Report on

Variation in combustion process due to change in geometry for different cylinder heads using ANSYS Fluent

Submitted in partial fulfilment of the course ME F266: Study Project

by

Jalaj Maheshwari (2011A4PS224G)

Shriniket Jakati (2011B5A4459G)

Under the supervision of

Dr. Varinder Singh

Assistant Professor, Department of Mechanical Engineering

BIRLA INSTITUTE OF TECHNOLOGY AND SCIENCE, PILANI

K.K. BIRLA GOA CAMPUS

29th November, 2014

Acknowledgements

We are very grateful to Dr. Varinder Singh, Assistant Professor at the Department of Mechanical Engineering, for mentoring us and giving us the opportunity to have a great learning experience while doing this project and without whom this particular study project could not have come through.

We are also thankful to Dr. Shibu Clement, instructor-in-charge of the course ME F266: Study Project.

Abstract

The study of combustion in engines has been under significant pressure for improvement. In this project we study the effects of varying geometry on the different parameters of the engine during combustion in a 4-stroke Compression Ignition (CI) engine operating on a diesel cycle, using diesel as a primary fuel.

We first discuss the various types of Internal Combustion engines and briefly discuss the cycles they work on, particularly the Otto cycle and the Diesel cycle. We have further defined two distinct geometries which we use for our simulations. Here we make a crude approximation about the combustion process involving the power stroke. We then move on to define the meshing that is done in order to proceed with the calculations.

The solution initialization and the boundary conditions are then set corresponding to our approximations, and the simulation is then carried of the two cylinder heads. The results are the noted and subsequently the conclusions are drawn.

CATIA V5 helps us prepare the geometry and the ANSYS Workbench – FLUENT helps us in carrying out our study and simulating the particular problem at hand.

Keywords: IC Engines, CFD, Energy Equation, (k-ε) Model, Non-Premixed Combustion

TABLE OF CONTENTS

1 INTRODUCTION

1.1 THE INTERNAL COMBUSTION ENGINE

An engine is a device which transforms one form of energy into another form. A heat engine is a device that converts the chemical energy derived from a particular fuel to thermal energy and further, uses this thermal energy to perform useful work. The two types of heat engines are:

- Internal Combustion (IC) Engine
- External Combustion (EC) Engine

The most widely used engine is the Internal Combustion Engine. All components of an IC Engine work at an average temperature which is much lower than the maximum temperature of the working fluid. This is because the high temperature of the working fluid persists for a very short duration of time. Further, higher thermal efficiencies can be achieved with fairly moderate pressure of the working fluid in the cycle. The disadvantage of the Internal Combustion Engine is the vibration caused by the reciprocating parts (Ganesan, 1994). Moreover, only certain fluids can be employed as the working fluid, which have specifications pertaining to the engine. Thus, Internal Combustion Engines are widely used in automobiles, aircrafts, ships and power units such as generators.

Types of IC Engines

IC Engines are broadly classified into two types:

1. *Rotary Engines*
   The rotary engine is a type of internal-combustion engine with in a radial configuration, in which the crankshaft remained stationary, with the entire crankcase and its attached cylinders rotating around it as a unit in operation. This type of engine was a relatively older engine than its much recent counterparts and was primarily used in aircrafts, but also found use in automobiles and motorcycles.

   A commonly used rotary engine is the Wankel engine. It is a type of internal combustion engine using an eccentric rotary design to convert pressure into rotating motion. The Wankel engine holds a certain advantage over the reciprocating engines in simplicity, smoothness, compactness, high rates of revolution, and a high power-to-weight ratio.

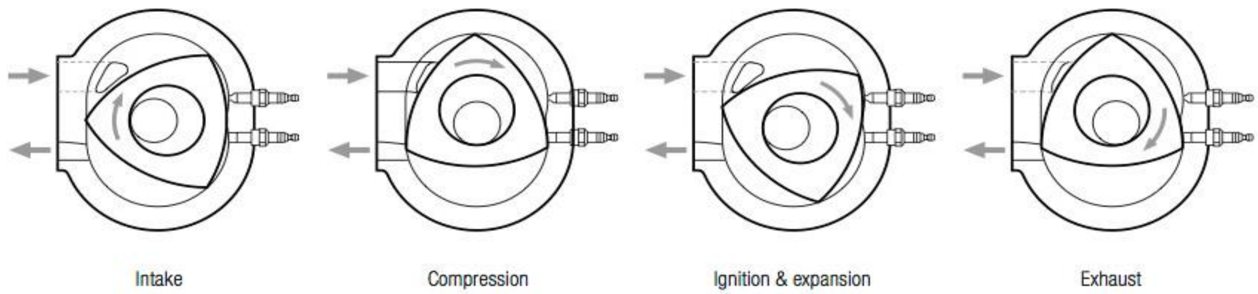

Figure 1: Working of Wankel engine (Source: autoevolution.com)

2. *Reciprocating Engine*
   A reciprocating engine or the piston engine, is a heat engine that uses one or more reciprocating pistons to convert pressure into a rotating motion. The different components of a reciprocating engine are shown.

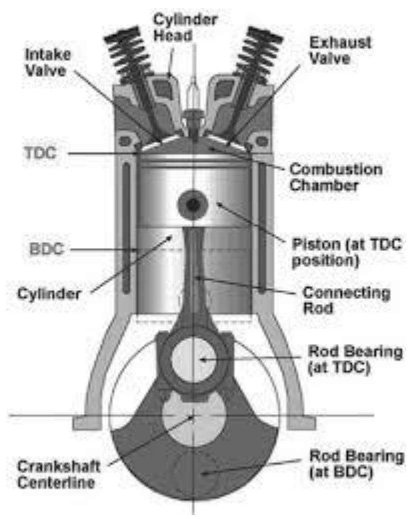

Figure 2: Model of Reciprocating Engine (source: what-when-how.com)

These reciprocating engines can be classified into different categories based on the number of strokes it takes to complete one cycle of operation. They can be classified as:

- 2-Stroke Engines
- 4-Stroke Engines

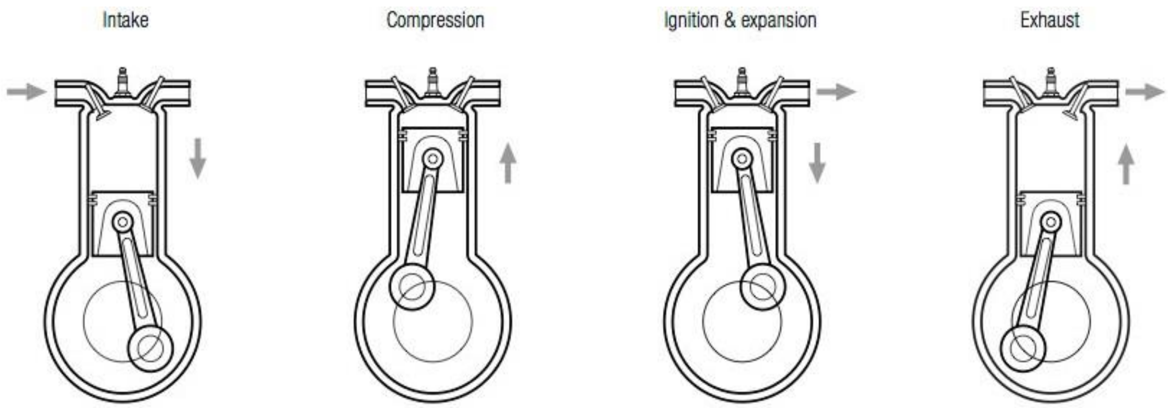

Figure 3: Working of 4-stroke Engine (source: autoevolution.com)

The main difference between the two is in the method of filling of new charge and removal of burnt gases from the cylinder. In 4-stroke engines, this operation is performed by the engine piston during suction and exhaust strokes. In 2-stroke engines, the filling process is accomplished by the charge compressed in the crankcase. The induction of compressed charge moves the product of combustion out through the exhaust ports.

On the basis of working principle, reciprocating engines can be classified into two types:

a) ***Spark-Ignition Engines***

The spark-ignition engine refers to internal combustion engines where the combustion process of the fuel-air mixture is ignited by a spark from a spark plug. Petrol engines are spark ignition engines. The working cycle for SI engines can either be two-stroke or four-stroke. The spark plug is positioned in the cylinder head which gives a pulse of electricity when the fuel-air mixture needs to be ignited. The spark plug is positioned in such a way that the tendency of knocking is minimized and there is a symmetric advance of the flame front. The electric discharge is controlled and is not random (Broeze, 1963). The duration of the spark, its size, intensity and the energy release are all factored according to the fuel being used. Different fuels require different energy pulse magnitudes to be combusted.

A four-stroke spark-ignition engine works on the Otto and it consists of following four strokes:

- Intake or suction stroke
- Compression stroke
- Expansion or power stroke
- Exhaust stroke

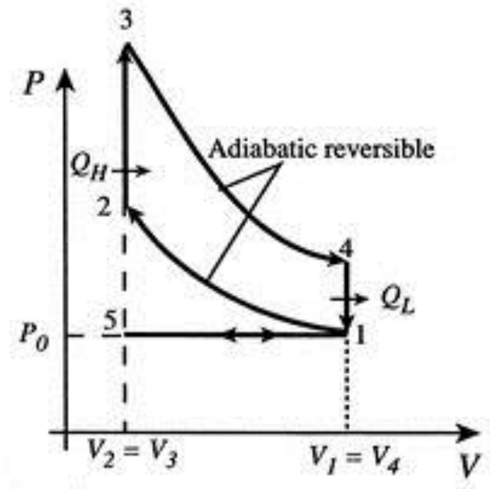

Figure 4: Otto cycle P-V diagram (source: web.mit.edu)

The different processes are shown in the figure:

1. Intake stroke (5 → 1).
2. Compression stroke (1 → 2).
3. Combustion of mixture (spark) (2 → 3).
4. Power stroke, expansion of combustion mixture (3 → 4).
5. Heat rejection (4 → 1).
6. Exhaust stroke, piston pushes remaining combustion products out of chamber (1 → 5).

b) *Compression Ignition Engines*

Compression ignition engines are similar to SI engines except for the fact that they do not consist of a spark plug. Moreover, the compression ratio for CI engines (16 to 20) is higher than that of SI engines (6 to 10). In CI engines, during the suction stroke, air, instead of an air-fuel mixture in inducted into the combustion chamber. Due to the high compression ratios the temperature at the end of the compression stroke is high enough to self-ignite the fuel which is injected into the combustion chamber through the fuel injector. The compression ratios in CI engines are always large as they do not have an external energy discharge in the form of a spark as in the case of SI engines. All the energy generated is because of the rise in the internal energy of the system.

A typical four-stroke CI engine operates on the Diesel cycle. It consists the same four strokes as the SI engine:

- Intake or suction stroke
- Compression stroke
- Expansion or power stroke
- Exhaust stroke

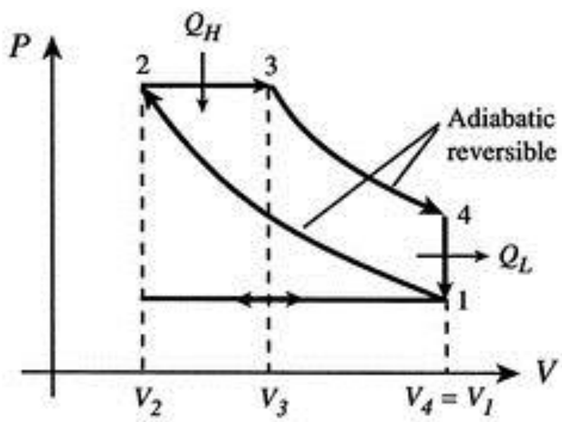

Figure 5: Diesel Cycle P-V Diagram (source: web.mit.edu)

The different processes are shown in the figure:

1. Intake and suction stroke (up to 1 at constant pressure).
2. Compression stroke (1 → 2).
3. Fuel injection, combustion of mixture (2 → 3).
4. Power stroke, expansion of fuel mixture (3 → 4).
5. Heat rejection (4 → 1).
6. Exhaust stroke.

1.2 ANSYS FLUENT

Figure 6: ANSYS Fluent Logo (source: ansys.com)

Solving complex fluid and thermal problems for scenarios in the real life takes tedious amounts of time when done theoretically. Thus, the need for computational simulations for the same is imminent. ANSYS Fluent software contains the broad physical modeling capabilities needed to model flow, turbulence, heat transfer, and reactions for industrial applications ranging from air flow over an aircraft wing to combustion in a furnace, from bubble columns to oil platforms, from blood flow to semiconductor manufacturing, and from clean room design to wastewater treatment plants. Special models that give the software the ability to model in-cylinder combustion, aeroacoustics, turbomachinery, and multiphase systems have served to broaden its reach.

Today, thousands of companies throughout the world benefit from the use of ANSYS Fluent (www.ansys.com) software as an integral part of the design and optimization phases of their product development. Advanced solver technology provides fast, accurate CFD results (Versteeg, 1995), flexible moving and deforming meshes, and superior parallel scalability. User-defined functions allow the implementation of new user models and the extensive customization of existing ones. The interactive solver setup, solution and post-processing capabilities of ANSYS Fluent make it easy to pause a calculation, examine results with integrated post-processing, change any setting, and then continue the calculation within a single application.

2 GEOMETRY

2.1 COMBUSTION APPROXIMATION

The fuel is injected into the combustion when the engine piston reaches the top dead center in the combustion chamber (Broatch, 2007). Due to the high pressure and subsequent temperature build up, the temperature is high enough to spontaneously combust the air-fuel mixture in the combustion chamber. The process 2-3 is a thus, a constant pressure heat addition process, where the temperature inside the combustion chamber rises to a maximum value (Kurniawan, 2007).

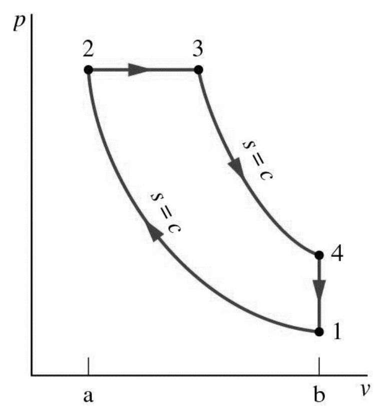

Figure 7: Constant Pressure Process 2-3 (source: www.mae.wvu.edu)

The following study is the analysis of the combustion stroke as a finite volume and pressure process. This is a crude approximation but it can be made, as the time it takes for the majority of the fuel-air mixture to get combusted is very small and hence the piston displacement can be considered to be small enough to be neglected. The combustion initiation occurs in a very small volume, usually the clearance volume. This approximation is done so that a simpler analysis can be performed for the above mentioned process.

2.2 GEOMETRY PREPARATION

To study the combustion variation in different cylinder heads, individual cylinder heads were modelled. The designing of each cylinder head was done using CATIA V5. Each model depicts the time frame of the cycle where the piston is at the Top Dead Center (TDC) and fuel is just injected in the combustion chamber and momentary combustion takes place.

2.2.1 CURVED CYLINDER HEAD

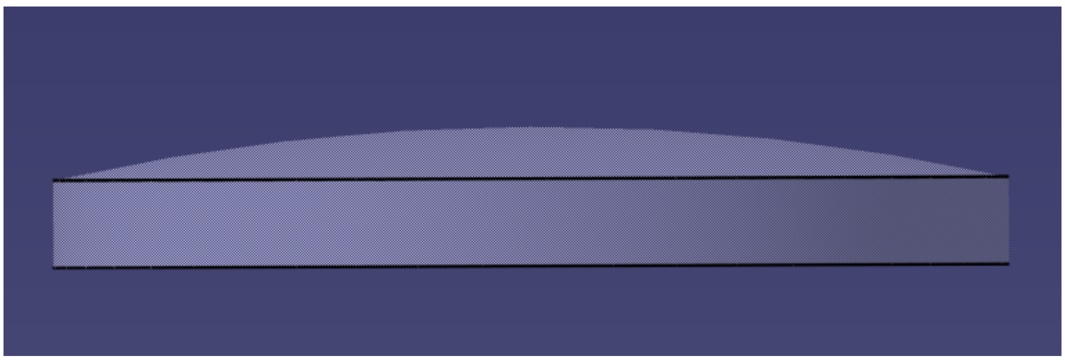

Figure 8: Curved Cylinder Head - Front View

The geometric specifications are:

- Bore : 80 mm
- Stroke : 65.7 mm
- Comp. Ratio : 9
- Clearance Height : 7.3 mm

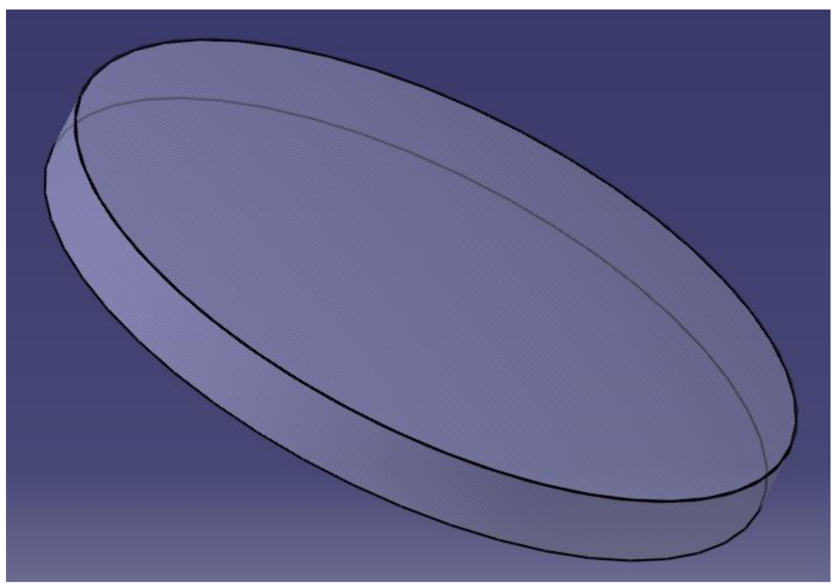

Figure 9: Curved Cylinder Head - Isometric View

2.2.2 STEPPED CYLINDER HEAD

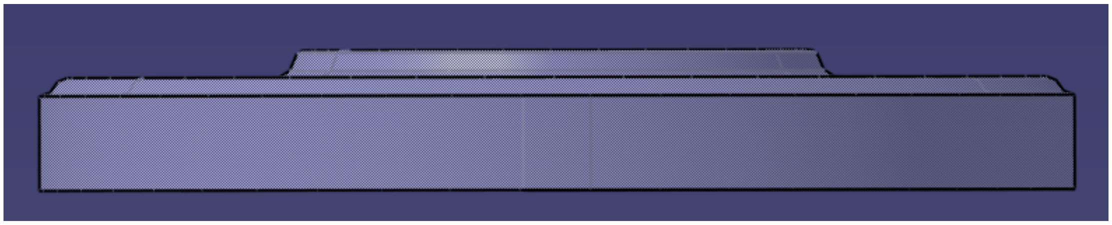

Figure 10: Stepped Cylinder Head - Front View

The geometric specifications are:

- Bore : 80 mm
- Stroke : 65.7 mm
- Comp. Ratio : 9
- Clearance Height : 7.3 mm

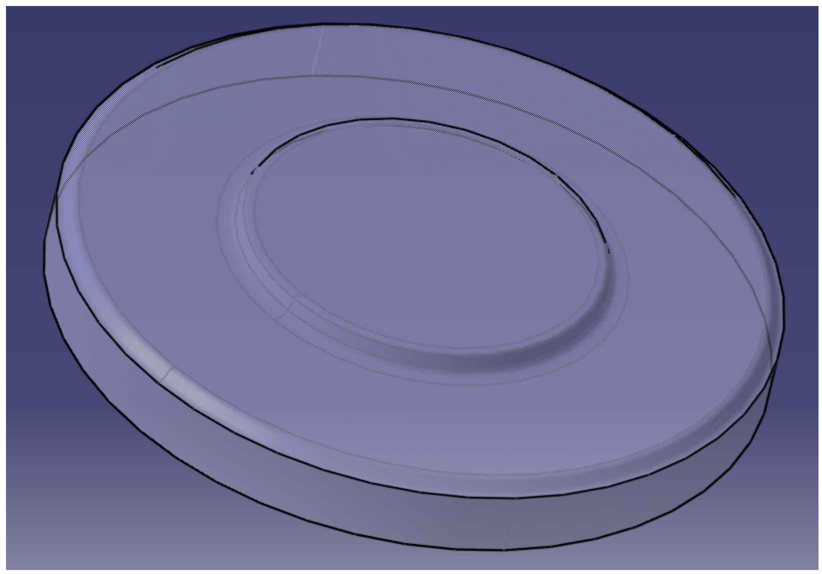

Figure 11: Stepped Cylinder Head - Isometric View

The bore length, stroke length and the compression ratio for all the geometries is kept constant in order to study the variation of combustion due to geometrical differences in the cylinder head design. In the mentioned cases, the clearance height is also kept the same.

3 MESHING

Each model developed for analysis is not the solid model for the engine head, but the fluid inside the engine head that will undergo combustion. This is a relatively easier method of solving computational fluid problems and properties of the fluid and boundary conditions can be set comparatively easily. After the geometry has been imported, the next step involve meshing of the geometry in order to solve our problem.

In computational fluid dynamics, meshing is a discrete representation of the geometry that is involved in the problem (Cebeci, 2005). Essentially, it assigns cells or smaller regions over which the flow is solved. Several parts of the mesh are grouped into regions where boundary conditions may be applied to solve the problem.

The meshing of a 3-dimensional geometry consists of coarse meshes and fine meshes. Fine geometry meshes give more accurate calculations are results as compared to the coarse meshing. However, the CPU processing time for the same is substantially longer than the coarse mesh. Thus, an optimum mesh size should be considered (Kong, 2003) to get accurate results, as well as reduce the meshing time.

3.1 MESHING FOR CURVED CYLINDER HEAD

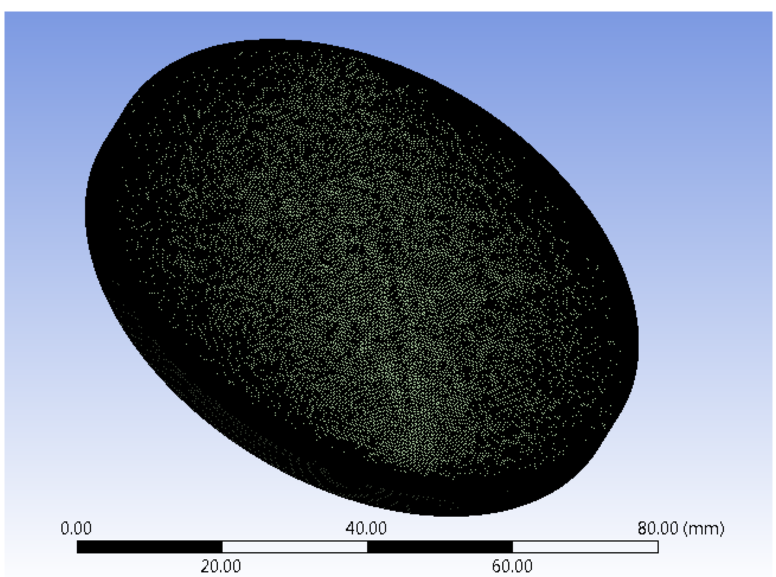

Figure 12: Curved Head Meshing - Isometric View

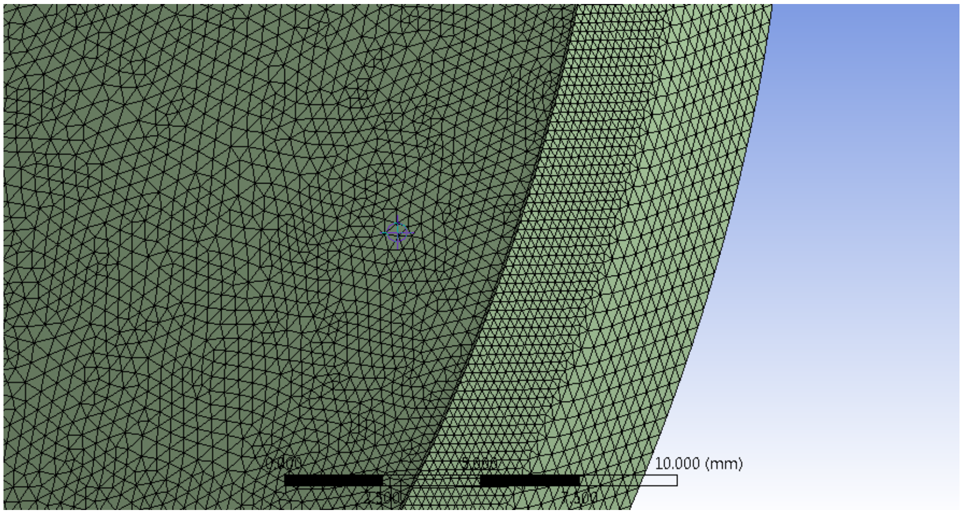

Figure 13: Curved Head Meshing - Close View

3.2 MESHING FOR STEPPED CYLINDER HEAD

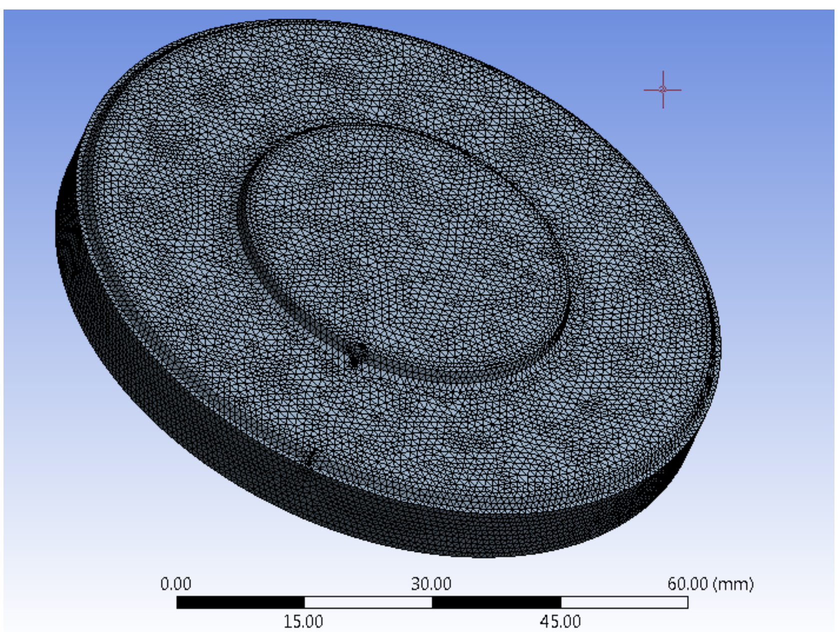

Figure 14: Stepped Head Meshing - Isometric View

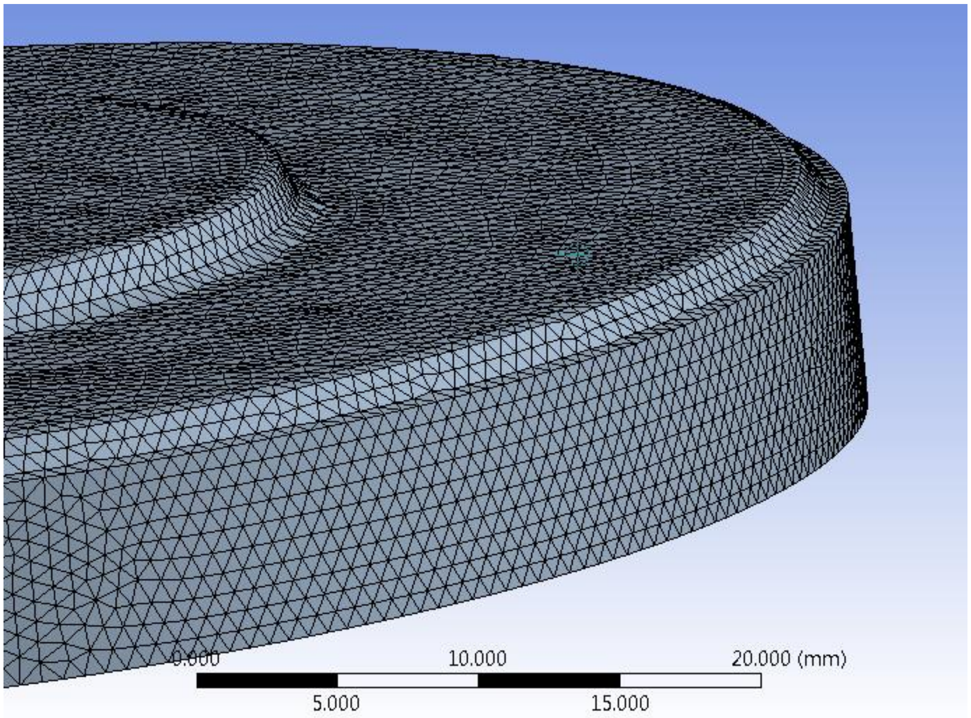

Figure 15: Stepped Head Meshing - Close View

The requirements of the computing mesh are:

1. It should properly fit the topography of the combustion chamber.
2. It should allow control of local resolution to obtain the maximum accuracy with a given number of grid points.
3. It should have the property that each interior grid is connected to the same number of neighboring points.

The first requirement tells that the whole model geometry needs to be simulated (Fletcher, 1991). The second requirement tells that the computing time increases with the number of mesh points while the third requirements tells that the mesh need to be in topologically appropriate shape so that highly efficient equations can be utilized.

4 SOLUTION INITIALIZATION

In order to simulate the combustion process, the different boundary conditions have to be set and different models have to be used to take into account the various aspects of the combustion process (Golovitchev, 2007). The setup for a compression ignition combustion requires enabling different equations and models in order to get the correct results.

4.1 GENERAL SETUP

For the given setup, the solver time setting is kept steady in order to keep the analysis simple and to obtain a momentary plot of the variation in parameters during the combustion process. We used a pressure-based solver with absolute velocity formulation.

4.2 MODEL

In order to simulate the compression ignition based combustion, various different models have to be enabled to solve the given problem based on the boundary conditions provided.

4.2.1 ENERGY EQUATION

The Energy Equation Model is the equation that governs the total heat transfer that occurs within the system, and also how it progresses throughout the geometry under study. The energy equation is given by:

$$\frac{\delta(\rho E)}{\delta t} + \nabla.[\vec{V}(\rho E + p)] = \nabla.\,[k_{eff}\nabla T - \sum_j h_j J_j + (\overline{\overline{\tau_{eff}}}\,.\vec{V})] + S_h$$

where, $k_{eff}\nabla T$ represents the conduction,

$\sum_j h_j J_j$ represents the species diffusion,

$\overline{\overline{\tau_{eff}}}.\vec{V}$ represents the viscous dissipation,

and E is the energy per unit mass represented by:

$$E = h - \frac{p}{\rho} + \frac{V^2}{2}$$

4.2.2 (k-ε) TURBULENCE MODEL

This the most common model used in Computational Fluid Dynamics to simulate mean flow characteristics for turbulent flow conditions (Aglave, 2008). It is a two equation model which gives a general description of turbulence by means of two transport equations.

1) *Turbulent Kinetic Energy (k)*

The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k).

2) *Turbulent Dissipation (ϵ)*

The second transported variable is the turbulent dissipation (ϵ) which determines the rate of dissipation of the turbulent kinetic energy.

The values of k and ε are obtained from the following (Wikipedia) two equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k$$

and

$$\frac{\partial}{\partial t}(\rho\epsilon) + \frac{\partial}{\partial x_i}(\rho\epsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right] + C_{1\epsilon}\frac{\epsilon}{k}(G_k + C_{3\epsilon}G_b) - C_{2\epsilon}\rho\frac{\epsilon^2}{k} + S_\epsilon$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients, G_b is the generation of turbulence kinetic energy due to buoyancy, and Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\epsilon}$, $C_{2\epsilon}$ and $C_{3\epsilon}$ are constants. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ , respectively. S_k and S_ϵ are user-defined source terms (Bakshan, 2000).

The standard k-ε model which governs the following two equations which can be summarized as:

Rate of change of k or ε		Transport of k or ε by diffusion +
+	=	Rate of production of k or ε -
Transport of k or ε by convection		Rate of destruction of k or ε

4.2.3 RADIATION

While solving the given combustion problem, another model called the radiation model (Ilbas, 2005) has to be enabled in order to account for the radiative heat transfer.

For the given problem, the P1 radiation model is chosen (Knight, 1964) due to the following reasons:

- The P-1 model can be described by a transport equation for radiation temperature, which is a conservative equation and can be easily incorporated into CFD.
- The P-1 model allows relatively simple implementation of radiative exchange between droplets and gas.

4.2.4 SPECIES

Since we assume that the combustion initiation takes place in a finite volume for a short duration, the fluid volume that is modeled is assumed to be the air inside the combustion chamber. The fuel used i.e. diesel, is injected through the fuel injector. As a result, we use non-adiabatic non-premixed combustion model to simulate the problem.

In non-premixed combustion, the fuel and the oxidizer are present in different streams and enter the reaction zone differently to combust. This is unlike the premixed combustion models, in which reactants are mixed at the molecular level before burning. This type of model is used in diesel combustion and is thus, incorporated while solving the given simulation (Nishida, 1989). The operation parameters have to be set according to the simulation and hand and can be adjusted in this particular model.

Moreover, the fuel for a non-premixed combustion model has to be specified. Since the corresponding simulations involve diesel as a fuel, we choose diesel as the primary fuel in the mixture from the database of pre-existing fuels in ANSYS (ANSYS, 2006). Since the average molecular formula for diesel turns out to be $C_{12}H_{23}$, we use the same hydrocarbon from the ANSYS database to run the simulations.

4.3 BOUNDARY CONDITIONS

In every CFD simulation, the boundary conditions to the geometry have to be provided so as to use the necessary equations and models to solve a particular problem (Kong, 1995). Thus, following are the basic boundary conditions that are enforced on the models.

4.3.1 FUEL INLET

Fuel inlet is set to mass flow rate inlet (Palipana, 1999), thus, representing the fuel that flows into the combustion chamber. The mass flow rate for the modelled geometries can be found out from the net power they deliver.

Considering the net power delivered by both geometries as the same and about 35 Kilowatts, by using the calorific value of the fuel and the number of fuel injectors, the mass flow rate for the geometries in calculated.

Power = 35×10^3 *Watts*

Calorific value = 43.4×10^6 *J/kg*

Thus, total mass flow rate = $\frac{Power}{Calorific\ Value}$ = *0.0081 kg/s*

Let no. of fuel injectors = *n*

Thus, mass flow rate per injector = $\frac{Total\ mass\ flow\ rate}{No.of\ fuel\ injectors}$

Thus, in this particular case, the mass flow rate through the fuel injector is equal to 0.0081 kg/s.

Since the net power delivered by both the cylinder head geometries is taken the same, and the fuel is kept the same i.e. diesel, the mass flow rate in both cases will be equal to 0.0081 kg/s.

4.3.2 OUTLET

The bottom base of the geometry is taken as a pressure outlet, to incorporate the effect of the cylinder piston. The pressure set is very high to act as a piston and consequently prevent the combustion mixture from flowing out of the combustion volume. This ensures a constant pressure process in the diesel cycle and acts as a crude approximation while simulating the combustion process.

5 RESULTS

Based on the simulation that was performed for the two different geometries, a series of contour plots are obtained which show the variation of different parameters over the geometry during the combustion process. These contours obtained are momentary plots obtained after the fuel has been injected into the combustion chamber.

5.1 PRESSURE (Pa)

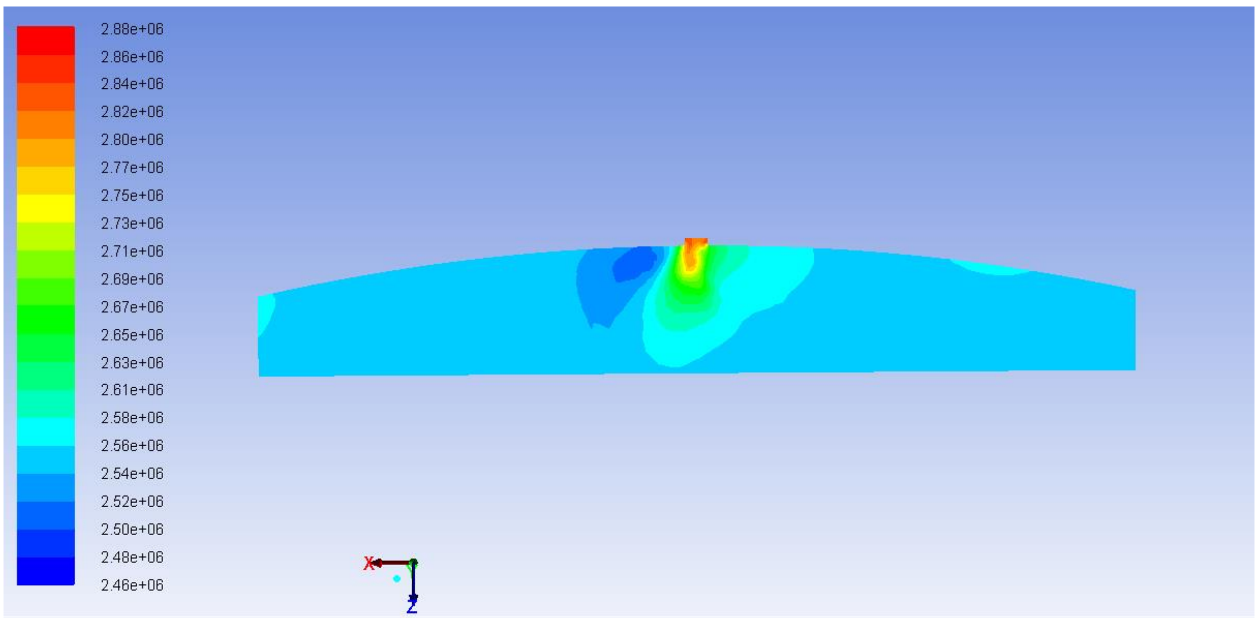

Figure 16: Pressure Contour for Curved Cylinder Head

The unit of pressure in both the cases is in Pascal and the internal pressure in both cases is consistent with the optimum pressure required during the combustion process.

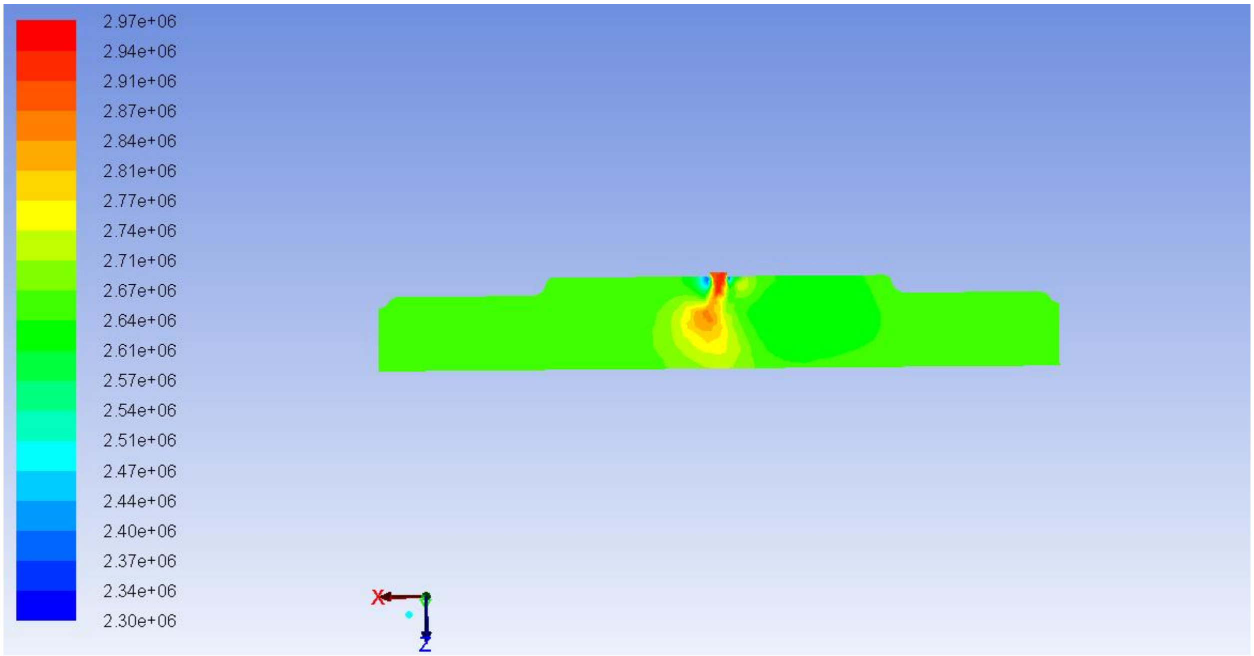

Figure 17: Pressure Contour for Stepped Cylinder Head

In both geometries, it can be seen that the pressure is maximum near the region where the fuel is injected into the combustion chamber, and gradually decreases as we move away from the fuel injector and as combustion progresses through the cylinder.

However, in the stepped cylinder head we can see that the overall average pressure in the cylinder is of a higher magnitude than it is in the curved cylinder head. This would mean that the stepped cylinder has a higher operation pressure and hence has slightly better efficiency.

Since there is sudden increase in the pressure when the fuel combusts, the engines should be designed in such a way as to compensate for the high pressure and should be able to withstand the high pressure in order to prevent excessive knocking conditions.

5.2 TEMPERATURE (K)

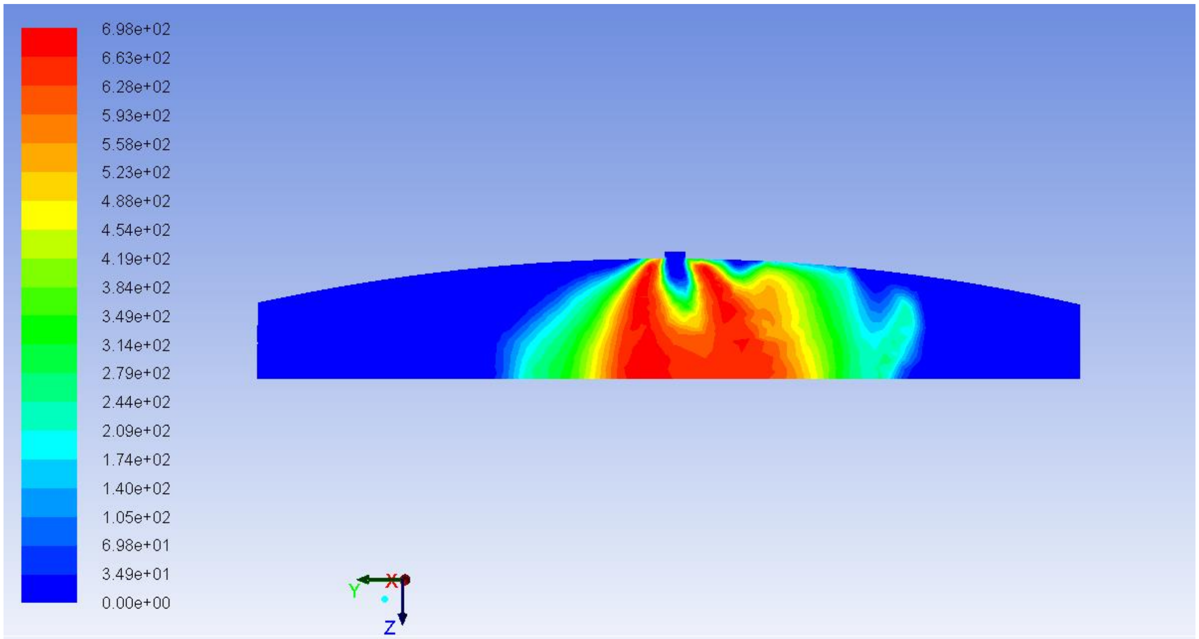

Figure 18: Temperature Contour for Curved Cylinder Head

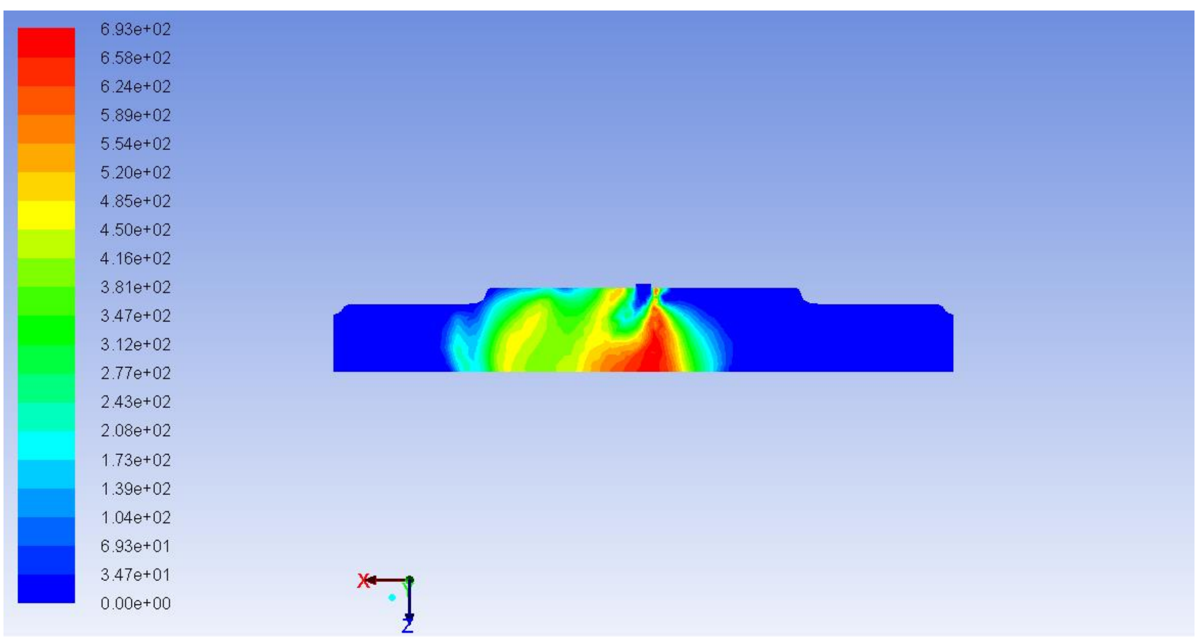

Figure 19: Temperature Contour for Stepped Cylinder Head

The temperature contour for both cylinder heads shows that there is no significant difference in the temperature inside the combustion chamber and the magnitude of maximum temperature is nearly the same. But it is worth mentioning that in the curved cylinder head, we have a larger volume with a higher average temperature as compared to the stepped cylinder head.

5.3 TURBULENCE (%)

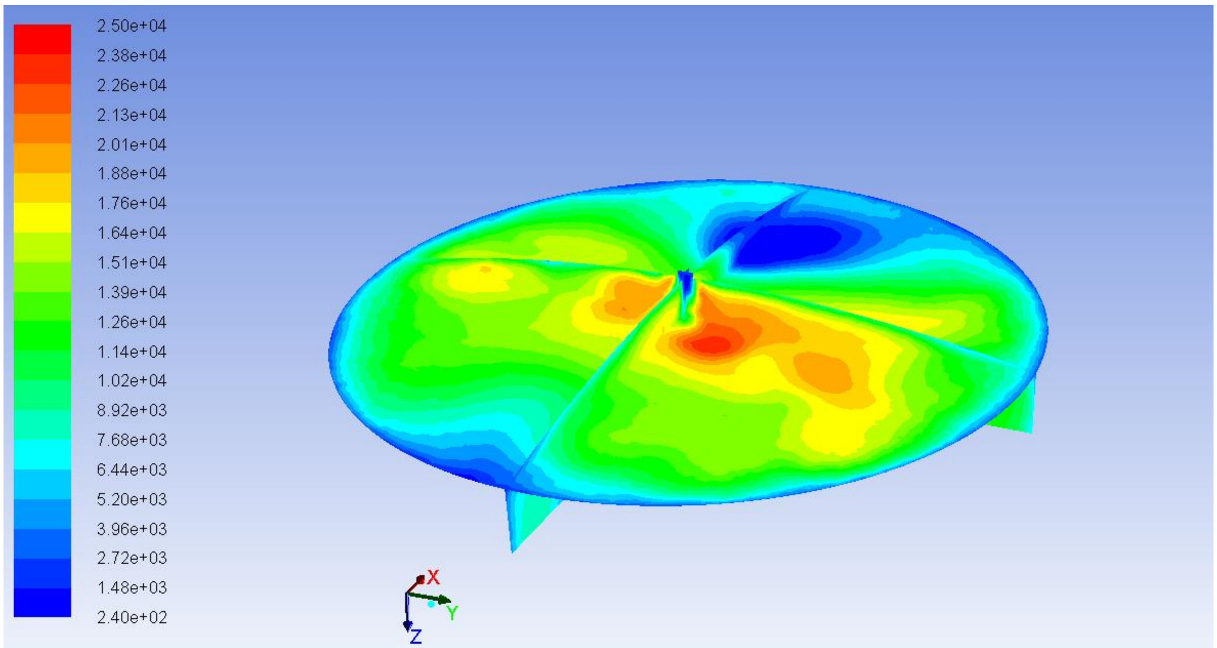

Figure 20: Turbulence Contour for Curved Cylinder Head

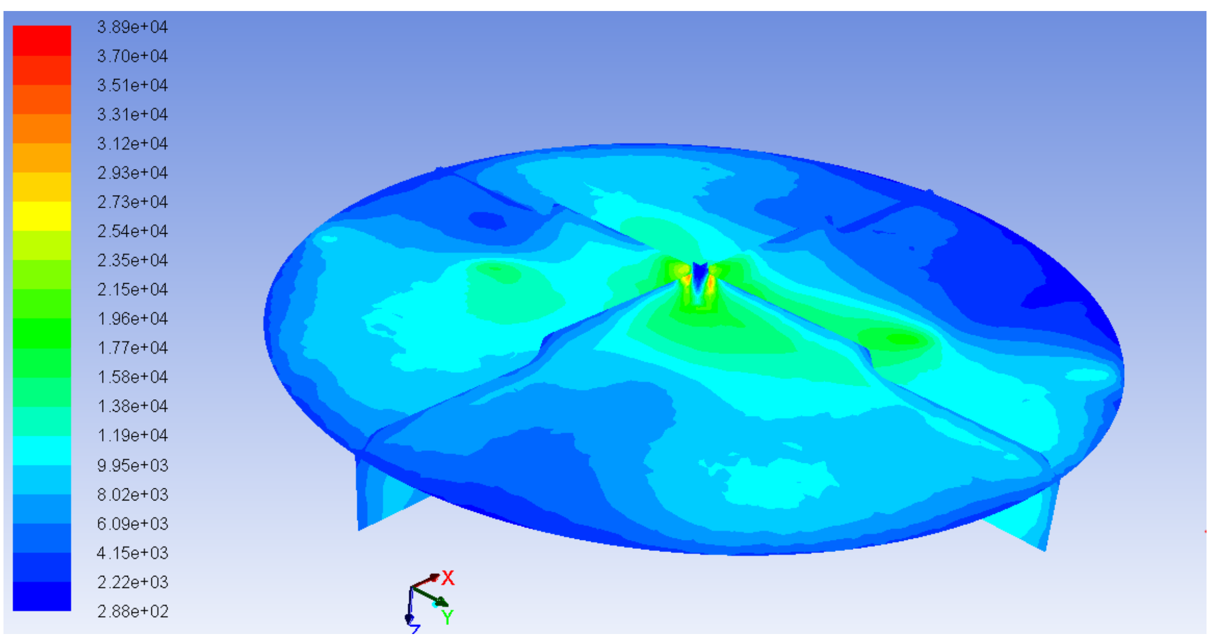

Figure 21: Turbulence Contour for Stepped Cylinder Head

The turbulence intensity contour plots show that the curved cylinder head has a more turbulent mixture and hence, mixing of the fuel and air is promoted which leads to cleaner combustion. In the stepped cylinder head we see that the overall turbulence is lower as compared to the curved head, but is more evenly spread out and unlike the counter head sign, there are no local high turbulence zones. The occurrence of local high turbulence can cause the flame front to dissipate quickly as there is no definite progression available to the flamelet, which can lead to an unfavorable dip in power.

5.4 OXYGEN MASS FRACTION

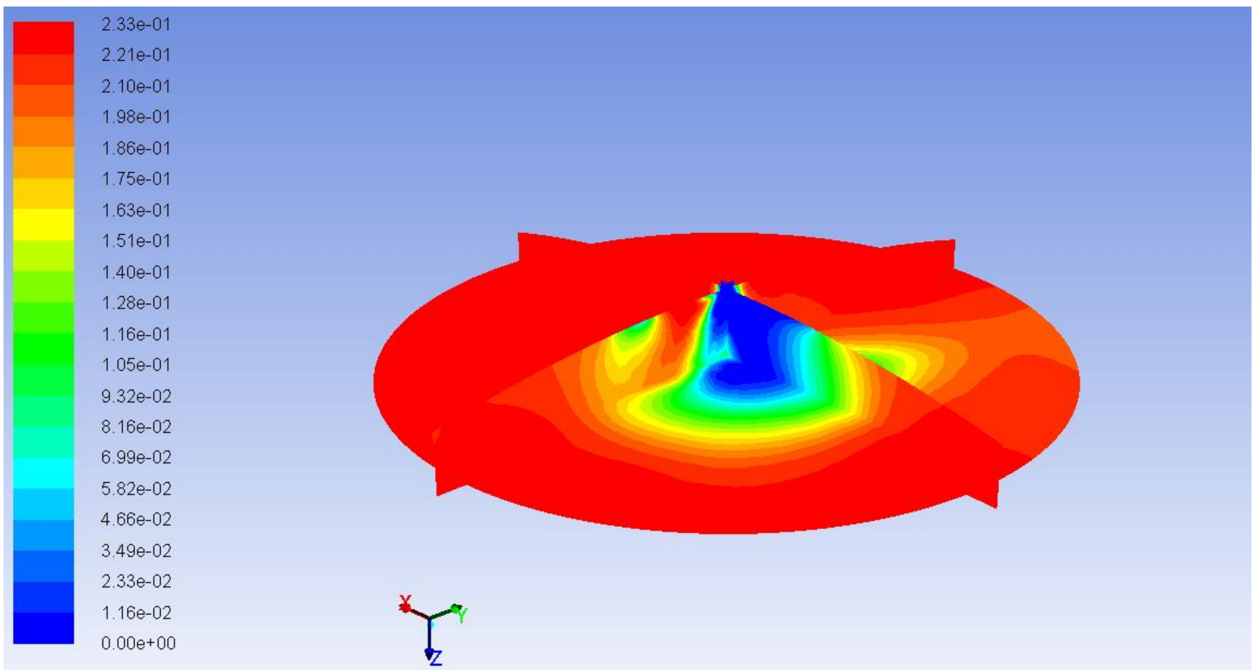

Figure 22: Oxygen Contour for Curved Cylinder Head

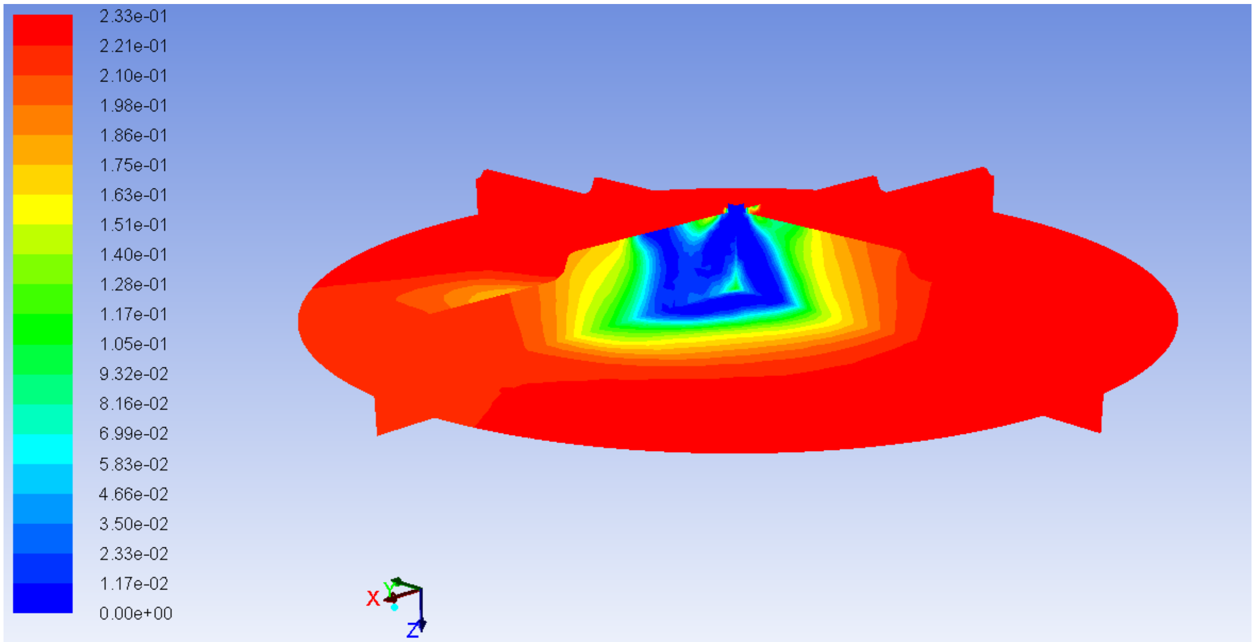

Figure 23: Oxygen Contour for Stepped Cylinder Head

The two contours above depict the mass fractions of oxygen in the cylinder geometry. Both contour plots are nearly the same, there is no major difference as such for the two different geometries. The mass fraction of oxygen near the region of fuel injection is nearly negligible. However, as we move away from the cylinder, the mass fraction of oxygen increases, thus, suggesting the spread of the flamelet through the combustion chamber.

6 CONCLUSION

Even with a minute change in the geometry of the engine cylinder head, the pressure inside the cylinder and the turbulence changes considerably even though temperature does not show much deviation. Thus, optimum combustion conditions can be used to model geometries of cylinders in order to increase the efficiency of the whole combustion process (Affes, 1998).

In case of CI engine, the positioning of the fuel injectors and the geometry of the cylinder plays an important role in deciding the results of combustion. Placing fuel injectors at angles and using multiple fuel injectors that spray the fuel to the cylinder at high pressure can cause the combustion to take more efficiently. However, using multiple fuel injectors also increases the chances of multiple flame fronts, which could result in excessive knocking. The injector could be placed in such a way that the temperature inside the cylinder reaches optimum combustion temperature, and the pressure inside the cylinder is well suited to prevent knocking characteristics in the engine, thus, prolonging life of the engine.

In case of SI engine, the positioning of the spark plug plays an important role besides the geometry of the cylinder. Similar to the CI engines, the spark may be initiated in such a way so as to get maximum output from the premixed air-fuel mixture inside the cylinder.

The scope of future experiments and simulations is vast in this particular field. Simulations can help understand the process of combustion inside SI as well as CI engines (Payri, 2003) without actually having a laboratory set up. In case of CI engine, the simulations may be expanded to using multiple injectors and cylinder heads with even complex designs, or have injectors placed at an angle to the direction of piston displacement so as to ensure better and thorough combustion of the fuel inside the combustion chamber. In SI engines, by defining the appropriate models, like the energy equation, k- ε model, and the proper species as in the case of CI engines, one can simulate the similar combustion study.

Moreover, unlike the steady case that is analyzed here, where particular moment is captured during the combustion process, the study can be extrapolated to transient cases. As assumed in the above study, the process is kept as a constant pressure (to include the effect of the piston) and constant volume (momentary combustion at TDC). This study can be made better if the piston movement during the whole process can also be simulated, so as to get a deeper insight into the actual combustion process in an engine. By simulating the piston movement, and by ensuring the appropriate time when the fuel in sprayed in the chamber (in case of CI engine), or the spark is initiated (in case of SI engine), a better combustion simulation result can be obtained.

7 TABLE OF FIGURES

8 REFERENCES

1. Affes, H., Trigul, N., Smith, D. & Griaznov, V. (1998) Shape Optimization of IC engine ports and chamber. *SAE Special Publication.*

2. Aglave, R., Reidel, U. & Warnatz, J. (2008) Turbulence- Chemistry interactions in CFD modelling of diesel engine. *Combustion Theory and Modelling,* 12, 303-323.

3. ANSYS (2006) ANSYS Documentation Notes.

4. Bakhshan, Y. & Mansouri, S.H. (2000) K-epsilon turbulence modelling of heat transfer and combustion processes in a Texaco controlled combustion stratified charge engine. *Proceedings of American Mathematical Society,* 128, 149-158.

5. Broatch, A., Margot, X., Gil, A. & Donayre, C. (2007) Computational study of sensitivity to ignition characteristics of the resonance in DI diesel engine combustion chambers. *International journal for Computer-Aided Engineering and Software,* 24.

6. Broeze, J. J. (1963) *Combustion in Piston Engines: Spark- Ignition and Compression- Ignition,* Keulen, De Technische Uitgeverij H. Stam N.V.

7. Cebeci, T., P. Shao, J., Kafyeke, F., Laurendeau, E. (2005) *Computational Fluid Dynamics for Engineers,* Springer.

8. Fletcher, C. A. J. (1991) *Computational techniques for fluid dynamics,* New York, Springer, Verlag.

9. Ganesan, V. (1994) *Internal Combustion Engines,* Mcgraw-Hill.

10. Golovitchev, V. I., Berman, M. & Montorsi, L. (2007) CFD modelling of diesel oil and DME performance in a two-stroke free piston engine. *Combustion Science and Technology,* 179, 417-436.

11. Ilbas, M. (2005) Effect of thermal radiation and radiation models on hydrogen- hydrocarbon combustion modelling. *International Journal of Hydrogen Energy,* 30, 1113-1126.

12. Knight, B. E. (1964) Problem of predicting heat transfer in diesel engines. *Proceedings of the Institution of Mechanical Engineers,* 179, 93-106.

13. Kong, S., Han, Z., and Reitz, R. (1995) The Development and Application of a Diesel Ignition and Combustion Model for Multidimensional Engine Simulation. *SAE Technical Paper.*

14. Kong, S.-C, & Reitz, R.D. (2003) Numerical study of premixed HCCI engine combustion and its sensitivity to computational mesh and model uncertainties. *Combustion Theory and Modelling,* 7, 417, 433.

15. Kurniawan, W. H. & Abdullah, S. (2007a) Numerical Analysis of the combustion process in a four stroke compressed natural gas engine with direct injection system. *The 9th Asian International Conference on Fluid Machinery.*

16. Nishida, K. and Hiroyasu, H. (1989) Simplified Three-Dimensional Modeling of Mixture Formation and Combustion in a D.I. Diesel Engine. *SAE Technical Paper.*

17. Palipana, A. S., Malalasekara, W. & James, E. H. (1999) A CFD parametric study of natural gas engine combustion. *1999 Spring Technical Conference,* ASME.

18. Payri, F., Benajes, J. Margot, X. & Gil, A. (2003) CFD Modelling of the in-cylinder flow in direct-injection Diesel engines. *Computer and Fluids,* 33.

19. Versteeg, H.K. & Malalasekera, W. (1995), *An Introduction to Computational Fluid Dynamics,* Pearson Education.

20. Wikipedia, *en.wikipedia.org*