ISSN : 2248-9622, Vol. 4, Issue 7(Version 1), July 2014, pp.74-79

RESEARCH ARTICLE OPEN ACCESS

Cfd Studies of Two Stroke Petrol Engine Scavenging

S Gavudhama Karunanidhi, NithinV S, G Subba Rao
Assoc. Professor, Department of Mechanical Engg MES College of Engg Kuttipuram, Kerala
PG Scholar, MES College of Engineering, Kuttippuram,Kerala, Calicut University
Principal, Geethanjali Institute of Science and Technology, Nellore.

ABSTRACT
This project deals with the numerical analysis of 2 stroke engine scavenging in two cases. One with an existing condition (Flat headed pistons) and another with a new design (Dome headed piston) .The numerical analysis is done with help of CFD software ANSYS FLUENT 14.5. Here, the modeling of engine piston with flat headed type and with dome headed types was done in workbench. In ANSYS FLUENT after the geometrical design, for the dynamic motion meshing is used and set up species transport model also. At first the scavenging effect of flat headed piston is analyzed. Later the simulation of piston with dome headed type was also checked. Analyzing the variations from each and selected the best method for scavenging. Finally the scavenging efficiency is calculated for both type arrangements.
Keywords – CFD, Air-Mass fraction, CO2-Mass fraction, Scavenging, k-ε model, FVM, FEM, FDM

I. INTRODUCTION

Internal combustion engines have undergone significant advancement since 16th century. The influence of fuel powered engines in the modern world is very high so that we cannot avoid them for emission reduction reason. The latest trend in IC engine industry is to develop power plants with higher efficiency and to make them running on alternative fuels to meet modern emission norms. Reductions in fuel consumption can be achieved by a variety of measures, including improved aerodynamics, using different blends of fuel and hybrid power trains. The world in the 21st century presents many critical challenges. One of the most important challenges is the efficiency it directly depends on the fuel consumption and engine life. But in nowadays the two stroke engines are outdated due to their inefficiency. To improve engine performance of 2 stroke engines is important. To design a new engine we need to know the engine performance in earlier cases and then finding the remedy. The increased computing power has led to advancement in the area of CFD. Here scavenging is the parameter selected for study.

II. SIGNIFICANCE

It is widely known that the scavenging process plays a very important role in the performance and efficiency of two-stroke engines. Briefly, scavenging is the process by which the fresh charge displaces the burnt gas from the cylinder. Due to the difficulties associated with the measurement techniques, CFD (Computational Fluid Dynamics) is a very helpful tool to analyze the flow pattern inside the cylinder. CFD simulations can provide more detailed information than experimental studies. Successful simulation of processes in a two-cylinder two-stroke engine involves the interaction of various physical models operating in a three-dimensional (3D) geometry and additionally with moving boundaries.[8] The complex physical phenomena inside the engine such as turbulence, phase change, chemical reaction, etc. increase the time of calculation and thus make it impracticable for the development process of a new engine. The time of calculation can be reduced considerably with a smart simulation strategy and therewith the simulation produces successful results within a shorter computational time.

2.1 SCOPE OF THE PROJECT

CFD tool gives a clear picture of scavenging by giving various contours. We know that scavenging is the process of expelling of exhaust gas from the engine cylinder by making use of the swirling action of the charge at inside. In actual it is clear that scavenging is taking place after the combustion process, and one more thing is after the combustion the temperature inside the cylinder will be very high. It is very complicated to measure that temperature and pressure, and there by the scavenging process also. In earlier cases scavenging is analyzed only with the help of ideal Otto cycle . To some extend the pressure and temp is measured by using piezoresistive sensors, and is very costly. By the use of CFD tool the need of such type of sensors can be avoided to greater extent. This helps in saving of money, material as well as time. The use of CFD tool in proper way we can create correct combustion and there by correct values in all analysis also. And is more easier to get the results than experimental way.

2.2 MECHANICAL ASPECTS

A two-stroke engine is an internal combustion engine that completes the process cycle in one revolution of the crankshaft or two strokes of the piston: an up stroke and a down stroke. Both spark ignition and compression ignition engines exist today. Spark ignition engines are employed in light applications (chainsaws, motorcycles, outboard motors, etc) due to its low cost and simplicity. On the other hand, diesel compression ignition engines are mainly employed in large and weight applications, such as large industrial and marine engines, heavy machinery, locomotives, etc

A two-stroke, two-cycle, or two-cycle engine is a type of internal combustion engine which completes a power cycle in only one crankshaft revolution and with two strokes, or up and down movements, of the piston in comparison to a "four-stroke engine", which uses four strokes to do so. This is accomplished by the end of the combustion stroke and the beginning of the compression stroke happening simultaneously and performing the intake and exhaust (or scavenging) functions at the same time.

III. DISCRETIZATION METHODS

The space, where the flow is to be computed - the physical space, is divided into a large number of geometrical elements called grid cells. This process is termed grid generation (some authors use the term mesh with identical meaning). It can also be viewed as placing first grid points (also called nodes or vertices) in the physical space and then connecting them by straight lines - grid lines. The stability of the chosen discretization is generally established numerically rather than that of analytically as with simple linear problems. Special care must also be taken to ensure that the discretization handles discontinuous solutions gracefully. The Euler equations and Navier–Stokes equations both admit shocks, and contact surfaces. Some of the discretization methods being used are:

3.3.1 Finite volume method

The finite volume method (FVM) is a common approach used in CFD codes, as it has an advantage in memory usage and solution speed, especially for large problems, high Reynolds number turbulent flows, and source term dominated flows (like combustion).In the finite volume method, the governing equations partial differential equations (typically the Navier-Stokes equations, the mass and energy conservation equations, and the turbulence equations) are recast in a conservative form, and then solved over discrete control volumes[9]. This discretization guarantees the conservation of fluxes through a particular control volume. The finite volume equation yields governing equations in the form,

$$\frac{\partial}{\partial t}\iiint Q\, dV + \iint F\, d\, A \qquad (1)$$

where Q is the vector of conserved variables, F is the vector of fluxes (see Euler equations or Navier–Stokes equations), V is the volume of the control volume element, and A is the surface area of the control volume element.

3.3.2 Finite element method

The finite element method (FEM) is used in structural analysis of solids, but is also applicable to fluids. However, the FEM formulation requires special care to ensure a conservative solution. The FEM formulation has been adapted for use with fluid dynamics governing equations. Although FEM must be carefully formulated to be conservative, it is much more stable than the finite volume approach. However, FEM can require more memory and has slower solution times than the FVM.

In this method, a weighted residual equation is formed

$$R_i = \iiint W_i\, Q\, dV^e \qquad (2)$$

where R_i is the equation residual at an element vertex i, Q is the conservation equation expressed on an element basis, W_i is the weight factor, and V^e is the volume of the element.

3.3.3 Finite difference method

The finite difference method (FDM) has historical importance and is simple program. It is currently only used in few specialized codes, which handle complex geometry with high accuracy and efficiency by using embedded boundaries or overlapping grids (with the solution interpolated across each grid).

$$\frac{\partial Q}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} + \frac{\partial G}{\partial z} = 0 \qquad (3)$$

where Q is the vector of conserved variables, and F,G and H are the fluxes in the x,y, Once the basic performance of two-stroke engines was described, the methodology to simulate the scavenging process will be treated in this section.

3.3.4 Governing equations

The governing equations of the flow inside the cylinder are the Navier-Stokes ones. The energy equation is also needed to compute the thermal problem. Finally, as there are two components (air and burnt gases), one more equation must be added to characterize the propagating interface. These equations are briefly described in what follows.

In Cartesian tensor form, the continuity equation is given by

$$\frac{\partial \rho}{\partial t}+\frac{\partial (\rho u_i)}{\partial x_i}=0 \qquad (4)$$

where ρ is the density and u the velocity. It is very common to consider the flows as ideal gasses, so the density can be calculated as follows: .[1]

The momentum conservation equation is given by:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_{i,}u_j)}{\partial x} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} \quad (5)$$

where τij is the stress tensor. If the fluid is treated as Newtonian, the stress tensor components are given by:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_i} + \frac{\partial u_i}{\partial x_i} - \frac{2}{3}\delta_{ij} \quad \frac{\partial \mu_k}{\partial x_k}\right) \quad (6)$$

As only the scavenging process and not the combustion is treated on this chapter, only two components need to be computed: burnt gas and unburnt gas (air). In order to characterize
the propagating interface, the following equation is solved:

$$\frac{\partial(\rho y_{air})}{\partial t} + \nabla . y_{air} = \rho v \quad (7)$$

where *Yair* is the mass fraction of the air. The mass fraction of the burnt gases, *Ygas* , is given by the restriction that the total mass fraction must sum to unity:

Ygas= 1- Yair (8)

and z directions respectively.

IV. ENGINE MODELING

The engine selected for project is 2 stroke JAVA 200 CC. The engine designing is having 3 ports, they are scavenging port, exhaust port, inlet port. Its design is a bit complicated because of the port positions and its design. The designed engine specification is given below.

Parameter	Value
Engine	2 stroke JAVA engine
Displacement	200.3 cc
Bore x Stroke	60 x 64 mm
Compression rate	9.3:1
Max Power	11.3 ps @ 6000 rpm
Max Torque	1.03 Kgf @4000 rpm
Pressure from BDC_ TDC	1.3 bar -13 bar

As per the given engine specification the geometry is created. Now for this project it need it model two type geometries. They are,

1 Flat headed piston

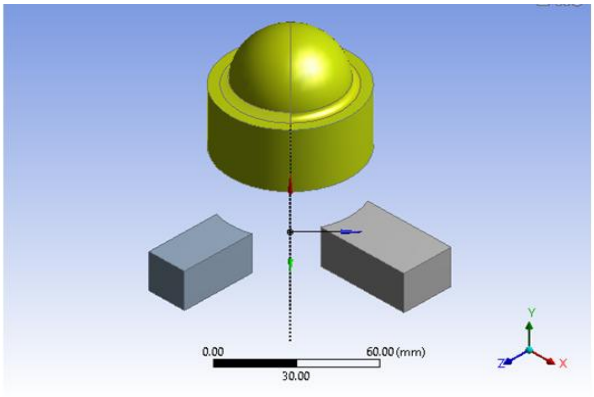

Fig 1 Flat piston model

2 Dome headed piston

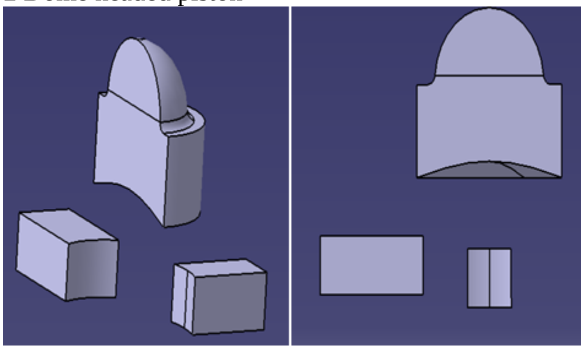

Fig 2 Dome headed piston

To ensure a high-quality product, diagrams and lettering MUST be either computer-drafted or drawn using India ink.

4.1 Meshing

In CFD analysis meshing is an important stage. Here all the parts of the design are subjected to mesh in accordance with our meshing style, size etc. And there are certain methods is there to follow during the meshing process.The engine geometry from design modular is now imported into meshing in ANSYS FLUENT. In this phase we are doing only the general meshing not the dynamic one. And the meshing is completed with hexahedral meshing which is suitable for dynamic meshing during the analysis. For the finite volume analysis the element size selected is
0.0025mm.After the meshing get completed, the different parts are named using face selection method. The different parts are, inlet port, exhaust port, scavenging port, moving cylinder wall, fixed cylinder head. After completing this meshed geometry is updated.

4.2 Engine incylinder details

Table.5.6.Incylinder parameter

parameter	Values
Crank shaft speed(rpm)	750
Starting crank angle(deg)	75
Crank period(deg)	360
Crank angle step size(deg)	0.25
Crank radius(mm)	34.08
Connecting rod length (mm)	120

V. RESULTS

5.1 Air mass fraction contours for flat headed piston

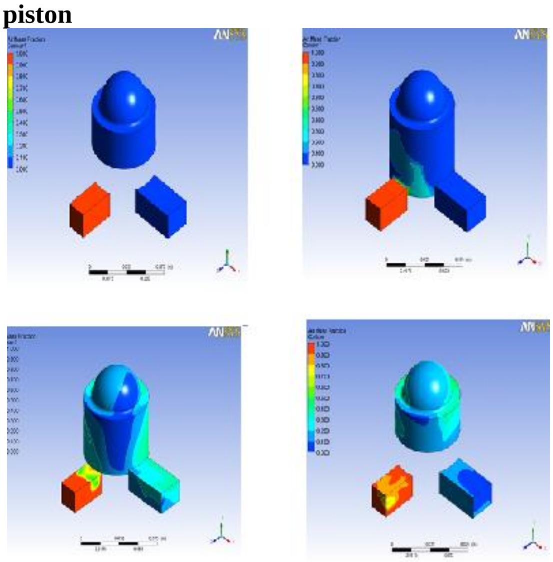

Fig 3 Air mass fractions at 0, 75, 150, 210 degree crank angles.

5.2 CO2 Mass fraction for flat headed piston

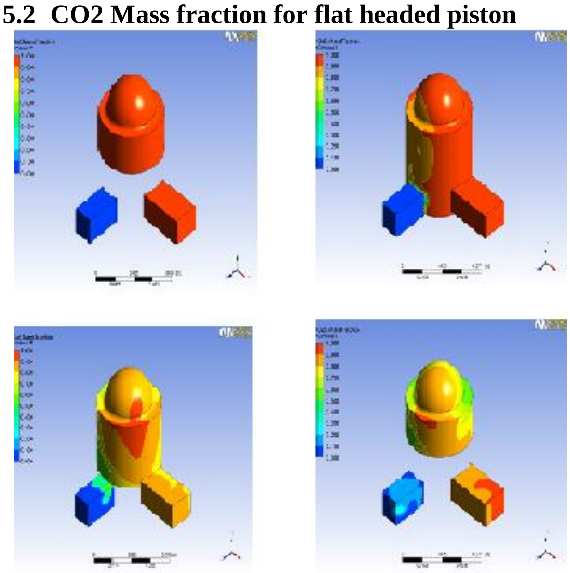

Fig 4:co2 mass fractions at 0, 75, 150,210 degree crank angle.

5.3 Air-Mass fraction at dome headed piston

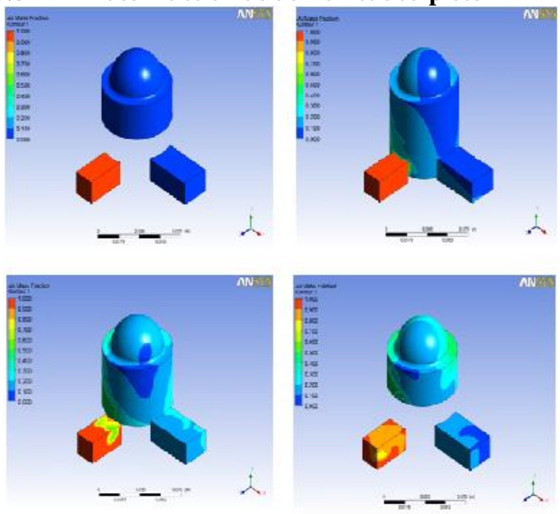

Fig 5: Air- mass fractions at 0, 75, 150,210 degree crank angle.

5.4CO2-Mass fraction at dome headed piston

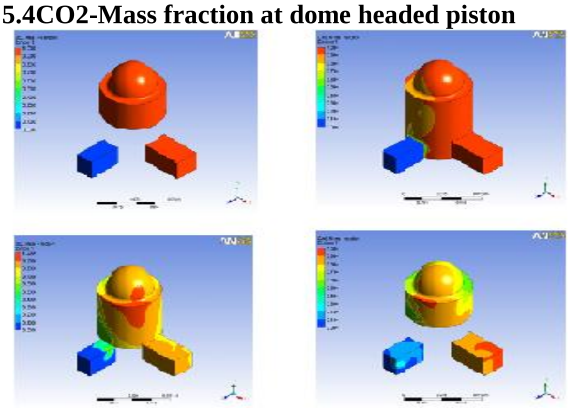

Fig 6: co2 mass fractions at 0, 75, 150,210 degree

Mass fraction fields for Air and CO2 were shown above, in that for air mass fraction red shows that inlet is completely filled with fresh air and in chamber blue indicates the burned gases. But in CO2 mass fraction, inlet with fresh charge is shown by blue colour and burned gases with red colour in the chamber.

Consider the co2 mass fraction, we can see that after combustion at 0 degree crank angle engine cylinder completely filled with co2 and inlet with fresh air .At 75 (deg) some quantity of co2 get displaced and that position is occupied by fresh air. This continues till the final stage and maximum quantity of burned gas get expelled by fresh air. By analyzing the quantity of burned gases inside the exhaust ,the scavenging efficiency is calculated.

$$\eta = \frac{\text{Mass of air retained in exhaust}}{\text{Mass of mixture in the cylinder}}$$

Scavenging efficiency for flat headed piston is 45.03 % and it is for dome headed piston is 49.3%. By applying the same operating conditions for both the cases, scavenging efficiency is higher for dome headed piston. Hence in this way we can identify the better engine design condition also for better performance

5.5 Comparison of Air Mass fraction

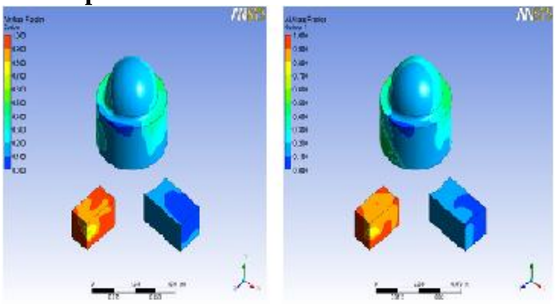

Fig 7 Air Mass fraction at 210 (deg) for flat & dome shaped pistons respectively.

The above given figure shows the air –mass fraction at 210 degree crank angle where maximum quantity of burnt gases is expelled by fresh air in both the cases. From the figure itself it is clear that ,dome headed piston displaces more exhaust gas than flat type piston.

5.3 Comparison of CO2-Mass fraction

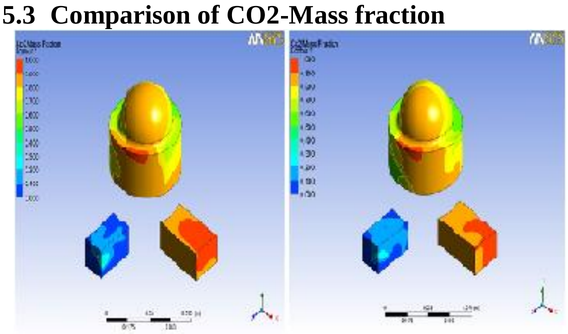

Fig 8 CO2-Mass fraction at 210 (deg) for flat & dome shaped pistons respectively.

From Fig 8 the exhaust port in dome headed piston (second figure) showing lesser place with red colour, that means more quantity of co2 is expelled through it than comparing with the flat piston.

Finally, the mass fraction field is useful for checking the filling of fresh gases into the cylinder and detecting problems of short circuiting and gas drag.

5.4 Comparison of pressure contours.

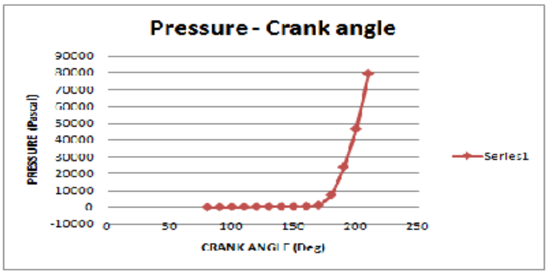

Fig 9: Pressure Crank angle graph for flat piston

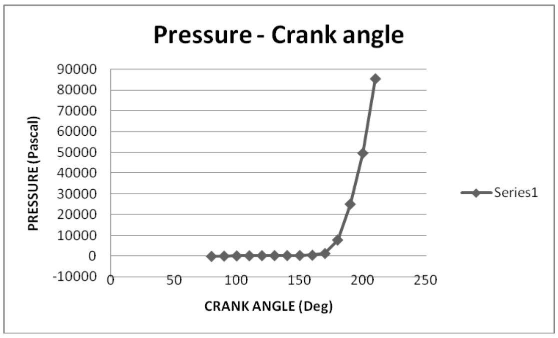

Fig 10: Pressure Crank angle graph for dome headed piston

On comparing both the pressure – crank angle graphs, it is clear that at 210 (deg) crank angle 79361.3 pascal for flat piston and 85467.9 pascal for dome headed piston. These graph showing gauge pressure values at various crank angle during the scavenging process. When the pressure inside the cylinder increases the expelling of gas through exhaust port will also increases and thereby increases the scavenging. That means for dome headed piston is more effective than flat headed piston.

5.5 Comparison of temperature contous.

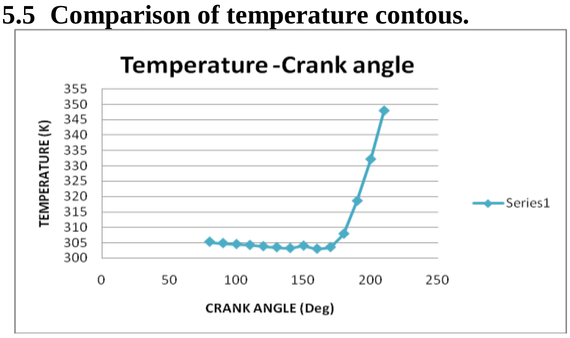

Fig 11 : Temperature –Crank angle diagram for flat headed piston.

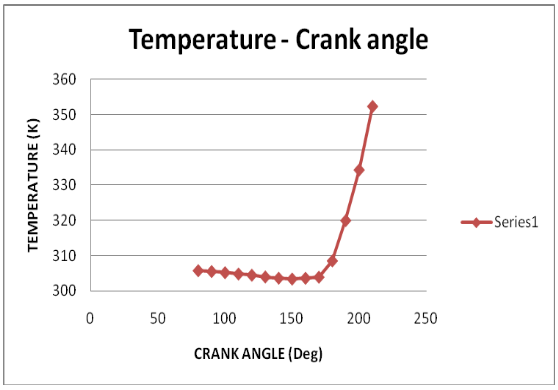

Fig 12: Temperature crank angle for dome headed piston

On comparing both the temperature – crank angle graphs, at 210 (deg) crank angle 348 k for flat piston and 352.4 for dome headed piston.

VI. CONCLUSION

This CFD analysis was carried out to study the scavenging process of two-stroke engines. In general, this study shows that CFD predictions yield reasonably accurate results that allow improving the knowledge of the fluid flow characteristics. This model is very useful to design the scavenging system of new two-stroke engines. From the various results shown such as pressure plots, temperature plots, mass fraction plots, and velocity plots it is possible to identify various problem regarding the engine performances, for example the pressure field is useful for identifying areas where the gas flow is inefficient and should be corrected. The velocity field is useful for locating areas with too high, too low or inadequate velocities. The process of verification and validation of a CFD model is necessary to ensure the numerical model accurately captures the physical phenomena present. By comparing numerically obtained results with experimental results, in the numerical model is achieved. Once thoroughly validated, a numerical model may be used for accurate analysis.

From the analysis presented, it is clear that the scavenging process in dome headed piston is more efficient than flat headed piston. Now it is finalized that all the processes in combustion are capable of analyzing in CFD method with more accurately than experimentally to some extent. From the data we obtained the analysis of scavenging, some ways to increase the efficiency are found. Viz,

- Increasing the inlet pressure.
- Changing the angle between the ports.
- Changing the angle of inlet port.

It is hoped that the result of this analysis will help to evolve a better method to design the shape of piston head in SI engines.

6.1 Future work

In the present scope of the project ,the experimental investigation is not included ,as it involves extensive measurements for pressure ,temperature ,velocity etc ,for which time constraints and the unavailability of required instruments are two major limitations.

This experimental investigation can be taken up as a future work ,for compatible validation of the CFD analysis..

REFERENCES

[1] *A computational study of free-piston diesel engine combustion* R. Mikalsen, A.P. Roskilly

[2] *Analysis of Combustion and Emissions in a Large Two-Stroke Marine Diesel Engine, Using CFD and T-φ Mapping*

[3] *CFD Modeling and Experimental Validation of Combustion in Direct Ignition Engine Fueled with Diesel.* ISSN 09764259.

[4] *CFD-based optimization of fuel injection strategies in a diesel engine using an adaptive gradient method* Franz X. Tanner, Seshasai Srinivasan.

[5] *CFD modeling of heat transfer and fluid flow inside a pent-roof type combustion chamber using dynamic model*.

[6] *Computerised Simulation of Spark Ignition Internal Combustion Engine, e-ISSN: 2278-1684 Volume 5, Issue 3 (Jan. - Feb. 2013), PP 05-14*

[7] *Development of combustion system in low speed two-stroke diesel engine using CFD*,paper number 208.

[8] Heywood, John B , "*Internal combustion engines fundamentals*", 1988,pp:668-671

[9] Philipp A, lang O, Schutz R. and Weng V., 2002 "*CFD simulation of diesel injection and combustion* ", SAE, 2002010945.

[10] Simulation Strategy and Analysis of a Two-Cylinder Two Stroke Engine Using CFD Code Fluent, Dalibor Jajcevic,Raimund A. Almbauer, Christian Doppler Laboratory "*Thermodynamics of Reciprocating Engines*", Graz University of Technology, Austria Stephan P. Schmidt Institute for Internal Combustion Engines and Thermodynamics.