	Items	Values/Details
1	Engine	K3PREP/100198 |Pent-roof |4-valves
2	Connecting rod length	0.147m
3	Bore	0.092m
4	Stroke	0.08423m
5	Clearance	0.00115m

The engine RPM is 1500. The details for fuel and combustion are provided in Table 2.

Table 2 Fuel and other details for combustion.

S. No	Items	Values/Details
1	Fuel	Iso-Octane
2	Laminar flame speed	0.42 m/s
3	Stoichiometric ratio	15.0336
4	Thermo-physical model	*heheuPsiThermo*(referOpenFOAM manual)

Computational domain

The computational domain for simulating SI engine is chosen based on the engine geometry and it is shown in Fig. 4The geometry consists of cylinder, piston, liner and cylinder head. The boundary conditions are applied according to the physical conditions. The generated grid is fed to the solver for solving the governing equations (refer *kivaTest*example).

Boundary conditions

- Piston
- Liner
- Cylinder Head

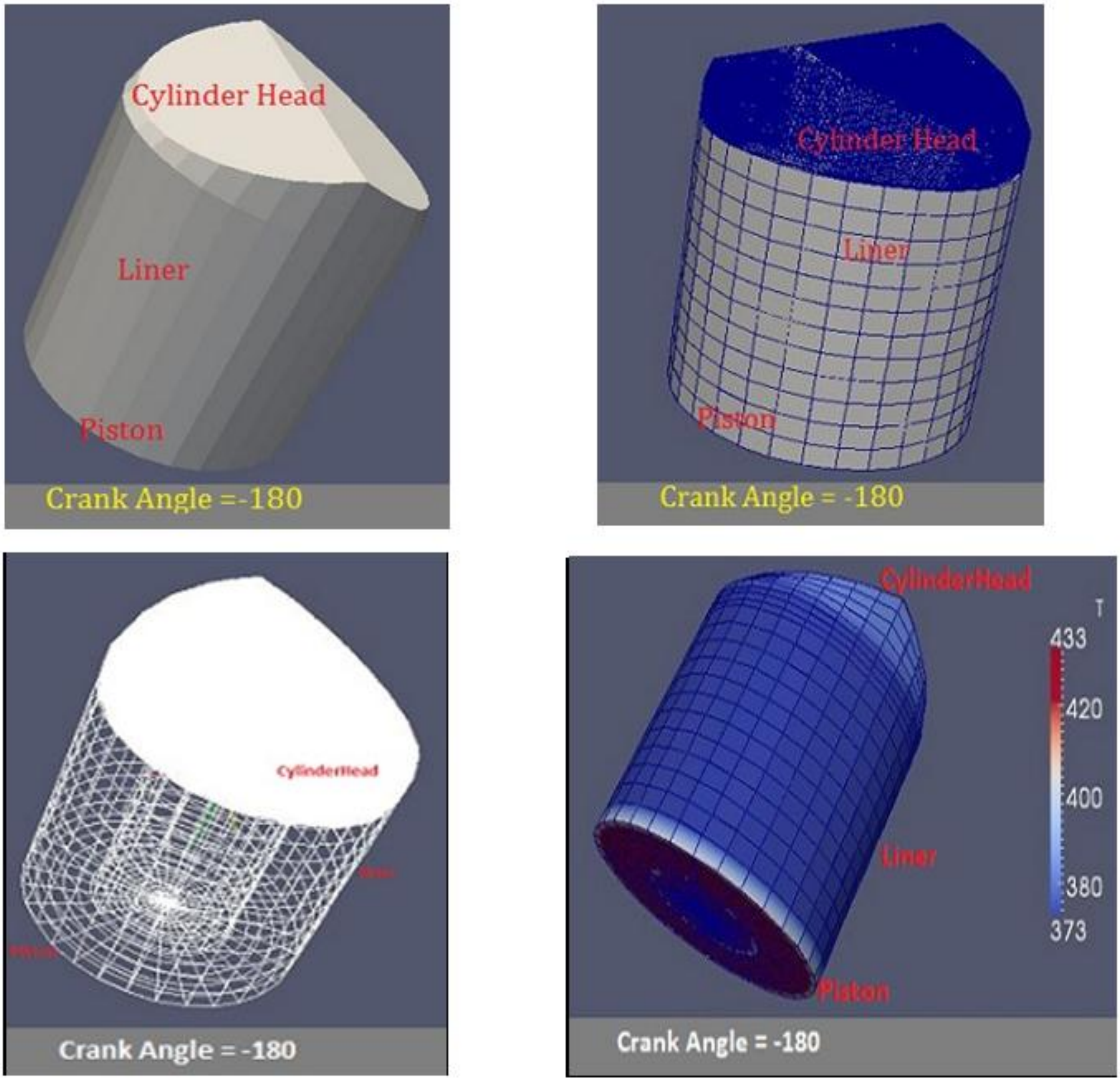


Fig.5Engine geometry, computational domain with mesh and boundary conditions.

- Governing equations and other details

The simulations for SI type of IC engines require several governing transport equations and models for combustion. Theequations and models to be used in the present work are describedas below.

- RANS equations along with standard k—ε turbulence modelfor simulating the flow field
- Energy equation and transport equation for regress variable
- Transport model for Xi
- Gulder's correlation is used for laminar flame speed alongwith unstrained model.

Numerical procedure

- The solver application namely *engineFoam* is invoked for solving the governing equations. The simulation will be carried out in two steps. The two steps are from CA −180 to −15and −15 to 60 with different time step values for avoiding the Courant number related errors. The simulated results are post-processed with the help of Paraview.

Results and discussions

- To know the combustion behaviour inside the cylinder of SI engines, in the present work results can be compared with the simulations of *kivaTest* combustion cases. The predicted results in this project work are to be drawn by taking the reference of *kivaTest*discussed in the following sections. In our project work the expected temperature contours for various Crank Angles are to be compared with the existing *kivaTest* as shown below

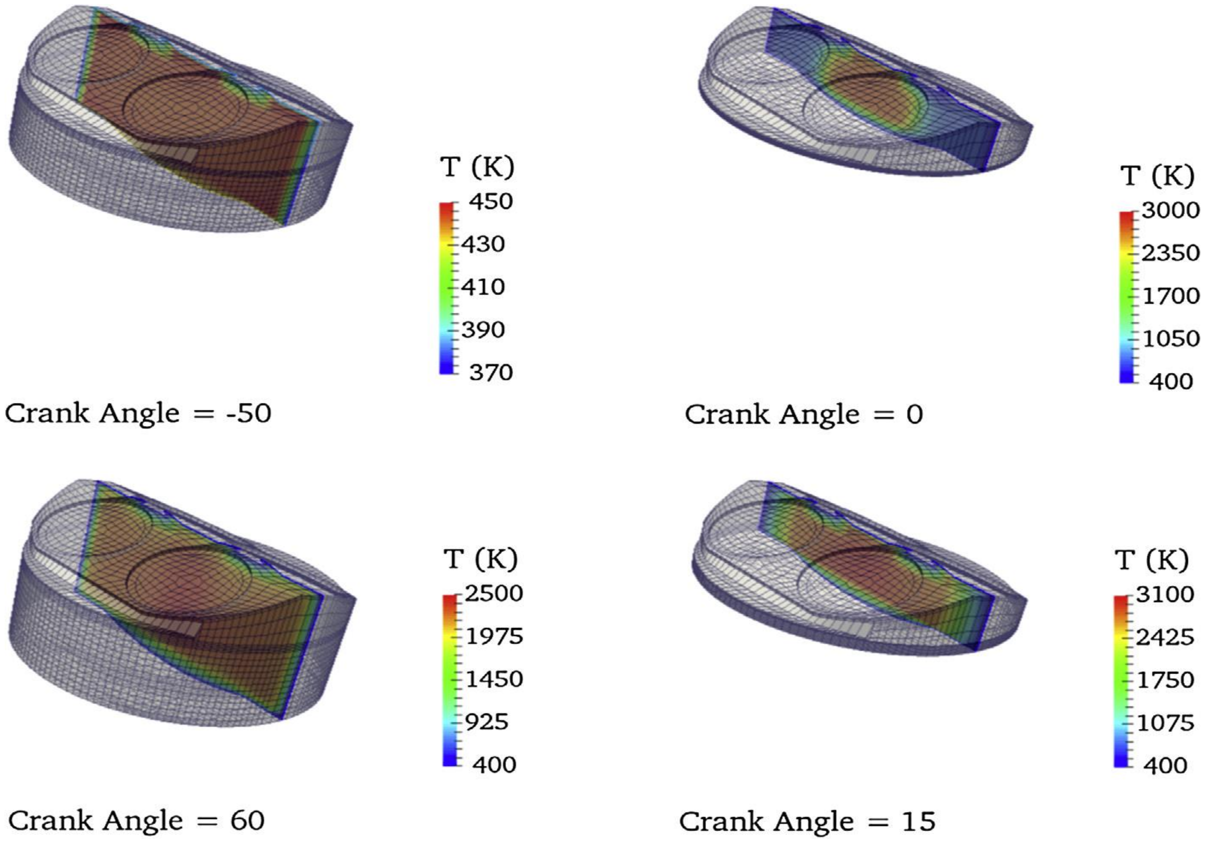


Fig. 6 Temperature contours for various CA (see clockwise from top left).

- *Average temperature and pressure versus Crank Angles*

The *engineFoam*solver has averaging routines and it provides output for processing the engine simulations.Theaverage in-cylinder temperature and pressure is recordedfor all CA considered in the present study. The ignition starts from CA −15and combustion process is initiated which results in the risein both temperature and pressure.

- *Temperature contours for various Crank Angles*

The variation of in-cylinder temperature is shown as contourplots in Fig. 5The top two figures show temperature rise dueto compression and due to ignition source. The bottom twofigures show the variation of temperature due to combustionand expansion. The maximum temperature attained showsthe proper simulation of combustion for Iso-octane fuel.

III. CONCLUSIONS

1. The numerical results based on our geometry reported in this paper can provide flow field data for Spark Ignition (SI) engines using an open source CFD tool OpenFOAM. In this study, a computer simulation was performed, to visualize fluid flow and combustion characteristics of a spark ignition engine.
2. Simulations are to be done for compression ratio and rpm engine speed. C8H18 (isooctane) was used as the engine fuel. In the numerical simulations to model turbulent flow field, k -ε turbulence model was selected.
3. In results, the pressure, temperature and fuel concentration after spark ignition will be investigated and interpreted in details.
4. By using our own geometry of combustion chamber using blockMesh and STL, Cold flow compression and combustion simulations will be compared in terms of temperature and pressure for various CA.
5. The simulations that we are going to carry out in the present work provide physically possible solutions. It is highly recommended to investigate openFOAM modifications and its effect on results for different cases.
6. The overall results indicate that the open source CFD code OpenFOAM can be an ideal choice for engine designers.

IV. ACKNOWLEDGMENT

The authors would like to present their sincere gratitude towards the Faculty of Mechanical Engineering in RajarshiShahu College of Engineering, Pune.

REFERENCES

1. GanesanV.(2007) *Internal Combustion Engines*. 3rd ed. Tata McGraw Hill.

2. Heywood JB (1988) *Internal Combustion Engine fundamentals*.1st ed. McGraw Hill.

3. Ganesan V. (1996) *Computer Simulation of Spark-Ignited Engine Process*. Universities press (India) Limited.

4. Kannan, B.T.(2015) Numerical simulation of spark ignition engine using OpenFOAM.Proc. Eng. 127, 1292—1299.

5. Ender HEPKAYA, Salih KARAASLAN(2010), 'A case study of combustion modeling in a spark ignition engine using coherent flame model',Department of Mechanical Engineering, Faculty of Engineering, Gazi University.

6. Cho K., Grover R. O., Assanis D., Filipi Z., Szekely G.,Najt P., Rask R., 2010, Combining Instantaneous Temperature Measurements and CFD for Analysis of Fuel Impingement on the DISI Engine Piston Top, *J. Eng. Gas.*

7. Malaguti S., Fontanesi S., 2009, CFD Investigation of Fuel Film Formation within a GDI Engine under Cold Start Cranking Operation, *Proceedings of the ASME Internal Combustion Engine Division 2009 Spring Technical Conference (ICES2009)*, Milwaukee, Wisconsin, USA, 555-562.

8. Koten H., 2010, Comparison of Various Combustion Models within a Multidimensional Framework Applied To Heavy Duty CI Engine, *Proceedings of ICFD 10: Tenth International Congress of Fluid Dynamics*, Egypt.

9. D'Errico G., 2008, Prediction of the Combustion Process and Emission Formation of a Bi-fuel SI Engine, *Energy Conversion and Management*, 49, 11, 3116–3128.

10. Dinler N., Yucel N., 2010, Combustion Simulation in a Spark Ignition Engine Cylinder: Effects of Air-Fuel Ratio on the Combustion Duration, *Thermal Science*, 14, 4, 1001-1012.

11. Lecocq G., Richard S., Michel J.B., Vervisch L., 2011, A new LES model coupling flame surface density and tabulated kinetics approaches to investigate knock and preignition in piston engines, *Proceedings of the Combustion Institute*, 33, 2, 3105–3114.

12. Lucchini T, Cornolti L, Montenegro G, D'Errico G, et al. A comprehensive model to predict the initial stage of combustionin SI engines.*SAE Technical Paper*, 2013-01-1087; 2013.

13. Lorenzo Bartolucci (December 9, 2014), EngineFoam: implementation of adifferent combustion model and thenew Janaf thermo equations.

14. OpenFOAM.,(July 2009.) "OpenFOAM, The Open Source CFD Toolbox, User Guide", Version 1.6.

15. OpenFOAM project web site. http://www.OpenFOAM.org, 2007.

