CFD analysis of combustion chamber of SI engine during suction stroke

Thwahir M. P. thwahirntm@gmail.com
Malabar College of Engineering and Technology, Thrissur, Kerala

Shibu Augustin shibuta2002@gmail.com
Malabar College of Engineering and Technology, Thrissur, Kerala

ABSTRACT

This work is suitable for the IC engine design and analysis industry in understanding the various performance characteristics of the flat and dome head piston. The design of the piston and chamber has a vital role in the optimized performance of an engine. There are lots of studies undergone about the new piston geometries and designs to achieve the optimized results. Studies about the piston and combustion process include both the structural and thermal analysis because both pressure and temperature are the two important properties which affect the performance of the piston and chamber. This work designs a new model of flat and dome piston for an assumed optimized output of the existing engine and chamber. This work will be helpful for the new inventions in engine development. In this work piston and chamber are modelled in CATIA v5 software and these models are imported into ANSYS software and analysis is employed. Structural and thermal analysis of the piston and air flow analysis of the chamber is employed for both flat and domed piston.

Keywords— Optimized intake stroke, CFD Analysis, Flat and domed piston, IC Engine

1. INTRODUCTION

A Spark-Ignition Engine (SI engine) is an internal combustion engine generally a petrol engine, where the combustion process of the air-fuel mixture is ignited by a spark from a spark plug. This is in contrast to compression-ignition engines, typically diesel engines, where the heat generated from compression together with the injection of fuel is enough to initiate the combustion process, without needing any external spark.

Advancement in internal combustion engine design has taken new dimensions in optimization methods of improving engine performance. Over the year’s simulation of optimization, characteristics have followed dynamic changes in conducting laboratory experiments. In recent times Computational Fluid Dynamics (CFD) analysis has been the forefront of advanced research techniques in determining air-fuel flow profile suitable for internal combustion engines. This involves the use of computer algorithms to model and simulate flow characteristics which literally predict the occurrences in the combustion chamber that are normally difficult to witness in operation. CFD optimization methods are designed for aerodynamics problems and computing this experimentally normally takes several hours or even days to derive results. Computational time could be expensive and difficulties could arise as a result of inaccurate modelling attributes. CFD codes are generated on adjoint solvers and concatenation of models are computationally expensive in terms of CPU effort in geometrical processing. Dealing with multi-objective optimization is of great importance in the design of internal combustion engines.

Turbulence modelling is described as the function of using a model to analyse turbulence. Turbulence is the spontaneous and erratic behaviour of fluids where indiscriminate changes occur in fluid parameters within a specified boundary.

Turbulence modelling has been one of the most difficult to analyze flow problems in the engineering field. There are numerous turbulence models developed over the centuries to describe the turbulent flow. Navier-Stokes (NS) equations are one of the common and unsolved turbulence models in the world. The closure problem arising from many unknown terms in the equation has been unsolved for many years. Reynolds Averaged Navier-Stokes (RANS) adopts the principle of time- averaging to simplify Navier-Stokes equation. Among the continuum categories of turbulence models are zero- equation models, one-equation models, two-equation models, however, the most common and simplified models include: K-epsilon (k-e) model K- omega (k-w) model Spallart-Allmaras Reynolds stress equation model.

In understanding the principle of operation for conducting optimization regarding the combustion chamber it requires the knowledge of turbulent flow characteristics and parameters. Equations have been developed to assign variables in CFD experimentation using one of the common turbulence models. An important and unfortunate aspect of CFD research is the limitations of a complete description of all characteristics occurring in the combustion chamber but it is also important to note the rapid change in the effects of simulation parameters. The problem of heat interaction with the fluid has been considered in this experiment also to include the heat transfer between the cylinder walls and the thermo-fluid. Engine temperature adopted for CFD experiment is based on assumptions.
2. COMBUSTION CHAMBER ANALYSIS

ANSYS is industrially recognized software used to model flow problems. It is used to simulate fluids such as viscous fluids, gases, solids also the conditions such as boundary conditions, physics conditions etc. High-speed computers are used to perform calculations which usually generate reliable solutions. In this report, the aim virtually stresses the research into the modelling of turbulent flows. Though, more time is dedicated to the geometrical processing of the problem including meshing as well as running the simulation. ANSYS is programmed with turbulence models which will be used to analyze the flow of air at the intake stroke. Temperature phenomenon is accustomed to internal combustion engines, Heat flux generated from the combustion process and surrounding heat have an effect on gases. This analysis includes the effect of temperature variables at the intake.

2.1 Governing equations

Mass continuity equation for incompressible
\[ \nabla \cdot \mathbf{U} = 0 \]  

Starting with the equations of motion, the Navier- Stokes equation is given as:
\[ \rho \frac{d\mathbf{U}}{dt} + \rho \mathbf{U} \cdot \nabla \mathbf{U} = -\nabla p + \mathbf{F} + \eta \nabla^2 \mathbf{U} \]  

Air enters the combustion chamber at high speeds and is sometimes supersonic in nature therefore, it is important to present assumed values for the inlet velocity. The turbulent model adopted for performing the analysis is k-epsilon (k-ε) model. Before using the k-ε model it is important to state the variable parameters for analysis.

2.2 Assumptions

Inlet velocity (u) = 15m/s
Engine temperature (TE) = 1273K
Combustion temperature (TC) = 2773K
Pressure (P atm) = 101210pa
Density (ρ) = 1.205kgm⁻³
Kinematic viscosity (ν) = 1.544x10⁻⁵m²s⁻¹
Dynamic viscosity (μ) is given by
\[ μ = \nu \rho = 1.8605x10^{-5} \text{kgm}^{-1}\text{s}^{-1} \]  

Specific heat (H0) = 1007JKg⁻¹K⁻¹
The diameter of the intake valve (L) = 0.04m

Reynolds Number:
\[ (Re) = \frac{uL}{v} = 38860.10363 \]  

Length scale;
\[ (l) = 0.07xL = 2.8x10^{-3} \text{m} \]  

Turbulence Intensity
\[ (I) = 1.06(Re)^{0.125} = 0.04276958 \]  

Turbulence Kinetic Energy
\[ K = 2/3(Iu) = 20.61736747 \text{J/kg} \]  

Turbulence Dissipation Rate:
\[ (Cμ)^{0.75}l^{1.5} = 28.468929 \text{m}^2/\text{s}^3 \]

2.3 Modelling of geometry

The existing of the combustion chamber is modelled using CATIA software as per the following dimensions.

Bore length = 52.4mm Stroke length = 57.9 mm

2.4 Meshing

Volumetric meshing technique is adopted while a suitable mesh type for simulation is polyhedral mesh. Polyhedral mesh presents good volume mesh and mesh concentration is used at the relative boundaries. The base size of 1 mm is used for mesh size.

2.5 Physical boundary conditions

It is important to select the appropriate physics condition by specifying the turbulence model and the values of the state variable. The options selected for the physics model are as follows;
- “Three dimensional” The model was developed in three dimensions.
- “Steady” The air-fuel mixture has a steady flow characteristics.
- “Gas” The mixture is in a gaseous state.
- “Constant density” The mixture’s density does not change with time.
- “Segregated flow” Intake stroke is characterized by a segregated flow phase index.
- “Turbulent” Intake air-fuel mixture is turbulent with a supersonic flow rate.
- “K-Epsilon” The K-ε turbulence modelling is selected for the analysis.
- “Segregated vortex temperature” The temperature of the analysis is considered. The boundary conditions used to simulate the flow include;
  - “Velocity inlet” for the intake port
  - “Wall” which was assumed the exhaust valve is closed with temperature 2773K.
  - “Wall” for the cylinder walls and piston head with a temperature of 1273K.
  - “No-slip condition” for the Fluid-Wall contact.

2.6 Post processing

Post processing: Extraction of results after the computation is completed. This is a visual display of variable functions about the geometry. Various parameters distribution for both flat and domed piston is shown below:
Fig. 3: Dynamic pressure distribution

Fig. 4: Total pressure distribution

Fig. 5: Temperature distribution

Fig. 6: Velocity distribution

Fig. 7: Viscosity distribution

Fig. 8: Kinetic energy distribution
2.7 Comparison of flat and domed pistons

<table>
<thead>
<tr>
<th>S no.</th>
<th>Properties</th>
<th>Flat piston</th>
<th>Dome piston</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Static pressure(Pa)</td>
<td>1.16x10^49</td>
<td>1.27x10^49</td>
</tr>
<tr>
<td>2</td>
<td>Dynamic pressure(Pa)</td>
<td>1.48x10^4</td>
<td>4.9x10^8</td>
</tr>
<tr>
<td>3</td>
<td>Total pressure(Pa)</td>
<td>1.41x10^4</td>
<td>1.4x10^4</td>
</tr>
<tr>
<td>4</td>
<td>Total temperature(K)</td>
<td>1.27x10^3</td>
<td>1.17x10^3</td>
</tr>
<tr>
<td>5</td>
<td>kinetic energy(m2/s2)</td>
<td>8.63x10^4</td>
<td>3.01x10^4</td>
</tr>
<tr>
<td>6</td>
<td>Viscosity(Kg/m-s)</td>
<td>2.03x10^4</td>
<td>1.2x10^4</td>
</tr>
<tr>
<td>7</td>
<td>Velocity(m/s)</td>
<td>4.94x10^4</td>
<td>2.83x10^4</td>
</tr>
</tbody>
</table>

Heat transfer is very important in piston design and the process of combustion is mainly to generate heat.

Heat variation across the cylinder walls and the effect it has on gases within the combustion chamber are necessary to optimise the cooling rate of the cylinders. Comparison between the flat head and dome head pistons with temperature effects as displayed in table 1 shows that the flat head piston developed a slightly higher pressure than the dome head piston. The total temperature and velocity indicate that flat head piston is at a faster rate than in dome head type. Air-fuel mixture velocity in the flat head piston model is very higher than in dome type which is significant to the effect of surface-to-volume ratio. The dome head obviously has more surface area than that of flat type. Thus combustion chamber design is better optimized with a flat type than a dome type piston.

3. DESIGN OF NEW PISTON

After the comparison of both the piston, we have to design a new piston with optimized output value suitable for the above combustion chamber.

3.1 Values for calculation

- Factor of safety=2.25
- Thermal conductivity k=174.15(W/m°C)
- Higher calorific value of fuel HCV=47000 HS/kg (petrol).
- Brake power of engine per cylinder=4kw (by considering N=1500rpm, compression ratio 16.5)
- M=mass of fuel used for brake power per second=0.25/3600kg
- Pw=maximum allowable pressure on the cylinder wall=0.042m

3.2 Calculations

**Thickness of piston head (th)**

\[ th = D/3P/16 \text{ et} = 5D/3x8/16x124 = 5.49mm = 5.5 \text{ mm} \]

**Heat flows through piston head (H)**

\[ H = CxHCVmxBP = 0.05x47000x0.25x36000x4 = 0.6527 \text{ KJ/s}. \]

On the basis of heat dissipation, the thickness of piston is given by

\[ th = H/12.56xK(Tc-TE) = 0.6527/12.56x174.15x75 = 3.9 = 4 \text{ mm} \]

Where K=174.15 W/m°C for aluminium alloy Tc-TE=75 for aluminium alloy.

**The radial thickness of the piston ring (t1)**

\[ t1 = D/3Pw/\sigma \]

\[ = 50\sqrt{3x0.042}/124.4 = 1.6 mm = 2 mm \]

Where, value of P ranges from 0.025N/mm2 TO 0.042 N/mm2 \( \sigma = 124.4 \text{ mpa for aluminium alloy}. \)

**Axial thickness (t2)**

Value of t2 varies from 0.7t1 to 1t1 0.8t1=0.8x2 = 1.6mm

**Number of rings (nr)**

\[ t2 = D/(10nr) \]

1.6=50/10nr nr=50/16 = 3.12≈3nos.

**The maximum thickness of the barrel at the top end (t3)**

Radial depth of piston ring grooves is 0.4mm more than the thickness of piston rings.

\[ b = 0.4 + t1 = 0.4 + 2 = 2.4 \text{ mm} \]

**Width of top land (b1)**

Width of top land varies from 1 to 1.2th=1.1th=1.1x5.5=6.05mm

**Width of other ring lands (b2)**

Width of other ring lands varies from 0.75t2 to t2 =0.85x1.6=1.36mm

**Thickness of piston barrel at open end (t4)**

\[ t4 = 0.25 \text{ to } 0.35t3 = 0.3xt3 = 0.3x8.4=2.52 \text{ mm} \]

**Piston skirt**

Length of piston skirt is taken as0.65 to 0.8 times the cylinder bore

Length of skirt=0.75x50 =37.5mm

**Total length of piston (L)**

\[ L = \text{length of skirt}+\text{length of ring section}+\text{top land} = L+(3t2+2b2)+b1 = 37.5+4.8+2.72+6.05 = 51.07 \text{ mm} \]

**Location of piston pin**

Location of piston pin varies from 0.2 to 0.4D times above the centre of the skirt.

**Piston pin diameter =0.3D=1.5mm**

3.3 Piston geometry

Design specification before optimization Length of the Piston (L) = 51.07mm Cylinder bore/ outside diameter of the piston (D) = 50 mm The thickness of the piston head (H) = 4 mm The radial thickness of the ring (t1) = 2 mm Axial thickness of the ring (t2) = 1.6 mm Width of the top land (b1) = 6.05 mm Width of other ring lands (b2) = 1.36 mm

Similar to combustion chamber geometry the piston is modelled by using CATIA software.
4. RESULTS
Both structural and thermal analysis is carried out on the new piston.

4.1 Structural analysis

4.2 Thermal analysis

5. CONCLUSION
Analyzed the combustion chamber of the existing engine during suction stroke and identified the various parameters during the stroke for both flat and domed piston. Comparison is done between the flat and domed piston on the basis of the stimulation result. A new piston is designed for optimum output value suitable this combustion chamber and both the structural and thermal analysis is employed.

It can be deduced that individually, thermal and mechanical stress proportions have a direct influence on the coupled thermal-mechanical stress hence during design each load can be considered and reduced independently. It can be concluded that the piston can safely withstand the induced stresses during its operation.

6. REFERENCES