CFD Studies of Combustion in DI Diesel Engine at Different Injection Timing and Ethanol Addition

1Maheboob Pasha, 2Yeshwanthray ashtagi, 3Rahul G Panchal

1. Student M.Tech Department of Thermal Power Engineering, V.T.U. PG Centre RO, Gulbarga, India
2. Assistant Professor Department of Thermal Power Engineering, V.T.U. PG Centre RO, Gulbarga, India
3. Assistant Professor Department of Mechanical Engineering Al Ameen COE and MS Pune Maharashtra, India.

ABSTRACT

In recent years, due to increasing demand for fuel economy and tightening legislation for emissions, there has been growing requirement to develop more efficient and cleaner engines in a shorter time scale. Computational Fluid Dynamics, as a rapid and cost effective tool, is being increasingly used in different stages of engine design and optimization. By using CFD tools effectively it is easy to predict and analyse various details that are technically difficult like in cylinder process of diesel combustion, temperature & pressure distribution and emissions etc. prior to experimental tests to reduce the number of investigated parameters as well as time and thus costs. Here in our approach we have used Species Transport Model of ANSYS FLUENT14.5 to find the complex phenomenon of in cylinder process of combustion, temperature and pressure distribution, CO₂ and NOₓ emissions etc. The process is carried out at different injection timing of the Diesel liquid, the cases considered are 1. Injection at 2 degree after TDC, 2. Injection at 5 Degree bTDC, 3. Injection at 10 Degree bTDC. Once we get an optimized value of Injection than 10 % of ethanol is added in order to increase the efficiency of the combustion and thus still reduce the NOₓ emissions.

Keywords- Diesel combustion FLUENT 14.5, NOₓ, CFD, CO₂ emission, Combustion modelling, k-ε model, Ethanol.

I. INTRODUCTION-The Internal Combustion (IC) engines play an important role in the fields of transportation of goods and passengers, agricultural and industry. They develop power by consuming precious fossil fuels and cause pollution. Among different types of engines, the direct-injection (DI) diesel engine exhibits the best fuel economy along with lowest engine-out emissions. Efforts have been put to improve exhaust emissions and fuel economy continuously. The complex task of improving IC engines, which have reached a higher degree of sophistication, can be achieved by combination of advanced experiments and computational studies. Modern methods of experimental investigations are being developed to provide more insight. The modelling of combustion engine processes is useful to carry out extensive parametric studies, rather than hardware development and Experimentation. Depending on the various possible applications, different types of models for engine combustion processes have been developed. Rapid Increase in pollution levels, increase of fuel prices, and depletion of hydrocarbon reserves of the world have forced the engineers to look for appropriate technology and alternative fuels to provide to the ever-increasing demands of energy.[1]

II.COMPUTATIONAL PROCEDURE

The combustion simulation of CI engine was developed using fluent software (ANSYS 14.5 package) and the various equations of the multi-dimensional model were solved by the software automatically. The main inputs include engine speed, injection details, bore, stroke, connecting rod length, initial pressure and temperature. The program concerning the simulation model predicts the cylinder pressure, cylinder temperature, heat release rate, emission etc. The results including graphs and various contours (temperature, pressure etc) were generated by fluent software as outputs to the program for given inputs.[2],[3].

III.MODEL DEFINITION AND MESHING

A 2D cylinder geometry with centrally located injector was considered .The mesh was created using GAMBIT 2.4.6. The engine geometry details and specification details are given below. Mesh generation plays an important role in obtaining accurate results. A quadrilateral mesh was created uniformly throughout the area and analysed using FLUENT, ANSYS 14.5 package. The complete meshed geometry contains 821098 Elements and 145400 nodes. Fig.1 shows the meshed geometry.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Connecting rod length</td>
<td>10.35 inch</td>
</tr>
<tr>
<td>Bore</td>
<td>130mm</td>
</tr>
<tr>
<td>Stroke</td>
<td>158mm</td>
</tr>
<tr>
<td>Crank radius</td>
<td>55 mm</td>
</tr>
<tr>
<td>Crank shaft speed</td>
<td>1500 rpm</td>
</tr>
</tbody>
</table>
IV. GOVERNING EQUATIONS AND MODEL

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier-Stokes equation) and Energy equation. The flow of most fluids may be analyzed mathematically by the use of two equations. The first, often referred to as the Continuity Equation, requires that the mass of fluid entering a fixed control volume either leaves that volume or accumulates within it. It is thus a "mass balance" requirement posed in mathematical form, and is a scalar equation. The other governing equation is the Momentum Equation, or Navier-Stokes Equation, and may be thought of a "momentum balance". The Navier-Stokes equations are vector equations, meaning that there is a separate equation for each of the coordinate directions (usually three). The basic approach of in-cylinder diesel combustion models are typically compressible turbulent flow. Apart from the complexity of turbulent model, the high pressure spray and resulting spray penetration, evaporation, and involvement of multiphase, multi-component nature only increases its complexity. Even then the nature of fluid is still governed by the basic equations including continuity (mass conservation), momentum (Navier-Stokes equation), energy and turbulence (k-ε model) equations. Of the three combustion modelling (thermodynamic, multidimensional and phenomenological modelling) the software use multidimensional modelling, i.e. CFD modelling.

V. COMBUSTION SIMULATION

In engine environments, the combustible mixture is subject to a turbulent flow and, once mixed, undergoes subsequent elementary reactions which convert the fuel vapour to complete and incomplete combustion products with the accompanying release heat. So, in diesel engine simulations, combustion modelling mainly deals with two processes: first, low-temperature chemistry, which leads to auto ignition and produces intermediate species, and second, these intermediate species trigger high-temperature reactions that contribute the main heat release, as well as further complete and incomplete combustion products. Another important task for combustion models is to properly account for the significance of the effect of turbulence on the combustion processes. The combustion of diesel fuel was considered which is simulated using FLUENT, ANSYS 14.5 package. A single cylinder, single zone, multi-dimensional model was considered in analysis process. The mixture material, diesel-air, properties were copied from the Fluent material database for combustion. Using species transport model of fluent the analysis is done. While using this model conservation equation for chemical species, ANSYS FLUENT predicts the local mass fraction of each species through the solution of convection diffusion equation for the ith species.[4].

VI. METHODOLOGY & BOUNDARY CONDITIONS

After modelling the geometry, for analysis viscous standard k-ε model is enabled for considering eddy dissipation. Since geometry is subjected to motion of piston, dynamic meshing is enabled subjected to suitable boundary condition for piston, cylinder, walls etc. Combustion in a diesel engine involves the transient injection of finely atomized liquid fuel into the air at high temperature and pressure. The fuel injection parameters like location of the injector, size of the injector, injection temperature and pressure, mass flow rate etc are having significant effect in diesel combustion modelling. The injection parameters and specifications are given below:

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>X-Position</td>
</tr>
<tr>
<td>2</td>
<td>Y-Position</td>
</tr>
<tr>
<td>3</td>
<td>Z-Position</td>
</tr>
<tr>
<td>4</td>
<td>Diameter</td>
</tr>
<tr>
<td>5</td>
<td>Temperature</td>
</tr>
<tr>
<td>6</td>
<td>Injection Type</td>
</tr>
<tr>
<td>7</td>
<td>Velocity Magnitude</td>
</tr>
<tr>
<td>8</td>
<td>Cone Angle</td>
</tr>
<tr>
<td>9</td>
<td>Total Flow Rate</td>
</tr>
<tr>
<td>10</td>
<td>Particle Type</td>
</tr>
<tr>
<td>11</td>
<td>Starting Crank Angle</td>
</tr>
<tr>
<td>12</td>
<td>Starting Crank Angle</td>
</tr>
</tbody>
</table>

VII. RESULTS AND DISCUSSIONS

The below section shows the results of the cases considered in this project.
CASE 1: INJECTION OF FUEL AT 0 DEGREE BEFORE TDC.

The above figure shows variation contours of start of fuel injection & start of combustion for 2D cylinder.

1. Cylinder pressure and Cylinder temperature

The modelled cylinder pressure data is shown in fig.4. From the graph the peak pressure generated by the software was approximately equal to 20 bar which is in agreement with the theoretical value. Also the maximum cycle temperature obtained was near 2400K (Fig.5). The static pressure and static temperature contours at the end of expansion was also generated by the software (shown in fig.4. and fig.5).

2. CO\textsubscript{2} Emission

The mass fraction of CO\textsubscript{2} was generated by the simulating software and was found to be near 0.0800 at the end of expansion. (fig.6)

3. NOx Emission

The mass fraction of NOX was generated by the simulating software and was found to be near 0.0160 at the end of expansion. (fig.7)

Case 2: Injection 5 Degree before TDC

The Results for Injection of Diesel Liquid 5 Degree before TDC are shown below
1. Cylinder pressure and Cylinder temperature
The peak pressure generated by the software was approximately equal to 19 bar which is in agreement with the theoretical value. Also the maximum cycle temperature obtained was near 2400K (Fig.11). The static pressure and static temperature contours at the end of expansion was also generated by the software (shown in fig.10 and fig.11). [7]

2. CO₂ Emission
The mass fraction of CO₂ was generated by the simulating software and was found to be near 0.0800 at the end of expansion

3. NOx Emission
The mass fraction of NOx was generated by the simulating software and was found to be near 0.0140 at the end of expansion

CASE 3: Injection of Diesel Liquid 10 Degree before TDC
In the Third Case the Injection was performed 10 Degree before TDC and the Results are Shown in the Coming Slides
1. Cylinder pressure and Cylinder temperature

The modelled cylinder pressure data is shown in fig.16. From the graph the peak pressure approximately equal to 19 bar which is in agreement with the theoretical value. Also the maximum cycle temperature obtained was near 2400K (Fig.17). The static pressure and static temperature contours at the end of expansion was also generated by the software (shown in fig.16. and fig.17).

Fig-16: Variation of Pressure V/S flow time.

Fig-17: Variation of Temperature V/S flow time.

2. CO₂ Emission

The mass fraction of CO₂ was generated by the simulating software and was found to be near 0.0750 at the end of expansion

Fig-18: Variation of CO₂ V/S flow time

3. NOx Emission

The mass fraction of NOx was generated by the simulating software and was found to be near 0.0120 at the end of expansion

Fig-19: Variation of NOx V/S flow time.

Case 4: Addition of Ethanol, to diesel liquid.

1. In the case 4 we have used the injection timing as per our previous analysis results in which we found that the injection with 10 degree before TDC is giving better results.

2. In this case we will add 10% of ethanol to further reduce the NOx emission

1. CO₂ Emission

The mass fraction of CO₂ was generated by the simulating software and was found to be near 0.1400 at the end of expansion

Case 4: Additon of Ethanol to diesel liquid.

1. In the case 4 we have used the injection timing as per our previous analysis results in which we found that the injection with 10 degree before TDC is giving better results.

2. In this case we will add 10% of ethanol to further reduce the NOx emission

Fig-20: contoures start of combustion

Fig-21: contours End of combustion
2. NOx Emission
The mass fraction of NOx was generated by the simulating software and was found to be near 0.0110 at the end of expansion.

Table 1: Comparison of Results with Base Data and Different Injection Timings

<table>
<thead>
<tr>
<th>Case No</th>
<th>Injection Timings</th>
<th>Max P(bar)</th>
<th>Max T(k)</th>
<th>CO2</th>
<th>NOx</th>
</tr>
</thead>
<tbody>
<tr>
<td>Base</td>
<td>0° b TDC</td>
<td>200 * 10^-5</td>
<td>2500</td>
<td>0.0754</td>
<td>...</td>
</tr>
<tr>
<td>Case 1</td>
<td>0° b TDC</td>
<td>200 * 10^-5</td>
<td>2500</td>
<td>0.076</td>
<td>0.0160</td>
</tr>
<tr>
<td>Case 2</td>
<td>5° b TDC</td>
<td>190 * 10^-5</td>
<td>2400</td>
<td>0.078</td>
<td>0.0140</td>
</tr>
<tr>
<td>Case 3</td>
<td>10° b TDC</td>
<td>190 * 10^-5</td>
<td>2400</td>
<td>0.0780</td>
<td>0.0140</td>
</tr>
<tr>
<td>Case 4</td>
<td>10° b TDC</td>
<td>160 * 10^-5</td>
<td>1800</td>
<td>0.140</td>
<td>0.0110</td>
</tr>
</tbody>
</table>

IX. CONCLUSION
The model was created using Gambit, and the combustion phenomenon was analysed using Fluent Ansys 14.5. Based on the present investigation the following conclusions have been drawn:

- From the above results we can conclude that the maximum reduction in NOx occurs when the fuel is injected at least 10 degree before TDC. NOx reduces when the peak temperature reduces. Maximum Pressure i.e. 200*10^-5 is obtained in the case of injection at 0 Degree before TDC, this high pressure may be harmful since it may cause high stresses inside the cylinder. From many experimental works it has been found that as the injection timing is reduced before TDC better combustion takes place and NOx reduces, so from our analysis we found that the ANSYS Fluent 14.5 was helpful in giving good results for the combustion modelling. The addition of Ethanol has been added only upto 10% and it has been found that the addition of ethanol has reduced further the NOx emission. So in future if India implemented ethanol blending with diesel and petrol the blending ratio of 10% of ethanol will be considered to be more effective than the neat diesel in every aspect. It has also been considered according to the production, demand and supply of the ethanol in India.

REFERENCES


