

# CFD Simulation of In Cylinder Gases of Multi-cylinder Diesel Engine for Estimation of Liner Temperature from Gas Side

N. B. Totla, C. L. Prabhune, N. K. Sane

**Abstract:** The simulation /estimation of cylinder temperature for power stroke of internal combustion engine along with liner temperature from gas side is pertinent/significant/essential to investigate the thermal distortion of a liner at different points along its longitudinal direction from coolant side. The present research/study includes estimation/finding of approximate temperature of in cylinder gases during power stroke and thereby liner temperature from gas side using water, and ethylene glycol as a coolant for present diesel engine. The present study includes numerical simulations based on Inbuilt ICE Combustion model in ANSYS 15.0 version where dynamic/time transient

meshing of combustion space above piston during power stroke is used. Appropriate averaged boundary conditions were set on different surfaces for the combustion model. The variation of temperature of cylinder combustion gases and temperature of liner along gas side at different crank angle is reported. Observations are done that the highest combustion gas temperature occurred during power stroke was about 2150 K and minimum temperature is found to be 800 K. Also the maximum temperature on liner from gas side along stroke was found to be 470K during power stroke. It has been also found that the maximum temperature of in cylinder gases and liner from gas side persists only during early power stroke

**Index Terms:** Combustion, liner, grid, dynamic mesh, simulation

## I. INTRODUCTION

Internal combustion engines are the preeminent prime mover/consistently good source of energy for all kinds of locomotive and industrial applications. Combustion research is wide ranging because of potent carving/sculpting method like CFD. In compression ignition (CI) the combustion chamber polyphase liquid dynamics parameters like diesel injection, kinetics of chemical reaction, creates an impact on the combustion process. Previously, the burning of fuel and burning growth/development had been carved/sculpted with many distinct models viz. the eddy dissipation model and its descendants, supplement of the coherent flame model like PDF time scale models, and the RIF model. Recent researches related to the evolution /progress of new and reliable models for burning of fuel process has been filed in the literature

## II. LITERATURE REVIEW:

For this entire research work following literature is reviewed  
**Carmen C. Barrios et al.** [1] explored the use of Computational Fluid dynamics (CFD) code FLUENT for shaping convoluted combustion process in diesel engine. On one-cylinder diesel engine, tests were performed at a steady speed of 1500 rpm on full load. Here Combustion process conditions such as pressure rise, combustion pressure and rate of heat liberation were estimated through experimentation. The FLUENT is also utilized to replicate fuel burning process. They reported that the experimental values were in good agreement with the predicted values of CFD simulations.

**F. Christodoulou and A. Megaritis et al.** [2] reported consequences of concurrent/coincident  $H_2 + N_2$  admission charge enhancement over the diesel engine emission and burning process. Here study of admission of conserved  $H_2 + N_2$  concurrently in the inlet pipe of the engine in 4% increments commencing from 4% till 16% (v/v) is done. Authors concluded that beneath performing/running particular conditions  $NO_x$ , BSN and CO emissions reduction are achieved by  $H_2 + N_2$  enrichment. Here other than controlled emissions, nitrogen emission ingredients are also calculated and shown to be negligible.

**W. B. Santoso et al.** [3] studied the combustion properties of a hydrogen fueled one CI engine. Prior the admission of the fuel in the combustion chamber, the hydrogen entered in the intake manifold with the help of mixer.

**C. Pana and C. Nutuet et al.** [4] recorded the improvement of engine efficiency and pollutant performance for truck diesel engine by using hydrogen as fuel. Results showed the refinement of the

**Revised Manuscript Received on May 22, 2019.**

**N.B.Totla**, Assistant Professor, School of Mechanical and Civil Engineering, MIT Academy of Engineering, Alandi, Pune, Maharashtra, India.

**C.L.Prabhune**, Professor, Department of Mechanical Engg, Zeal College of Engineering and Research, Narhe, Pune.

**Dr.N.K.Sane**, Adjunct Professor, Walchand College of Engineering, Sangli.



combustion process and reduction of carbon quantity in the emissions.

**A.P.Singh and A.K.Agarwal** [6] provided an empirical study for double cylinder engine, where first cylinder is altered to work with Homogeneous charge compression ignition form while other works in ordinary CI form. They showed that superior emission characteristics are seen in an HCCI Engine as compared to standard CI combustion. HC and CO exhausts are somewhat more as opposed to typical classic combustion. EGR regulates the combustion rate positively and automatically and boosts exhaust radiation nature at the expense of marginally lesser attainment.

**U.V. Kongre and V.K.Sunnapwar** [7] reported numerical analysis for combustion process in diesel engine. Tests were performed on one cylinder and direct injection diesel engine, at 1500 rpm under full load. The authors deduced that shaping of combustion process can rely on CFD modeling as a better tool.

**A.Siddique, S.Abdul, A.Raffi Mohammed et al.** [9] analyzed the consequences of engine configuration in concern with following emissions such as CO, CO<sub>2</sub>, NO<sub>x</sub>, HC and smoke density. The authors analyzed the distinction of the changed configuration of MINI-PETER diesel engine to the standard statistics and deduced that the turbulence results are also improved in it.

**R.Bisane and D. Katpatal** [10] studied the energy rich exhaust system advancement which utilizes minimum fuel with maximum usage of emission energy for decrease of the emission discharges with systematic use of recuperation of its energy systems such as in turbocharger, heat pipe for diesel engine.

**H.Sushma, and Jagadeesha.K.B.** [11] investigated the consequences of various piston configurations on movement of air and agitation /instability in the combustion space of a Direct Injection (DI) diesel engine using CFD code Fluent.

**Z.F. Tian and J. Abraham** [13] adhered that CFD can be utilized as a reliable means in diesel engine coaching and training. The numerical data for velocity fields, temperature, species concentration and pressure profiles with respect to gas and coolant side were noted.

In the present and depicted literature, the research had not been seen in the liner temperature estimation from gas side in multicylinder diesel engine. The literature is complying the CFD carried out for in cylinder combustion, but the estimation of the liner temperature from gas side is missing. Here the same gap is identified and taken for research/study.

### III.METHODOLOGY ADOPTED

The module selected for simulation of combustion process is inculcated module /incorporated module in ANSYS 15.0. i.e;ICE Module. Shaping of power stroke during the engine cycle, starting from shutting of valves to the termination of the compression stroke are steps of combustion simulation. Since the valves are shut in the process of closing, the in cylinder is the significant flow domain, and the piston is the individual moving part. These enactments are termed as "in-cylinder combustion" and though multi-dimensional, are less complicated in configuration than a port flow simulation. In addition, if the geometry is symmetric rotationally/spin similar and has a

single feature like a very high pressure spray that dominates the flow in the calculation, the entire area can be shaped as a sector to accelerate the estimation

Typically, the starting flow field at this stage is deduced from

- If the full geometry is utilized, then its cold flow enactment
- Packing -in based on aabove
- Movement the piston without combustion to get charged compression

As along cold flow, a propelling contorting mesh model is utilized for the piston movement. Geometric decomposition is not must/required here, as only the piston movement is included/taken in the simulation. Hence, in-cylinder combustion simulations typically do not include the modeling of the fluid dynamics in the valve port area and their result on combustion.

Various Models are used to account for the fuel spray, toxic waste generation, and combustion. For direct injection engines, the fuel drizzle from the tip of the nozzle injector is introduced at the fix crank angle and time using a spray exemplary. For port fueled engines, it is considered that the combustion charge is well mixed, uniform and homogeneuous. A chemical mechanism narrating the reaction of vapor fuel with air is used to detailise the combustion, and models for turbulence-chemistry synergy are specified. Sub-models for NO<sub>x</sub> and soot formation are taken into consideration to estimate pollutant formation, which can be conjoined with the combustion calculation or calculated as a post processing step. Within cylinder combustion, the main challenge comes when we deal with physics for spray modeling and combustion. The spray is composed of a column of liquid getting in the domain at great speed which eventually split into droplets due to aerodynamic forces. These droplets break into smaller droplets or can even combine into larger droplets, all while exchanging mass with the surrounding gases. Sub-models are used for estimation of coalescence and breakup, heat and mass transfer computation, for capturing spray dynamics. The CFD mesh has to be adequately resolved to seize the connection among/admist the liquid droplets and the gases in the cylinder exactly. If fuel splash hits on the cylinder walls, it may form a fragile liquid film which undergoes its own processes of motion and vaporization and requires a separate processing.

The analysis/ study of combustion process, includes study of accurate chemical mechanisms for pure fuels that compose/integrate the segments of diesel involving number of species and counterberations. These reactions are linked with the fluid dynamics as it is having similar time scales of liquid mechanical movements and chemical reactions. The energy discharge from combustion of fuel leads to increased pressures and temperatures for the fluid flow, which disturbs the liquid movements inside the combustion chamber. A direct estimation of this connected/conjoined communication excluding sub models while accounting comprehensive and thorough chemistry is amazing and astonishing costly and extravagant in terms of estimation time and is hardly possible for complicated configurations.

Reduced order mechanisms considers maximum decisive/pivotal



chemistry in a limited range of temperature and equivalence ratio, and are utilized in addition to a sub models for turbulence-chemistry interaction analysis. One such model is the Probability Density Function (PDF) approach which concede an adequate estimation of turbulence-chemistry association. Fire proliferation is modeled such as the Zimont model using a progress variable based approach, which estimates the short term flame front velocity and position. These approaches also allows estimation of the combustion process on large meshes in complicated configurations with a good estimation power.

Simplified/lucid mechanisms are used to estimate the NOx generation due to

- Excessive High temperature
- Nitrogen in the fuel
- Fuel reactions in the flame front
- Sulphur oxides in the diesel
- Soot generation

Since these pollutants are very scarce /less amount compared to the total mass in the cylinder, these estimations can be decollate/detached from estimation of the prime energy release. At certain instances, it is carried out as post processing operation at the conclusion, but it is appropriate to consider the pollutant generation in the simulation, especially about the pollutants arising from partial combustion that oxidize afterwards in the cycle. In terms of automation and process compression, the problem setup at the solver stage can benefit from automation.

#### IV. GOVERNING EQUATION

There are primarily three conditions we take care of in CFD issue. They are conservation of Momentum (Javier Stokes equation), conservation of mass and conservation of Energy equation. The first, regularly intimate to as the Mass Conservation Equation, necessitates that the lump of liquid coming inside a firm space either leaves that space under control or stores init. It is in this way a lump uniformity necessity presented as a scientific structure, and is scalar equation. The another administering condition is the Conservation of Momentum which are direction dependent along magnitude and has a different condition for each of X Y and Z spatial coordinates.

##### Transient IC engine analysis

The drafted exemplary of inlet path and outlet paths of diesel and exhaust with ignition chamber is inserted in ANSYS Fluent 15.0 to do engine dynamic transient investigation. The CFD simulation is limited to cold flow excluding ignition. In this part we have discussed solver settings determination methodology and choice of turbulence model and bestowing limit boundaries to investigate.

##### Limits and starting settings

Here the limiting case of consistent pressure at the inlet and outlet paths. Border limits are appended and described over the similar cell face near the cells above the valve. The no slip wall limit condition in accordance with logarithmic law is exploited here. Adiabatic treatment is given to walls.

##### Mathematical model

Here RNG k-ε model is utilized in this hypothesis on the grounds that, in this model 'k' signifies the turbulence disturbances in kinetic energy and is the changes of the variations in speed, having dimensions of (L<sup>2</sup>/T<sup>2</sup>), e.g. m<sup>2</sup>/s<sup>2</sup>

.The turbulence eddy dissipation is denoted by 'ε' (the rate at which the speed changes are subsided) having dimensions of (L<sup>2</sup>T<sup>-3</sup>), e.g. m<sup>2</sup>/s<sup>3</sup>.

The turbulent kinetic energy equation as simulated has many expositions from the precise equation.

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = T_{ij} \frac{\partial u_j}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ V + \nu / \sigma \frac{\partial k}{\partial x_j} \right] \quad (01)$$

$$\tau_{i,j} = -\overline{u_i u_j} = 2\nu_t S_{ij} - \frac{2}{3} k \delta_{ij} \quad (02)$$

The initial segment on the Right of equation is k's generation, the next segment is (ε) is the peculiar abate per divided by mass. The interpretation of the movement of 'k' by molecular and agitated and disordered dispersal/dissemination is done by final term. The typical/classic k-ε model is the delinquency turbulence model in Fluent. It arbitrates transport condition for the dissipation rate rather than analysis of length scale.

$$\frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} = T_{ij} \frac{\partial u_j}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j} \left[ V + \nu / \sigma \frac{\partial k}{\partial x_j} \right] \quad (03)$$

$$\frac{\partial \varepsilon}{\partial t} + u_j \frac{\partial \varepsilon}{\partial x_j} = C_{z1} \frac{\varepsilon}{k} \tau_{ij} \frac{\partial u_i}{\partial x_j} - C_{z2} \varepsilon^2 + \frac{\partial}{\partial x_i} \left[ V + \nu / \sigma \frac{\partial \varepsilon}{\partial x_i} \right] \quad (04)$$

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (05)$$

For adjusting the Flows with larger Reynolds numbers, this model was borrowed and integrated/adopted. It is appropriate for flows in case, the choppiness is approximately iso-tropic, suitable for cases of flows in which the energy precipitation continues with nearby balancing in line with its generation.

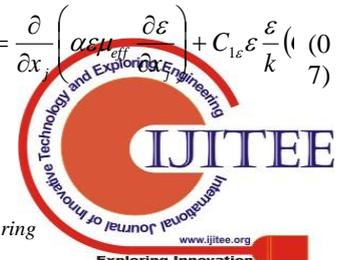
The turbulent flow model, consisting of high pressure splashing of fuel and effecting spray penetration, vaporization, entanglement of polyphase, composite nature grows increases its hardness. And still at the end the essential conditions of continuity and Navier-Stokes equation, k-ε turbulence model are used to represented the idea of liquid. At the point when zone of burning area enlarges, volumetric efficiency (1) will increase.

$$\eta_v = \frac{m_a (\text{mass of air inducted in cylinder per cycle})}{\text{Density of air inlet} \times V_d (\text{Displaced volume})}$$

Turbulence is known by change of speed field. Famous RNG k-ε model is utilized to simulate disturbance/agitation in this investigation. Utilizing an exhaustive credible strategy the RNG k-ε model was determined. It is closely resembling in structure to the typical/conventional k-ε model though having preference for accounting for impact of swirl, being significant for fuel burning investigation. The RNG k-ε Model has transport equations, these are,

$$\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{eff} \frac{\partial k}{\partial x_j} \right) + G_k \quad (6)$$

$$\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{eff} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \varepsilon \frac{\varepsilon}{k} \quad (7)$$



In these equations,  $G_k$  portrays/constitutes the production of agitated and disordered kinetic energy.  $G_b$  is the agitated and disordered kinetic energy on account to buoyancy.  $Y_M$  depicts the augmentation of the oscillating and alternating expansion and broadening in condensed and compact turbulence. The quantities  $a_k$  and  $a_f$  are the opposite and reverse convincing and compelling Prandtl numbers for  $k$  and  $\epsilon$ ,  $S_k$  and  $S_\epsilon$  are ascertained original terms. The  $C_{1\epsilon}$  and  $C_{2\epsilon}$  are model constants and determined provisionally from the RNG theory.

**Spray breakup model**

FLUENT gives following models, the wave model and the TAB model. In the current research TAB Model is utilized. This is based on the affinity admisti. A wavering and twisting droplet and ii.A spring mass system. For the present analysis, the contorting droplet development is taken.The equation administered by a damped, the movement of the droplet equator from its circular location aspect, force oscillator and state and uninterrupted phase. densities is taken. Relative velocity of the droplet is signified by  $u$ , the unconverted droplet radius is signified by  $r$ , droplet surface tension is signified by  $s$ .

$$F - kx - \frac{dx}{dt} = m \frac{d^2x}{dt^2} \tag{8}$$

$$\frac{k}{m} = C_k \frac{\sigma}{\rho l r^2} \tag{10}$$

**Geometry Decomposition for Sector Combustion Simulation:**

Selection of Sector is devised for engines with axisymmetric piston. The draft/state of the arch/dome geometry or the valve seats is insignificant as they will be cut afterwards. It is likewise prescribed also recommended for CI diesel engines as they are ignited by compression ignition.

The simulation of the engine commences when the geometry is imported/taken. The procured configuration/draft is fragmented into smaller volumes/segments before meshing. This enables/simplifies that each fragment to be meshed independently. Divided partitions /volume are further divided into sub-volumes and then the sub-volumes are meshed independently. Each fragment will be meshed into hexahedral or tetrahedral elements, contingent on the methodology.

There are some mesh topology prerequisites for valves and pistons. The piston should be at topmost position of combustion chamber prior the configuration/draft is configured /fragmented. With the piston at this position, the volume is the least. In general it is critical to fulfil the work topology necessity at TDC, however provides the advantage that the mesh will act correctly when the piston traverses away from TDC. However, the simulation requires a base valve lift between the valve and valve seat so that stacked cells can be put at the area of least valve lift. This complies that rift amidst/among the valve and valve seat will prevail. A disqualified interface is utilized to completely close the valve. Despite the fact that in principle a discretionarily little least valve lift can be utilized , in actual condition a value of 0.05 mm to 0.5 mm has been effectively utilized to run simulations using ANSYS Fluent. The meshing includes:

1. [Meshing for Sector Combustion Simulation](#)
2. [Global Mesh Settings for Sector Combustion Simulation](#)
3. [Local Mesh Settings for Sector Combustion Simulation](#)

The option Physics Preference enables the workbench to perform meshing depending on the physics of the analysis. The option Solver Preference allows the detailed view of mesh folder. The nearness and curvature for combustion simulation is accessed by Advanced Size Function under global mesh setting for sizing. This option provides/gives bigger control over the sizing functions with the available refinement mechanisms. The present research/study involves initially the simulation of gas temperatures inside the cylinder, and then simulation of liner temperature from gas side.

Following stepwise procedure is adopted for gas combustion simulation.

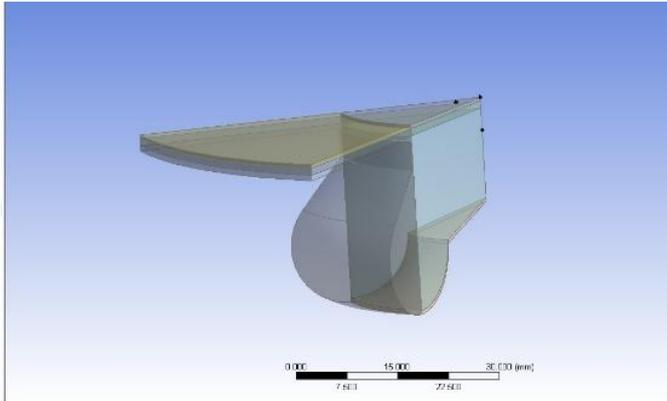
a. In-Cylinder Combustion Simulation.  
It incorporates simulation of the power stroke during the cycle, starting from shutting of valves to the end of the power stroke. As the maximum energy is evolved during early part of power stroke, the valves are shut or during the spent in shutting, the cylinder space above piston is the important stream domain, and the piston the only straight moving part. These simulations are also termed as "in-cylinder combustion" and however multi-dimensional, are less complex as far as geometry is concerned as compared to porous flow simulation. What's more if the, if the configuration is spin similar/proportional and has a one characteristic as a huge pressurized splash spray that dictates/controls the flow in the calculation/estimations, the full area can be drafted as a sector to accelerate/enhance the computation.

**Table 1: Initial inputs given for simulation to ICE Solver**

Parameter	Value/Details
Engine speed	1500 r.p.m.
Four stroke crank angles	720 °
Crank diameter	130mm
Connecting rod length	207 mm
Diameter of piston	105 mm
Distance between TDC to BDC	130 mm

b. Geometry arrangement:  
As space of piston and cylinder is axisymmetric, consequently sector configuration was chosen to portray describe the combustion space and was chosen to be configured. The burning space is configured as shown in Fig.1. It consists of the empty/void place/area on piston bowl, and the place overhead. The comprised angle of sector is 45°.





**Fig.1 Decided/configured Geometry for Combustion Simulation**

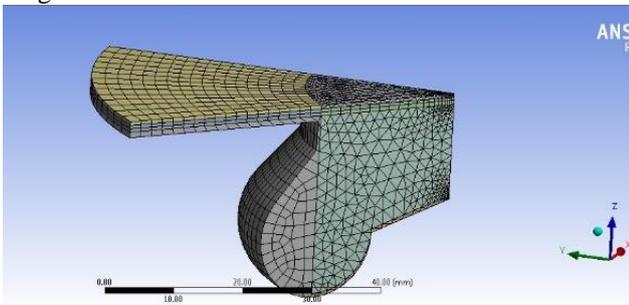
**c. Geometry clean up:**

The upward surfaces shown in Fig.1, are removed and the entire configuration is fragmented into 7 parts. These parts are:

1. Piston outer
2. Piston inner
3. Piston inflexion
4. Chamber outer bottom
5. Chamber inner bottom
6. Chamber outer top
7. Chamber inner top.

**d. Meshing:**

Drawn configuration was taken in the mesh modular. Both structured and unstructured meshes are utilized in this drafting. Structured mesh is six sided hexagonal shape and four sided quadrilateral as shown in Fig.2, whereas the unstructured mesh is three sided triangular.



**Fig.2 Grid of Combustion Space**

**e. ICE Solver set up:**

It mainly consists of following steps:  
Basic setting: Following inputs are given.

**Table 2: Inputs given during basic setting**

Type of input	Description
Type of Solution	Solution of Combustion Simulation
Nature of Simulation	Sector Simulation
Initialization	Yes
Report Mesh (Granted /Not granted)	Granted
Choice of Model	K-ε (k-Epsilon) Model.
Data sampling allowed	Yes

Auto save type	Crank angle
Frequency of AutoSaving	Every 10° crank angle.
No. of crank angle for run	180°
Swirl number	2.1

Physical and Chemical Setup: This step ensures allocation of proper chemical features.

**Table 3: Inputs given for physical and chemical step up of simulation**

Parameter	Details/Value
Pre Combustion mixture	Diesel and air
Start of injection	345° crank angle
End of injection	368° crank angle
Injection flow rate	1.54x10 <sup>-3</sup> Kg/sec
Evaporating species	n -heptane (C <sub>10</sub> H <sub>22</sub> )

Boundary conditions: Following boundary conditions are given.

**Table 4: Boundary condition**

Nature of Input	Description
Face of Cylinder	It is treated as wall

**Monitor definition:**

The values of computed properties viz. pressure, temperature, crank angle, volume integral, density etc. at any stage during iteration process can be monitored. The boundary conditions can be varied based on variations of these property values during iterations.

Initialization: The following input parameters are given to initiate the iteration process.

**Table 5: Inputs given during initialization**

Parameter	Value
Pressure at start of power stroke	164.98 bar
Temperature at the start of power stroke.	1000 K
<b>PATCHING ZONE:</b>	
Pressure	170bar
Temperature	2400 K

Post processing: Various cross section planes are considered to observe different properties of combustion process.

Set up: Following parameters are selected for computation.

**Table 6: Details of selections during computation by FLUENT**

Parameter	Details/Value.
Precision Method	Double.
Method of Results	Absolute velocity formulation
MODEL selection	k-ε Model
Cell Condition Fluid	Air
Mesh type	Dynamic
<b>Run calculation:</b>	
No of time steps	720
Iterations per time step	5

Results: The important combustion process



properties are noticed in CFD post processing method.

### V.RESULTS AND DISCUSSIONS

Results of upper mentioned simulation: Here sample of simulated temperatures are shown.(° Kelvin) at various crank angles.



Fig.3 Temperature Contour at Y-Z Plane, for Crank Angle 360.25°

Figure 3 shows the temperature counters of cross section of the cylinder at the start of the power

stroke. The contour shows that very high temperature gases are formed within the cylinder.

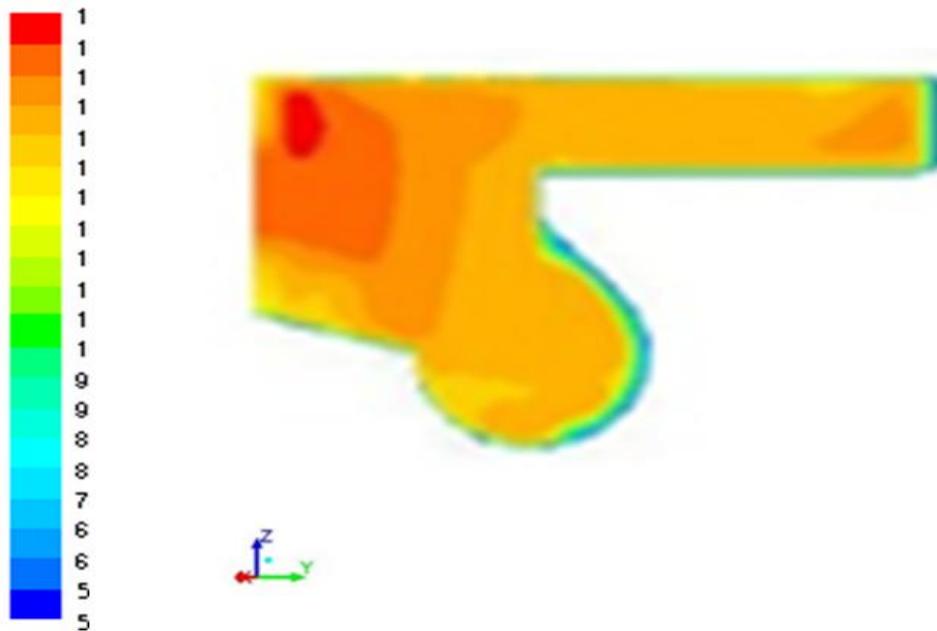
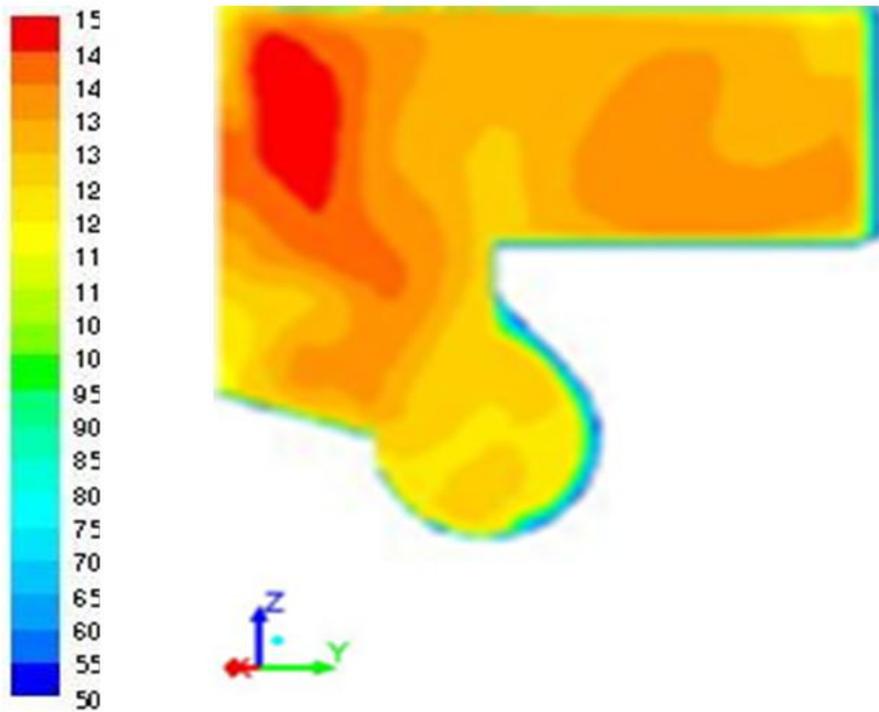


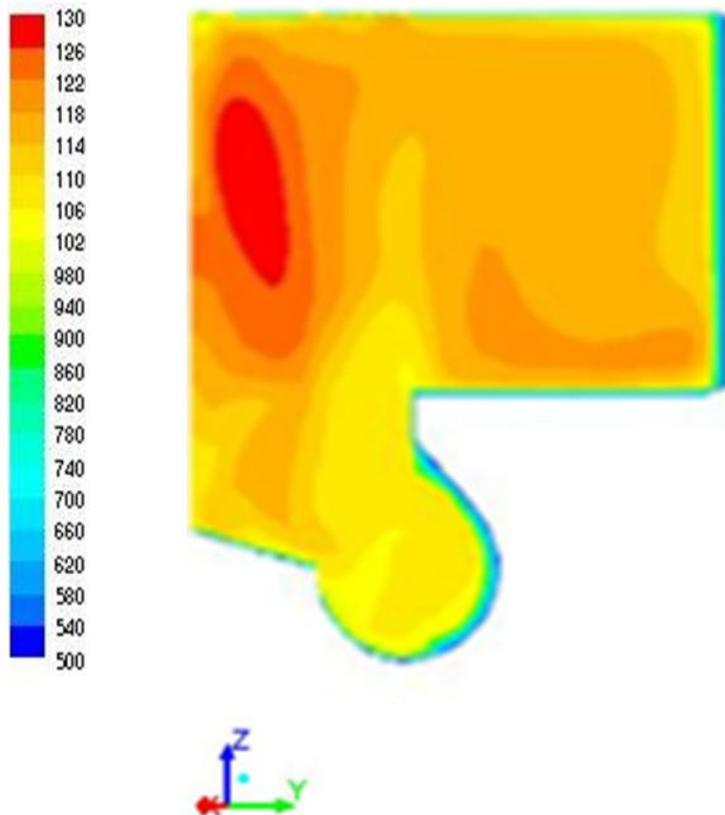
Fig.4 Temperature Contour at Y-Z Plane, for Crank Angle 384°



**Fig.5 Temperature Contour at Y-Z Plane, for Crank Angle 400°**

Figure 4 and 5 shows the flame propagation during the power stroke at crank angles equal to 384° and

400°. There is decrease of gas temperature as piston is going to BDC.



**Fig.6 Temperature Contour at Y-Z Plane, for Crank Angle 416°**

Figure 6 displays the temperature contour at the cross section of the cylinder at 416°. It is clear that there is further contraction in the gas temperature due to down

movement of the piston i.e. power utilization to move piston.

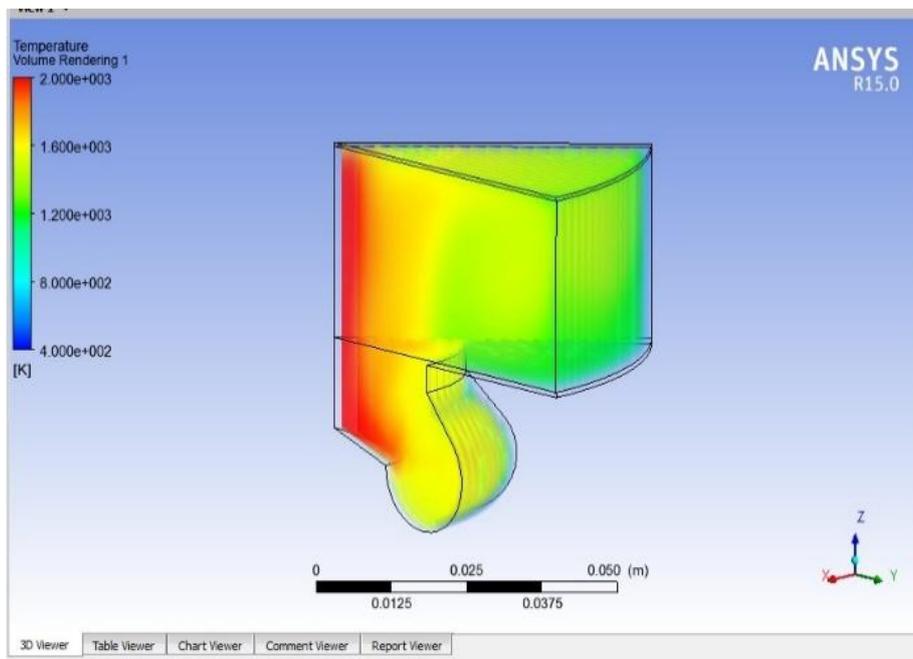


Fig.7 Three Dimensional Temperature Contour for Crank Angle 364°

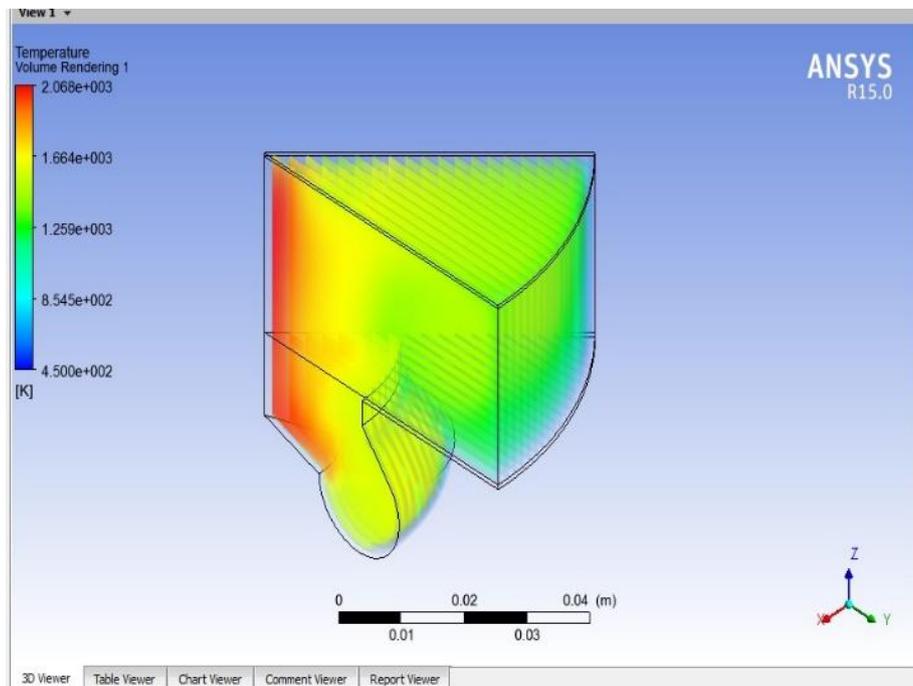


Fig.8 Three Dimensional Temperature Contour for Crank Angle 380°

As the diesel is injected in the combustion chamber during early part of power stroke, the preheated and compressed air gets mixed with diesel, combustion occurs giving maximum pressure rise due vigorous burning of fuel air mixture as the diesel has been already reached its ignition temperature, causing peak temperature rise to 2183 K in the cycle. As the flame spreads, part of the generated heat is given to piston and hence the temperature goes on reducing towards the liner wall. There are unaccounted radiation losses also. The heat from the flame is given to liner wall from gas side by

convection and due to thermal resistance offered by the gas the temperature of the liner wall further reduces.

Figure 7 and 8 shows three dimensional temperature contour plot of cylinder space at crank angles equal to 364° and 380°. The magnitude of temperature gradients with the cylinder space is significant, reaching maximum values at the middle of the cylinder and lowest at the cylinder wall, suggesting reduction in the temperature from middle of the cylinder to the wall. The mechanical power is transferred to the piston, thereby reducing the temperature of the gases and hence the temperature gradients within the cylinder.

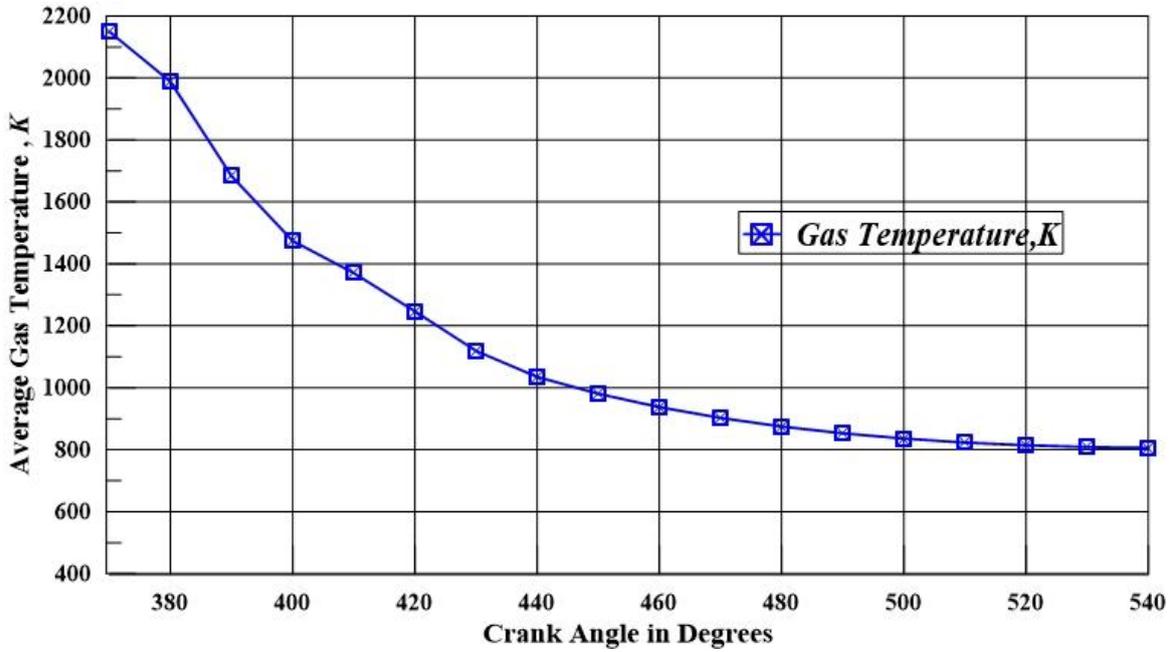


Fig.9 Variation of In-cylinder Temperature with Rotational Crank Angle

Above Figure 9 displays the discrepancy of average gas temperature with the crank angle. It is deduced that the highest temperature exists at the start of power stroke and it gradually decreases with increase in crank angle.

At the start of power stroke i.e. at crank angle 370°, high pressure and temperature gases formed the

temperature reaches to 2184 K. During the total expansion stroke when the piston comes to another dead centre the temperature of gases reached to 800 K. This provides temperature variation of gas in the longitudinal direction.

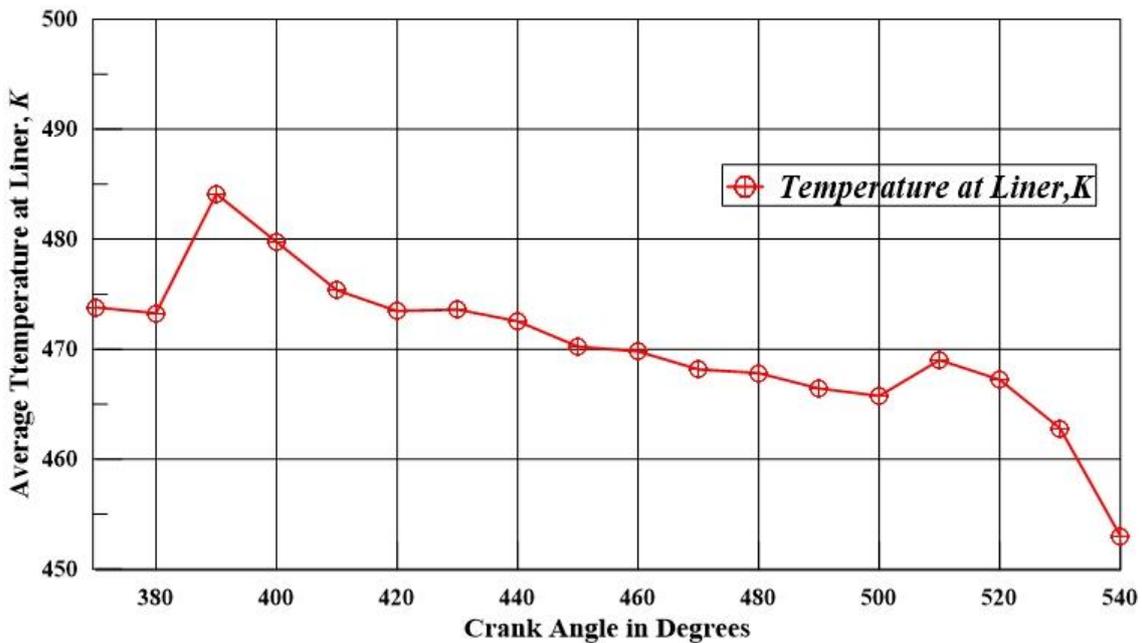


Fig.10 Variation of Liner Temperature with Crank Angle

Figure 10 displays the distinction of liner temperature with the crank angle. It is concluded that there is large temperature inequality between the gas side and the liner side. The temperature of liner also reduces as the crank angle increases.

At the start of power stroke i.e. at crank angle 360°, when the gas temperature reaches to 2184 K, the temperature at liner wall towards gas side is found to be 473K.

## VI. CONCLUSIONS

The simulation and modeling of combustion of selected diesel engine is carried out using ANSYS 15.0 workbench to investigate the temperatures of gases in power stroke which is conventionally difficult to measure due to limited accessibility of instrumentation at high temperatures and pressures in present era. The power stroke is intentionally selected for



simulation as the maximum temperatures in the cycle occur during it. The temperature of gas interior the cylinder is investigated numerically and further used, to determine the liner temperatures.

As the flame front travels towards the liner wall, initially due to spontaneous combustion of diesel, temperature is high and the fuel gets consumed and hence the flame front temperature decreases further reducing the temperature of liner wall. Also due to eventual transfer of gas power to piston the heat and eventual temperature of flame front and the cylinder space reduces thereby also reducing then the liner temperatures.

## VII. REFERENCES:

1. Carmen C. Barrios, Aida Domínguez-Sáez, Dévora Hormigo, Influence of hydrogen addition on combustion characteristics and particle number and size distribution emissions of a TDI Diesel Engine Fuel, 199 (2017) 162 – 168.
2. Taguchi optimization method”Materials and Design vol 27, March 2005 pp 853-861 Fanos Christodoulou, Athanasios Megaritis, Experimental investigation of the effects of simultaneous hydrogen and nitrogen addition on the emissions and combustion of a diesel engine, International Journal of Hydrogen Energy. 39 (2014), 2692 – 2702.
3. W. B. Santosoa, R. A. Bakara, A. Nurb, Combustion characteristics of diesel-hydrogen dual fuel engine at low load, International Conference on Sustainable Energy Engineering and Application, Energy Procedia 32 (2013) 3 – 10.
4. Constantin Pana, Christian Nutu, Experimental aspects of the hydrogen use at diesel engine, 10<sup>th</sup> International Conference Interdisciplinary in Engineering, 2017, 649 – 657. ghn
5. Rachan D. Shekar, H. R. Purushothama, Hydrogen induction to diesel engine working on Biodiesel: A Review. Global Challenges, Policy framework and Sustainable Development for Mining of Minerals and Fossile Energy Resources, Earth and Planetary science 2015, 385 – 392.
6. A.P.Singh, A.K.Agarwal, An Experimental Investigation of Combustion, Emissions and Performance of a Diesel Fuelled HCCI Engine. 2012 -28-0005 SAE International.
7. U.V.Kongre, V.K.Sunnapwar, CFD Modeling and Experimental Validation of Combustion in Direct Ignition Engine Fueled with Diesel, International Journal of Applied Engineering Research, Dindigul, Volume 1, No 3, 2010, 508 – 517.
8. Dr.M.J.Lewis, J.B.Moss, P.A.Rubini, CFD Modelling of Combustion and Heat Transfer in Compartment Fires, International Symposium on Fire Safety Science, Melbourne, 1997, 463 – 474.
9. MA.Siddique, S.A.Azeez, R.Mohammed, Simulation And CFD Analysis of Various Combustion Chamber Geometry of A C.I Engine Using CFX, International Refereed Journal of Engineering and Science (IRJES) Volume 5, Issue 8 (August 2016), 33 – 39.
10. Rajesh Bisane, Dhananjay Katpatal, Experimental Investigation & CFD Analysis of an Single Cylinder Four Stroke C.I. Engine Exhaust System, International Journal of Research in Engineering and Technology, 50 – 55.
11. H.Sushma and Jagadeesha.K.B, CFD modelling of the in-cylinder flow in direct injection Diesel engine, International Journal of Scientific and Research Publications, Volume 3, Issue 12, December 2013, 1 – 6.
12. Divyanshu Purohit, Pragma Mishra, Vishwanath Banskar, Flow Simulation of an I.C. Engine in FLUENT, ANSYS 14.0. International Conference on Emerging Trends in Mechanical and Electrical Engineering (ICETMEE- 13th-14th March 2014), 252 – 255.
13. Z.F. Tian, J. Abraham, Development of a two-dimensional internal combustion engines model using CFD for education purpose. 20th International Congress on Modelling and Simulation, Adelaide, Australia, 1–6 December 2013, 1575 – 1581.