CFD Simulation of In Cylinder Combustion of Multi-cylinder Diesel Engine

N.B.Totla[1*],Y.J.Bhalerao[2]
[1] Department of Mechanical Engineering, MIT Academy of Engineering,Alandi,Pune-412105, MS, India
*Corresponding Email - nbtotala@mitaoe.ac.in

ABSTRACT

The simulation /reproduction of in cylinder combustion for power stroke of multicylinder Diesel Engine is appropriate/fundamental to work upon for determination of temperature inside the cylinder. The current examination incorporates estimating of approximate temperature of in chamber gases during power stroke and accordingly liner temperature from gas side. The current investigation incorporates mathematical simulations dependent on Inbuilt ICE Combustion model in ANSYS 15.0 form, where burning space above cylinder during power stroke which is time transient is utilized. Correct averaged boundary limits were determined for different sections for the ignition model. The mesh modular is created. The prevailing of different ranges of temperature recorded in literature during power stroke along with differential liner temperatures from gas side are surveyed. Estimations are done for maximum and minimum temperatures also. The maximum in cylinder temperature was around 2150 K and least temperature was discovered to be 800 K. Likewise the most extreme temperature on liner from gas side along stroke was discovered to be 470K during power stroke. It has been additionally discovered that the greatest temperature of in chamber gases and liner from gas side endures just during early power stroke.

Keywords: Simulation; combustion; Temperature; approximate; cylinder

1. INTRODUCTION AND LITRETUTRE SURVEY

The shape of the piston bowl controls the movement of air and fuel as the piston comes up for the compression stroke before the mix is ignited and the piston is pushed downward. Engines have evolved since many decades along with pistons. They are obtaining smaller, light in weight i.e., smaller diameter piston. [1] Current pistons are made of aluminum alloys with more content of silicon. This improves thermal resistance and eventual distortion. Probably the greatest progression in cylinder innovation is the utilization of various cylinder "tops" or "crowns," the part that enters the ignition chamber and is exposed to burning. While ancient cylinder tops were generally flat from top, but currently many innovations had been there in piston tops effecting the combustion cycle. Nowadays the bowl piston shape is essentially employed in diesel engines. [2] As diesel is combusted by compression ignition hence the cylinder crown/top functions as an ignition chamber. These types of Engines frequently use cylinders with diversely formed crowns, despite the fact that with direct development of air and fuel takes place as the cylinder comes up for the compression stroke. There is a turbulent mixture comprising of air and fuel in the vortex form inside the space of bowl of piston before ignition occurs. By influencing the air/fuel blend, one can accomplish better and more effective ignition, which prompts more power. The piston top side have a wide range of shapes; some are likewise intended to improve efficiency, specific fuel consumption. With direct infusion turning into the better new innovation for fuel engines, expect interestingly bowled cylinders to turn out to be increasingly well known. In rapid direct-infusion Diesel engines, the stream conditions inside the chamber toward the finish of the compression stroke, are flawlessly focused near (TDC), are basic for combustion cycle. These are dictated by the air streaming into the chamber through the suction valves during the suction cycle and by its advancement during the compression stroke [6,7]. Numerous scientists had been concentrated on cylinder configuration affecting the air fuel turbulence on piston bowl. This section surveys the past distributed writings, which establishes the framework and reason for additional work in this undertaking. This assists with giving a superior comprehension about the subject and furthermore goes about as a rule for this hypothesis. Diesel engines are reliably great wellspring of energy for a wide range of locomotives train and modern applications. Ignition research is wide running a direct result of CFD. In compression stroke, diesel infusion, dynamics of reactions, makes an effect on the ignition cycle. Already, the consuming of fuel and development/advancement of fuel combustion had been cut/etched with numerous unmistakable models viz. RIF Model, supported to flame propagation model, time scale models, and the RIF's model. Ongoing investigators identified with the advancement/progress of new and solid models for consuming of fuel measurement has been documented in the writing. Here the exclusive CFD of combustion of in cylinder gases during power stroke is considered and carried out in ANSYS 15.0 Workbench using standard ICE module [3,4,9]. This is carried out for multi cylinder diesel engine. The mesh modular and post processing is carried out with the temperature estimations at all crank angles.

For this entire research work following literature is [1] Presented a new approach, focusing on optimizing the estimating time required needed to perform heat transfer simulations. It is relied on utilization of the Rate of Heat Release taken from CFD estimations to substitute the combustion process. The proposed approach is crosschecked for the heat flowing to the piston wall. The research unveils that the heat transfer and rate of heat release approach gives better results in terms of spatially averaged values for complete engine cycle and reduces considerable computational cost. Gap identified is it computed and correlated heat transfer and rate of heat release but the computation of in cylinder gas temperature is not done.[2] Developed an assimilated methodology, including suction and exhaust system, combustion system, cooling system, and components such as head/block/gasket were coupled. In particular, suction flow, ignition and burning, performance of cooling process and component temperatures were estimated in iterative way for V6 engine. The results show that characteristic time scale combustion model can create combustion rate satisfactorily over a wide range of engine speed. The identified gap is it integrated many systems for heat calculations but did not compute neither simulated in cylinder combustion process. [3] Focused on demonstration of the grid-convergent modeling perspective for simulating a one cylinder diesel engine. The simulated injector is specified exit diameter of 259 μm of nozzle. Simulations using various minimum grid sizes (ranging from 125 μm to 1000 μm) are differentiated for following parameters, ignition delay, NOx, HC, and soot emissions, pressure, heat release rate. The gap in this paper is the absence of meshing for temperature estimations inside cylinder. The idea of Grid convergent modeling is taken as an assistance for research in this paper.[4] A thermal insulation coating with a lesser thermal conductivity was manufactured for attaining this temperature swing. To ensure the swing concept, a technique to estimate the temperature in a diesel spray flame impingement section where a more of heat loss depletion would be forecasted was set out. Measuring the lifetime of laser-induced phosphorescence (abbreviated to LIP, below) was selected to measure the surface temperature which changes with a high responsiveness. Here, the insufficiency was observed in the sole in cylinder Gas combustion simulation by CFD which involved dynamic meshing and post processing for computation of gas temperatures in multicylinder diesel engine. The literature complied the CFD carried out for in cylinder combustion, here the same gap is identified and taken for research/study.[5] Put a brief resistor model for estimation of temperature of wall in Compression ignition engine with piston cooling . The model utilizes the transient in -chamber pressing factor and some typically estimated operational variables to foresee the temperature of the primary components of the engine. For all variables, an articulation as a component of the configuration engine, operating variables and properties of material was determined to prepare the model pertinent to other related engines. The model very well simulated the temperature of liner, cylinder head and piston at various conditions such the beginning of injection etc. The gap seen here is the model did not compute the in-cylinder gas combustion temperature. [6] Examined a mathematical approach of transient heat transfer which was a three dimensional process for V8 engine by adopting a 3-D transient limited volume strategy to tackle the conduction equation which is introduced first. This is trailed by the usage of the coupling conditions at the gas–solid interface into the KIVA code. The mathematical model is approved by a one-dimensional conduction equation. At last it presented a three dimensional simulation with conjugate heat transfer on FORD Engine.[7] Coupled temperature fields in both the fluid and the solid domains by imparting same heat flux and temperature at the interface. The articulation was initially ensured and crosschecked with analytical solutions. This was then put for simulation of the in-cylinder combustion process and the solid heat conduction in a compression ignition engine for variable operating conditions. Take away from this paper is the methodology adopted for in cylinder combustion process.[8] Researched the process of momentary temperature (cyclic) in the ignition chamber walls of a turbocharged compression ignition engine during dynamic operation after an unit increment in load. For this a tentatively approved simulation code of the thermodynamic pattern of the engine during transient conditions is utilized. This considers the transient activity of the fuel pump and the advancement of contact force utilizing an itemized per degree crank angle point sub model, This takes into account the transient operation of the fuel pump and the development of friction torque using a detailed per degree crank angle sub model, The thermodynamic model of the engine is appropriately combined to a wall periodic heat conduction model, which utilizes the gas temperature variation as input condition for entire cycle.[9] Applied methodology successfully to a compression ignition engine which is liquid cooled, four stroke and direct injection type and it carried computation of the piston and cylinder wall temperature. Here numerical simulations based on FEM models and experimental procedures based on the use of temperature measuring sensors. This research aimed at determination of e distortion in the piston, temperature and radial thermal stresses after thermal loading. Here gap determined is the absence of estimation of in cylinder temperature during power stroke.

[10] Studied, a three-dimensional, dual-fuel, in-cylinder model and developed the same. This is used to provide an enhanced know how of the working features arising from the interaction between the gaseous fuel and the pilot fuel, the prior ignition processes, and eventual combustion of the pilot fuel and gas during the piston movement. [11] Analyzed the consequences of engine configuration in concern with emissions by analyzing 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. The Gap of the research is that, the effect of so-called turbulence on exhaust emissions is studied but its effect on estimation of in cylinder thermal parameters is lacking. [12] Studied the energy 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. The gap identified is that, it did not study effect of minimum fuel usage on temperatures in the cylinder. [13] Investigated the consequences of various piston configurations on movement of air and agitation /instability in the combustion space of a diesel engine which is direct injection type using FLUENT Code involving CFD. The gap seen is that the effect of movement of air resulting from piston shapes on combustion is studied and thereafter temperature values are not studied. [14] 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. The take over from this research is the the evaluation strategy of temperature profiles is for present research. [15] In the present and depicted literature, the deficiency had been seen and noticed in the exclusive in cylinder Gas combustion CFD involving dynamic meshing and post processing for determination of gas temperatures in multycylinder diesel engine. [16] Explored the use of standard code of FLUENT which is Computational Fluid dynamics (CFD) in view of configuring convoluted combustion development in diesel engine. On single diesel engine, tests were performed at a full load and 1500 rpm, combustion process conditions such as pressure rise, combustion pressure and rate of heat liberation were estimated through experimentation. The FLUENT was also utilized to replicate fuel burning process. They reported that the experimental values were in good agreement with the predicted values of CFD simulations. The Gap identified is that, it did not simulate the temperature distribution of in cylinder gases. [17] Reported consequences of concurrent/coincident H2 + N2 admission charge enhancement over the diesel engine emission and burning process. Here study of admission of conserved H2 + N2 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 conditions NOx, BSN and CO emissions reduction are achieved by H2 + N2 enrichment. Here other than controlled emissions, nitrogen emission ingredients are also calculated and shown to be negligible. The Gap seen is that the study includes the effect on nitrogen and other emissions but lacks in temperature simulation existing in the cylinder. [18] Studied the burning properties of a CI engine fuelled with hydrogen which was admitted prior to the admission of the fuel in the combustion chamber with the help of mixer. The Gap found is that, it studied the combustion properties but it did not relate with the in cylinder temperature estimations. [19] Recorded the improvement of engine efficiency and pollutant performance for truck diesel engine by using hydrogen as fuel and showed that for determination gas side liner temperature the refinement of simulation is required . From this paper is, in cylinder temperature simulation methodology as a guiding tool for present research. [20] 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 and eventually 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 classical combustion. EGR regulates the combustion rate positively and automatically and boosts exhaust radiation nature at the expense of marginally lesser attainment. The Gap found was that, it evaluated the combustion rates but lacked simultaneous evaluation of in cylinder temperature simulation. [21] Reported numerical analysis for combustion process in diesel engine tests were performed at 1500 rpm on one cylinder which was direct injection in nature under full load and deduced that shaping of combustion process can rely on CFD modelling as a better tool. Take away from this paper were the CFD techniques required for this research

In the presented and depicted literature, the deficiency had been seen and noticed in the exclusive in cylinder Gas combustion simulation for temperature determination, involving dynamic meshing and post processing for determination of gas temperatures in multicylinder diesel engine. Here the same gap is identified and taken for research. The literature is complying the CFD simulation carried out in present research for in cylinder combustion. Hereby the Novelty and uniqueness of the undertaken research is ensured and confirmed as per above depicted extensive literature survey from reputed and peer reviewed Technical Journals, Research articles.

2. MATERIALS AND METHODS

Here the in cylinder combustion simulation is carried out first cylinder of four cylinder diesel engine

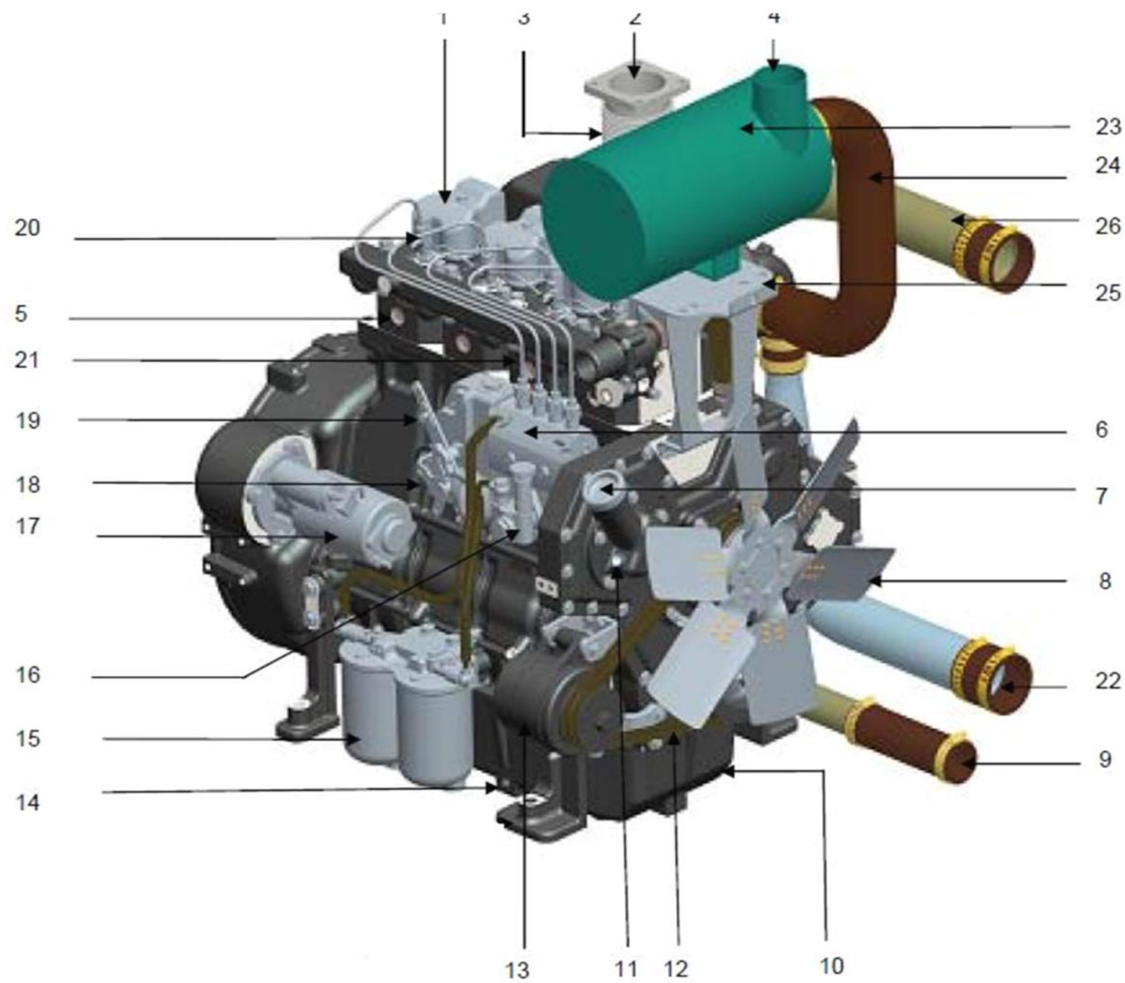
Engine selected for above depicted research:

1. It is a 4-cylinder Inline, direct injection and turbocharged diesel engine.
2. Detail specifications of Engine (Table1)
3. Engine Model. (Fig 1)

Table-1: - Engine Technical Specification- Genset Application

Sr. No.	Parameter	Unit	Specifications
1	Engine Model	-	4G11TAG3
2	Engine Rating	kW@rpm	114@1500
3	Bore X Stroke	mm	105X130
4	No. of cylinders	-	4
5	Engine Configuration	-	Inline

6	Working Principle	-	Four-Stroke
7	Type of Aspiration	-	Turbo-Charged, After cooled
8	Number of Valves per cylinder	-	2
9	Compression ratio	-	16.2:1 +/-0.5
10	Swept Volume	litre	4.5
11	Static Fuel injection timing	Deg BTDC	17 +/-1
12	BMEP (At rated power)	bar	20.3
13	Cooling	-	Water and Ethylene glycol cooled.
14	Water pump	-	Centrifugal –Gear driven
15	Lubricating oil pump	-	G rotor-Gear driven



1	Rocker cover	14	Oil drain plug
2	Exhaust out	15	Fuel filter
3	Exhaust bellow	16	Fuel feed pump
4	Air inlet	17	Starter
5	Cylinder head	18	Stop lever
6	Fuel pump	19	Throttle lever
7	Oil filler cap	20	Injector
8	Radiator fan	21	High pressure pipe
9	Water in	22	CAC air in pipe
10	Oil Pan	23	Air filter
11	Belts	24	Air filter to turbo connection pipe
12	Spout	25	Filter bracket
13	Alternator	26	CAC to inlet manifold connection pipe

Fig.1 Selected four Cylinder Diesel Engine

Approximate estimation of the liner temperatures from gas side:

The ANSYS workbench has been used to determine the approximate temperatures of in cylinder gases and inerliner from gas side. The version is 15.0. The CFD has been done of in cylinder combustion process. The power stroke has been simulated of first engine,as maximum temperatures occur during power stroke.

I) Steps involved in GAS COMBUSTION simulation

a) The inbuilt ICE system of ANSYS: This is a inbuilt module in this workbench which had been employed for combustion simulation . Combustion simulation involves simulation of the power stroke during the engine cycle, starting from closing of valves to the end of the compression stroke. Since the valves are closed or in the process of closing, the combustion chamber is the chief flow domain, and the piston the sole moving part. These simulations are known as "in-cylinder combustion" and though multi-dimensional, are less complicated geometrically than a port flow simulation. In addition, if the geometry is rotationally symmetric and has a single feature like a very high pressure spray that dominates the flow in the calculation, the entire domain can be Modelled as a sector to speed up the calculation. Here following inputs are given as depicted in Table 2. The module selected for computation and simulation is shown in Fig.3.The module selected is ICE Engine Module. Here, this module is inbuilt module for in cylinder combustion simulation for estimation of temperature of gases. No CFD Coding is required for this simulation. Hence no codes are written. The module incorporates and consists all conservation equations, i.e;Energy, mass and momentum. It also incorporates the variable air fuel ratio conditions, the swirl, the turbulence of air –fuel mixture. All the required inputs, boundary conditions, parameters are given and the simulation is carried out.

It consists of the empty/void place/area on piston bowl, and the place overhead. The comprised angle of sector is 450.As the configuration of piston cylinder is a symmetrical, by swapping or rotating the configured geometry, the whole piston can be simulated, as further it covers major portion, significant part of combustion space reflecting combustion phenomenon by the sectorial part with included angle of 45 degree. It has been reviewed that, to have results of simulation for full 360 degree sweeping of cylinder gas space, in horizontal plane, minimum 45 degree crank angle configured geometry is required , which had been taken and ensurity of same is seen.

Table 2. Details of input given for starting simulation to ICE Solver

Sr. No.	Parameter	Value/Details
1	Crank Shaft speed	1500 r.p.m.
2	Crank rotation period	720 $^{\circ}$
3	Crank radius	65mm
4	Connecting rod length	207 mm

5	Bore	105 mm
6	Stroke	130 mm

b) Geometry preparation-As the piston and combustion chamber space is symmetrical about axis, hence sector geometry of the combustion space was decided to be drafted. Here the combustion space was drafted. It consists of the hollow space on piston bowl, and the space above it. Included angle of sector is 45◦ [Fig.2] Though the air fuel ratios, at various section spaces along an horizontal plane varies, at various places simultaneously along stroke length, the chosen simulation module, i.e.; the ICE Module for the simulation ,the used empirical correlations in simulation and equations, take care of varying air fuel ratios at all above depicted place. Also further the rate of turbulence of air fuel mixture may vary ,but the term taken into consideration has been incorporated in the simulation equations in the simulation module.The pressure at all crank angles is known from the data which has been procured from Pressure-crank angle diagram. So it also takes into account all pressures existing at all crank angles during power stroke.

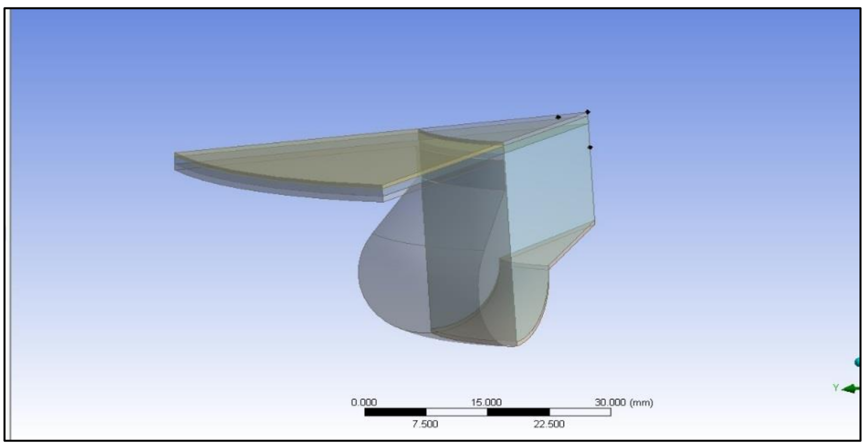


Fig.2 Drafted geometry for combustion simulation

c) Geometry clean up-In Fig.2 unwanted surfaces are removed. Total geometry is divided into various parts as shown

d) Meshing- Drafted geometry was imported in the mesh modular. Here both structured and unstructured meshes are used. Structured mesh is quadrilateral and hexagonal shape. Unstructured mesh is triangular. [Fig.3]

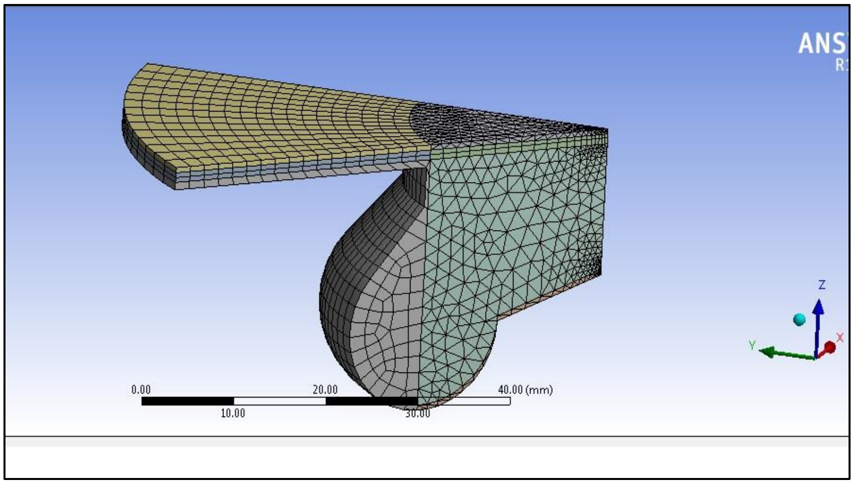


Fig.3 Meshing for combustion simulation

e) ICE Solver set up -It consisted of six steps

i) Basic setting-Here following inputs are given [Table 3]

Table 3. Inputs given during basic setting

Sr. No.	Type of input	Details/Values/Selection
1	Solution Type	Combustion Simulation Solution.
2	Simulation type	Sector Simulation
3	Initialization (Yes/No)	Yes
4	Report Mesh (Allowed /Not allowed)	Allowed
5	Model Selection	K-€(k-Epsilon) Model.
6	Data sampling allowed (Yes/No)	Yes.
7	Auto save type	Crank angle
8	Auto save frequency	Every 10° crank angle.
9	No. of crank angle to run.	180°
10	Swirl no.	2.1

ii) Physical and Chemical Set up-Chemical features such as material input is selected from FLUENT [Table 4]

Table.4 Details of input given for physical and chemical step up of simulation

Sr. No.	Parameter	Details/Value.
1	Pre-Combustion mixture	Diesel and air
2	Start of injection	345° crank angle
3	End of injection	368° crank angle
4	Injection flow rate	$1.54x10^{-3}$Kg/sec
5	Evaporating species	n -heptane $C_{10}H_{22}$
6	Cylinder Face	It is treated as wall

iii) Monitor definition-To observe computed properties at any stage of computation this monitoring is defined. It can be for pressure, temperature, crank angle, volume integral, density, etc. If they are not compatible, then the inputs at various stages and the boundary conditions can be changed.

iv) Initialization- In this, thermal parameters are given as input for initialization of post processing [Table 5]

Table 5. Inputs given during initialization

Sr. No.	Parameter	Details/Value.
1	Pressure at start of power stroke	164.98 bar
2	Temperature at the start of power stroke.	1000 K
3	PATCHING ZONE-	
a)	Pressure	170bar
b).	Temperature	2400 K

v) Post processing-Here images of various properties at planes are taken, discussed in next section.

vi) Set up- FLUENT is used for computation.

Again following added parameters are selected [Table 6]

Table 6. Details of selections during computation by FLUENT

Sr. No.	Parameter	Details/Value.
1.	Precision Method	Double.
2	Method of Results	Absolute velocity formulation
3	MODEL selection	k-€ Model
4	Cell Condition Fluid	Air.
5	Mesh type	Dynamic.

6	**Run calculation:**	
6.1	No of time steps	720
6.2	Iterations per time step	5

The mesh modular is used to mesh the selected sectorial configuration of combustion space. The meshes done for various crank angles during power stroke are depicted. [Fig.4 to Fig.18]

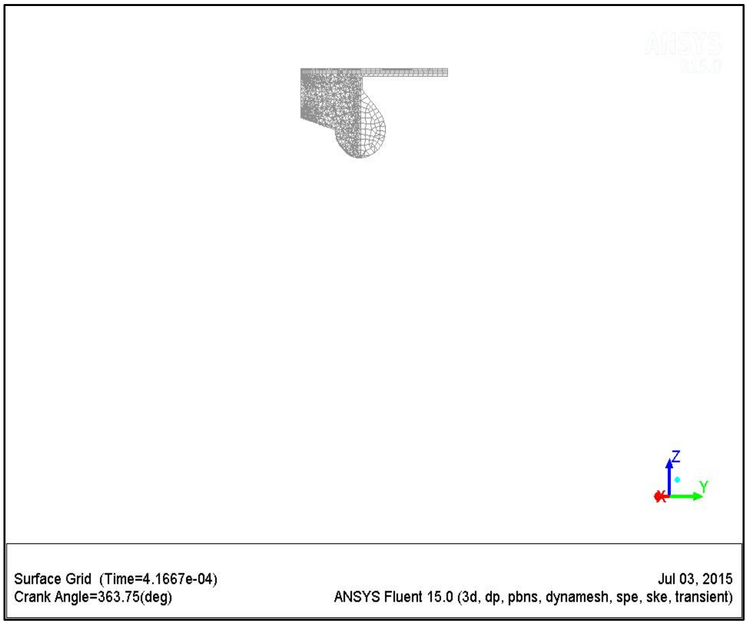


Fig.4 Meshing at 363.75° crank angle.

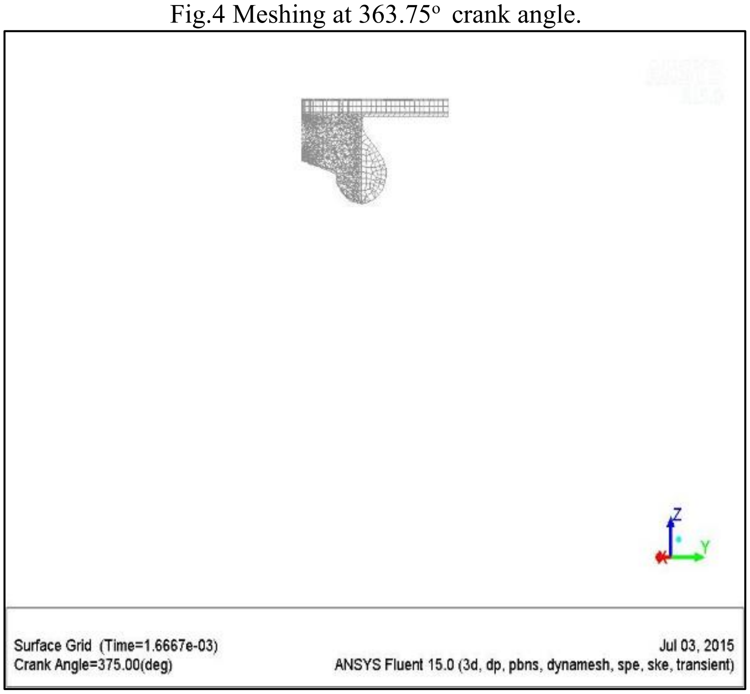


Fig.5 Meshing at 375° crank angle

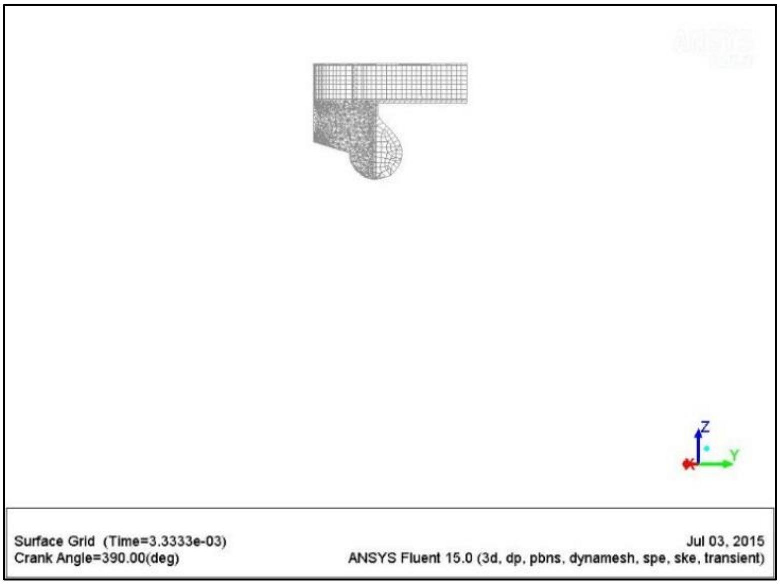


Fig. 6 Meshing at 390° crank angle

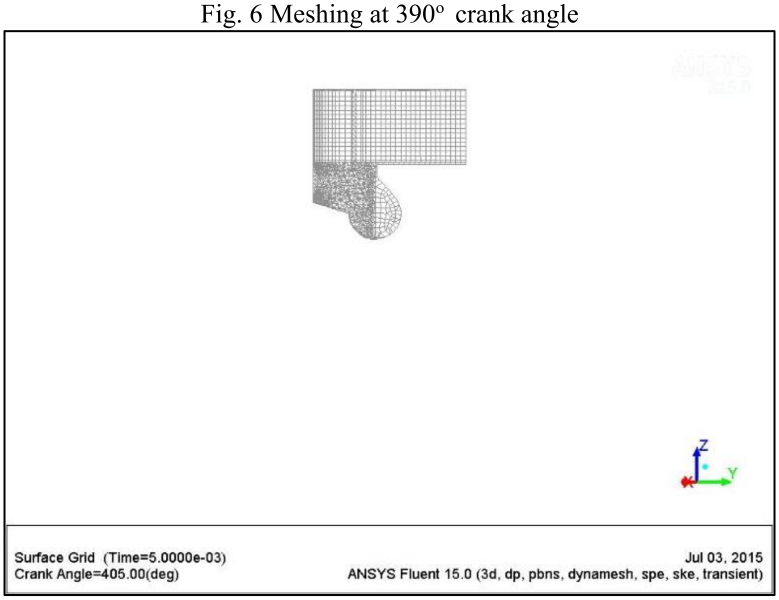


Fig.7 Meshing at 405° crank angle

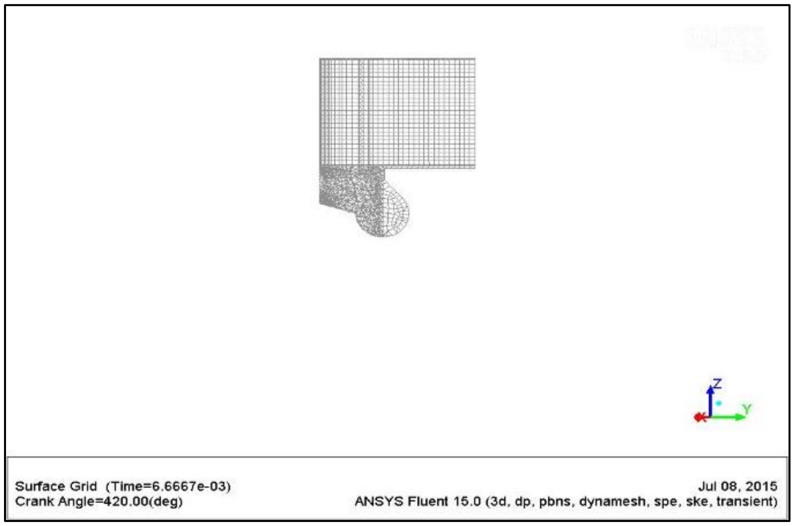


Fig.8 Meshing at 420^{o} crank angle

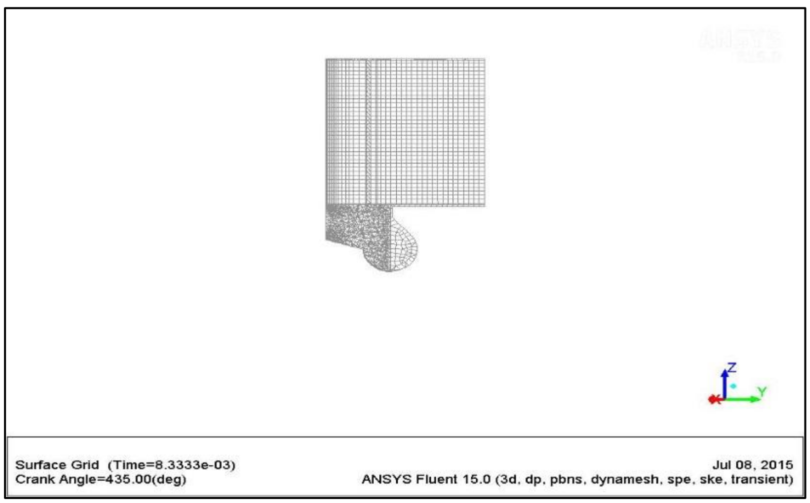


Fig.9 Meshing at 435^{o} crank angle

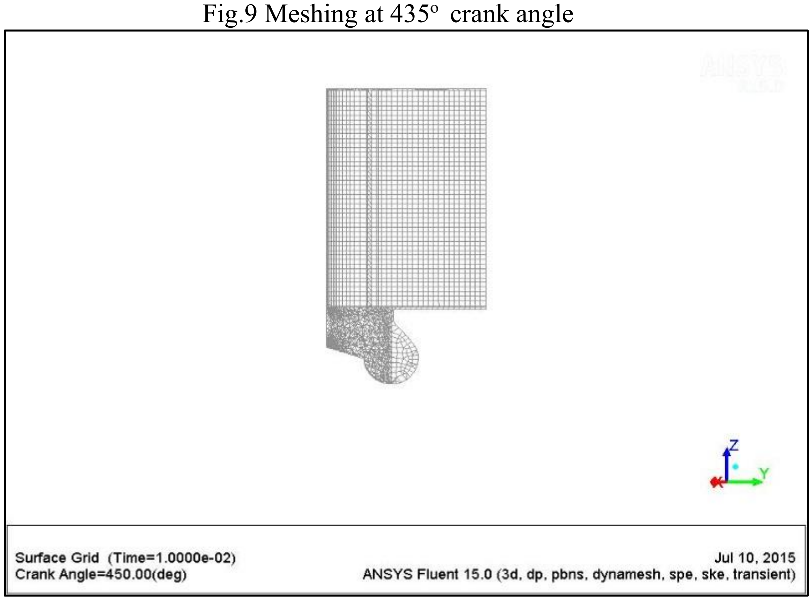


Fig.10 Meshing at 450^{o} crank angle

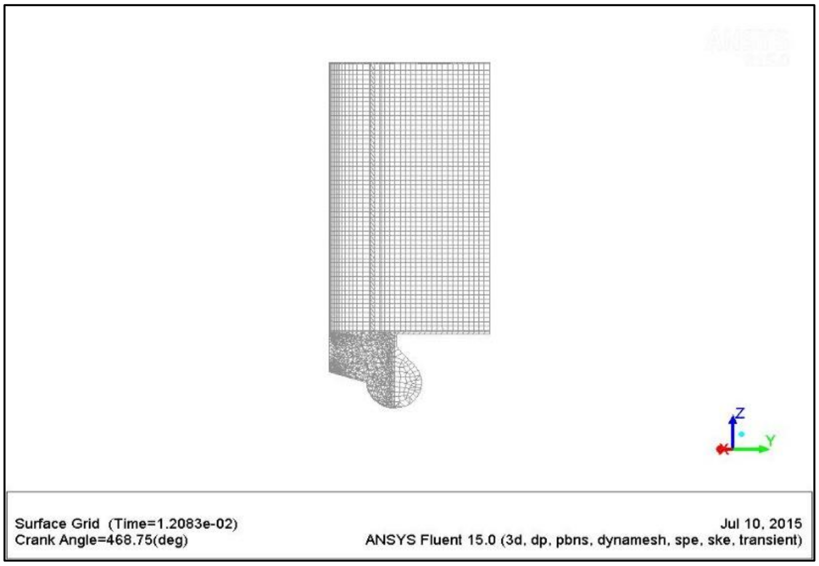


Fig.11 Meshing at 468.75° crank angle

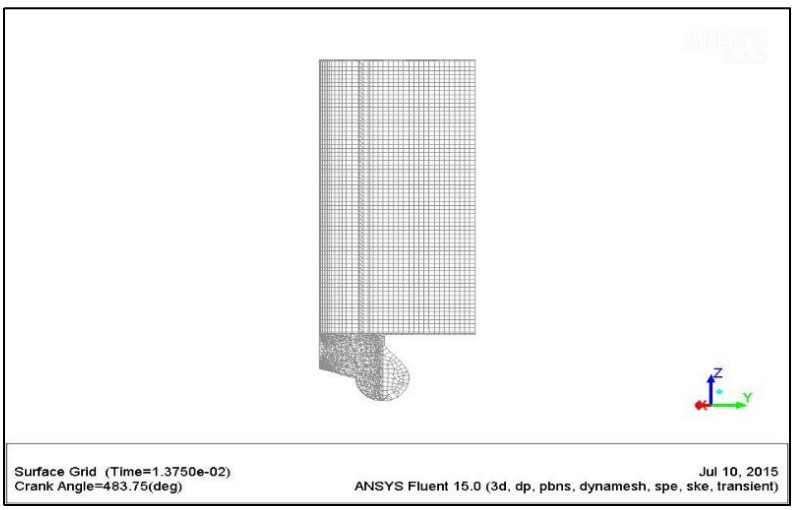


Fig.12 Meshing at 483.75° crank angle

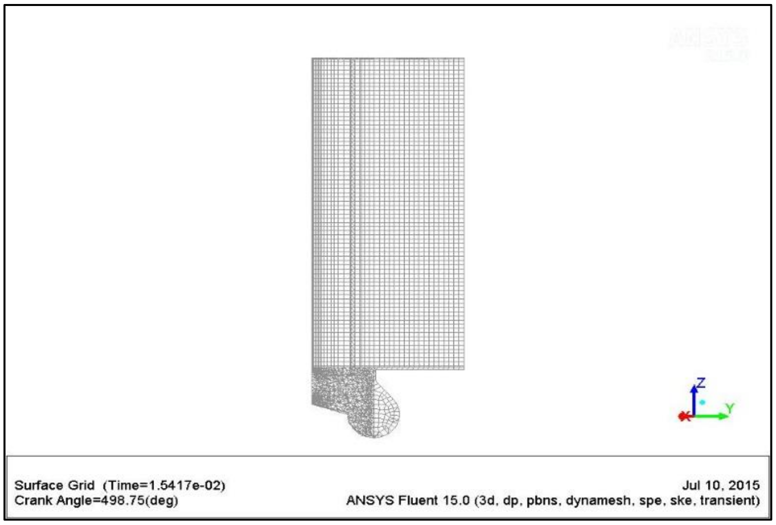


Fig.13 Meshing at 498.75° crank angle

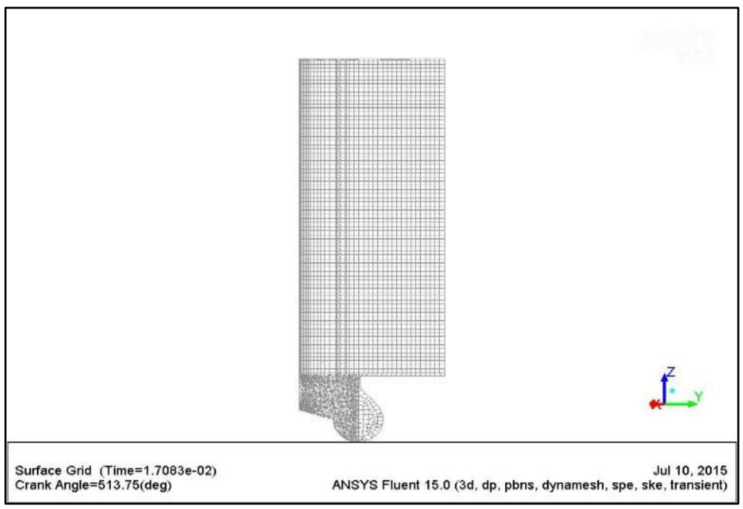


Fig.14 Meshing at 513.75° crank angle

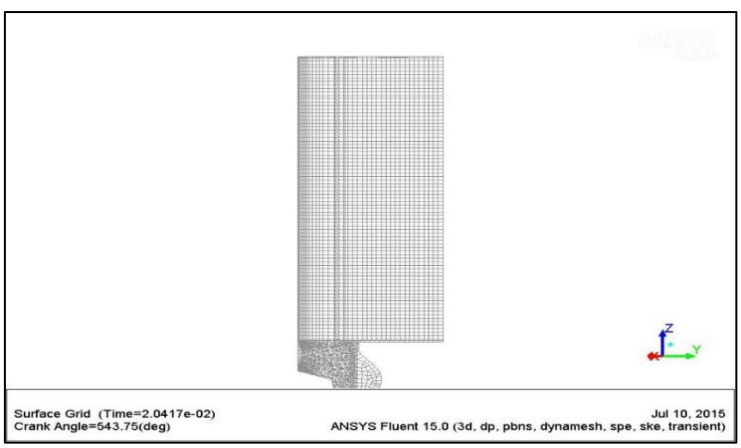


Fig.15 Meshing at 513.75° crank angle

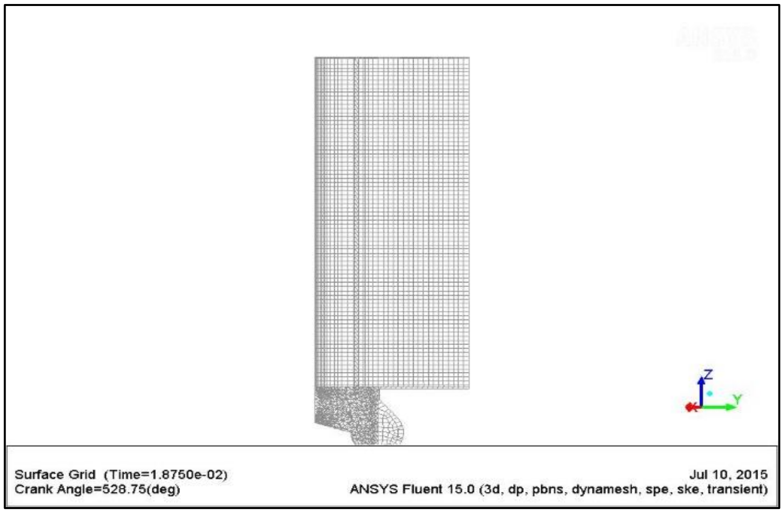


Fig.16 Meshing at 543.75^{o} crank angle

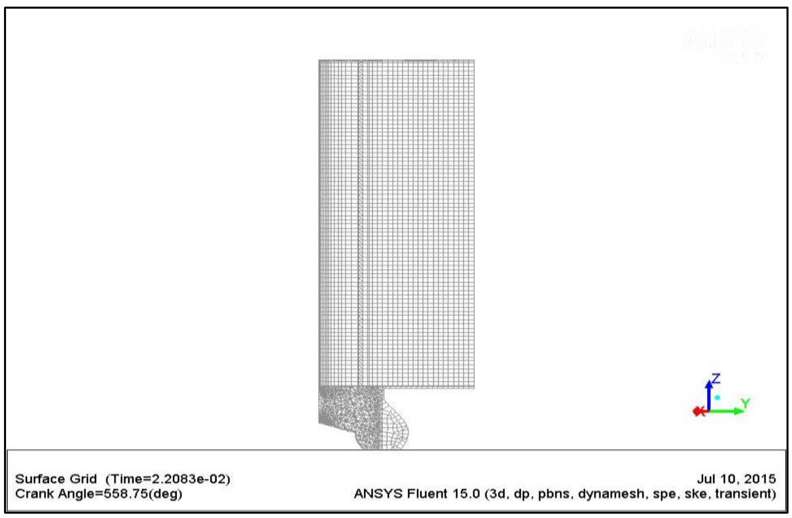


Fig.17 Meshing at 558.75^{o} crank angle

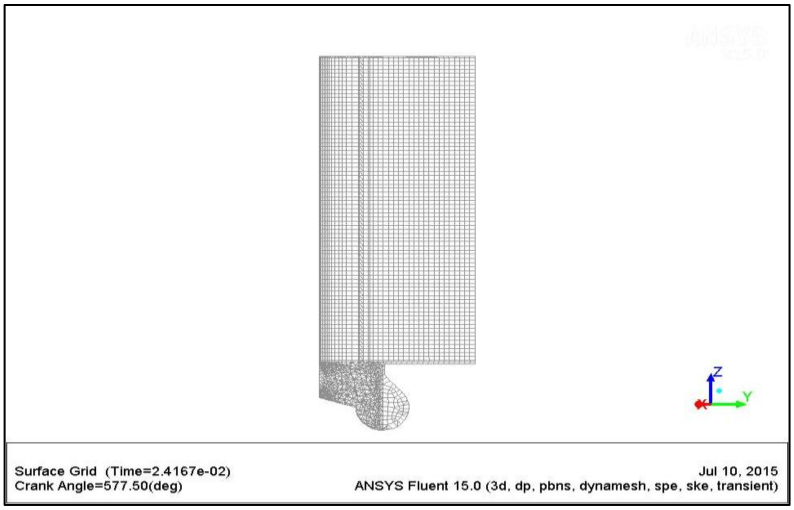


Fig.18 Meshing at 577.50° crank angle.

The post processing is done after meshing to get under mentioned results.

Grid Independence analysis:

Here in this particular simulation for determination of temperature of in cylinder gases during combustion and specially during power stroke, the convergence is attained for temperatures calculated in two different directions as mentioned under. This is for drafted geometry and configuration which is mentioned [Fig 3,4] .The said convergence is attained by 6-7 trials for each computations and estimations of temperatures at various places. Here in this research, the meshing, rather dynamic meshing are also carried out at all crank angles during power stroke at an interval of 0.25 degree crank angle rotation of crankshaft. That is, there are180x4=720 sets of temperatures of in cylinder gases. These are in longitudinal direction i.e.; along stroke length. Then next, the temperatures are also determined along horizontal plane also from central axis of piston to liner wall, gas side for power stroke. Huge readings, database and repository of readings and observations, mesh files,postprocessed resuts file had been resulted due to all above simulation done. These were all stored in computer. The storage space in the used computer system was insufficient to save and store all grid independence data/files which was done in all computations. However, some illustrious graphs and results ensuring that Grid independence analysis was being carried out, are presented and depicted here [Fig.19 and 20]

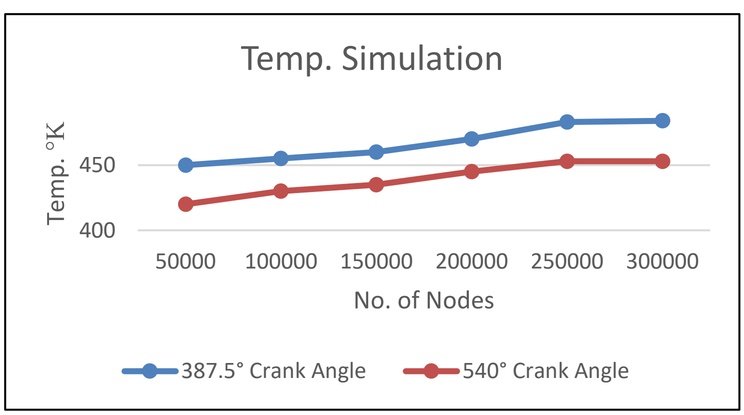


Fig.19 Grid independence results in the form of graph for simulated temperature at centre of piston top side (Gaseous temperature) at 387.5 ° and 540 ° crank angle.

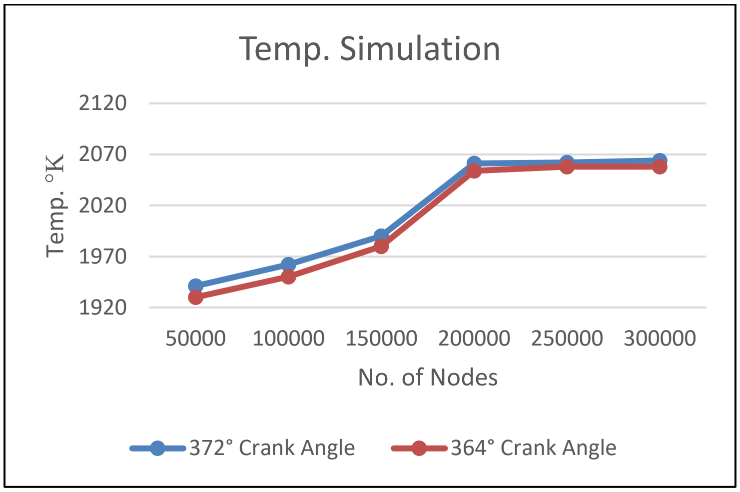
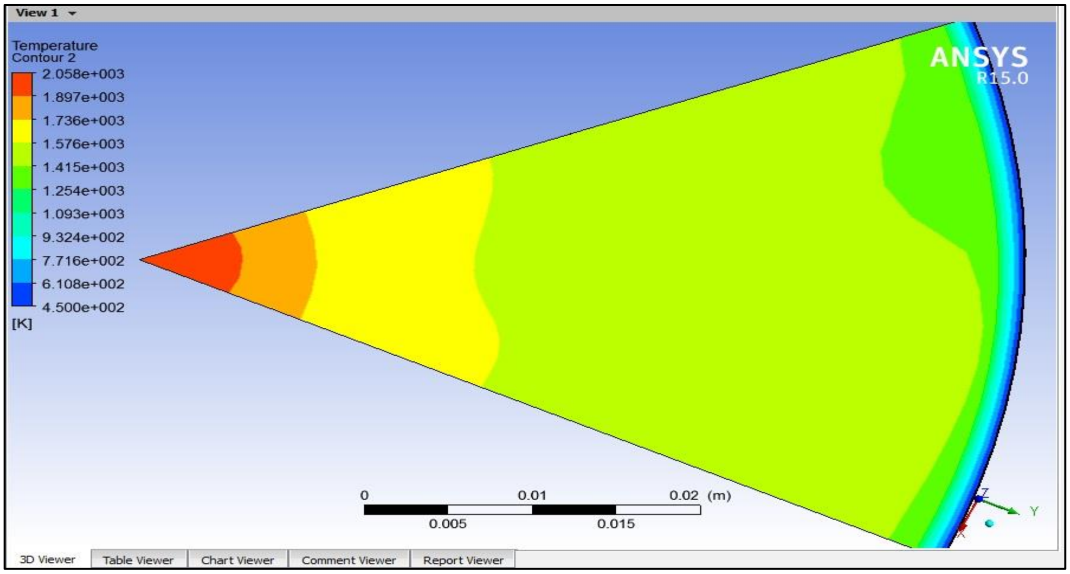


Fig.20 Grid independence results in the form of graph for simulated average temperature of liner corresponding to 372 ° and 540 ° crank angle of rotation.

From Fig. 19 and 20 it is seen that the convergence for estimated simulated temperature of combustion gases and liner from gas side at mentioned crank angle of rotation is attained at about 2,50,000 number of nodes. Initially the scale was coarser and then it became finer thereby achieving convergence. Similarly, all graphs for other estimations are drawn are there in repository. For sample basis, these are presented here.

3. RESULTS AND DISCUSSION

Results of upper mentioned simulation are as follows. Here sample of simulated temperatures are shown. (◦ Kelvin) at various crank angles. [Fig.21 to 24].The above mentioned figures shows the temperature existing along a horizontal plane in a combustion space at that particular crank angle of rotation.



.

Fig.21 Simulated temperature of in cylinder gases (Taken sector) at 364 ^{0}crank angle

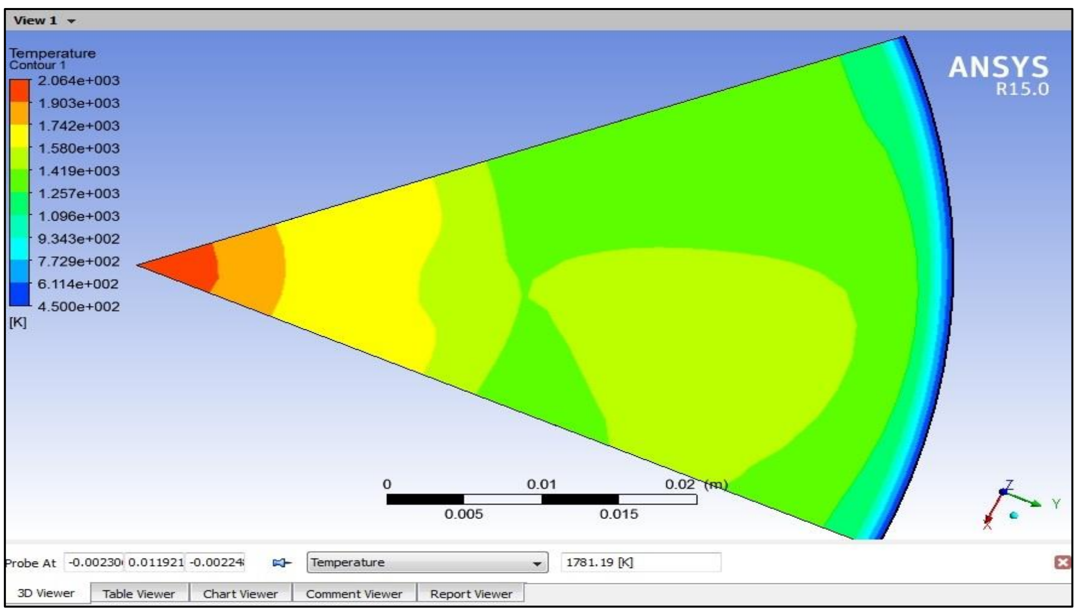


Fig.22 Simulated temperature of in cylinder gases (Taken sector) at 372 ^{0}crank angle

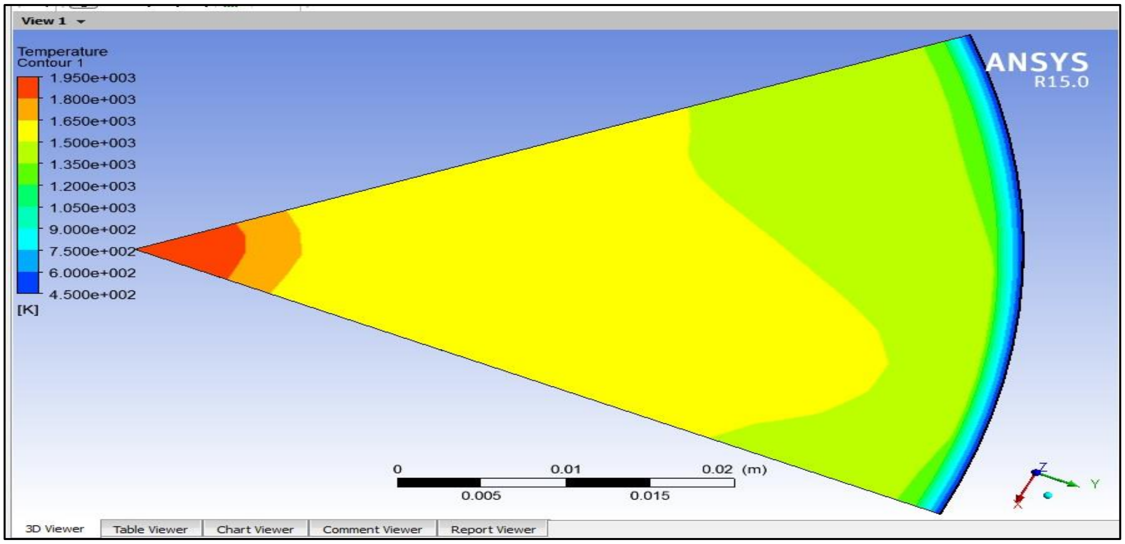


Fig.23 Simulated temperature of in cylinder gases (Taken sector) at 380 ^{0}crank angle

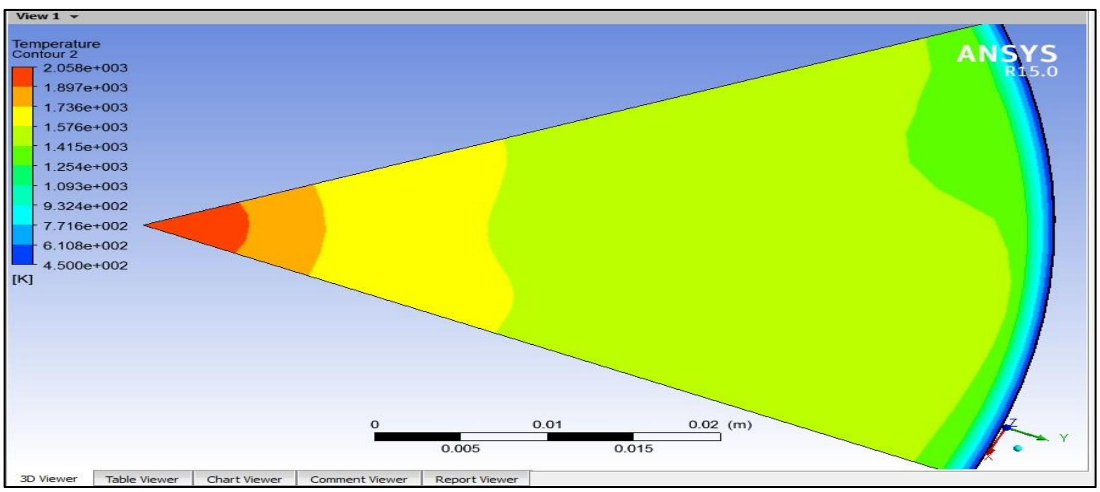


Fig.24 Simulated temperature of in cylinder gases (Taken sector) at 388 ^{0}crank angle

Fig.21 to Fig.24 show the distribution of temperature along the horizantal plane, as a sector at depicted crank angles just above piston. It is clear that the temperature are high near axis of piston and reduce eventually towards liner, along radius towards circumference. The trend is similar for all crank angles and least temperatures are seen at liner surface from gas side. The magnitude of temperature 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 s within the cylinder. Fig.25 to 30 show the temperature of all the space above the piston starting from top dead center at depicted crank angles. (45 degree sectorial angle)

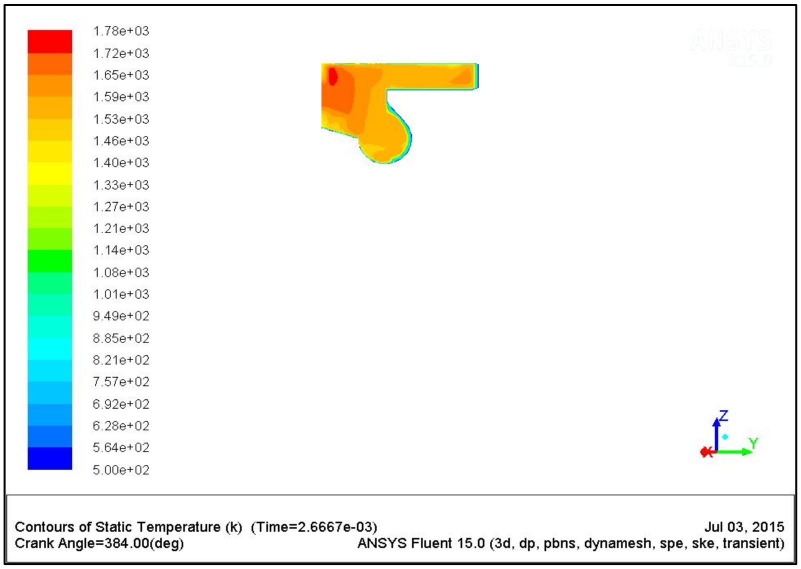


Fig.25 Temperature of gas (K) in space above piston from center of cylinder towards liner at 384° crank angle

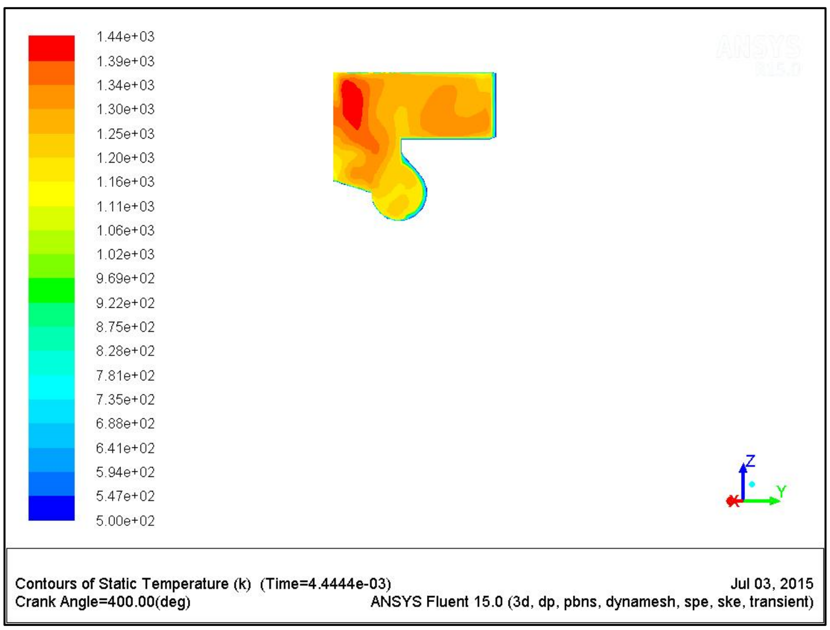


Fig.26 Temperature of gas (K) in space above piston from center of cylinder towards liner at 400° crank angle

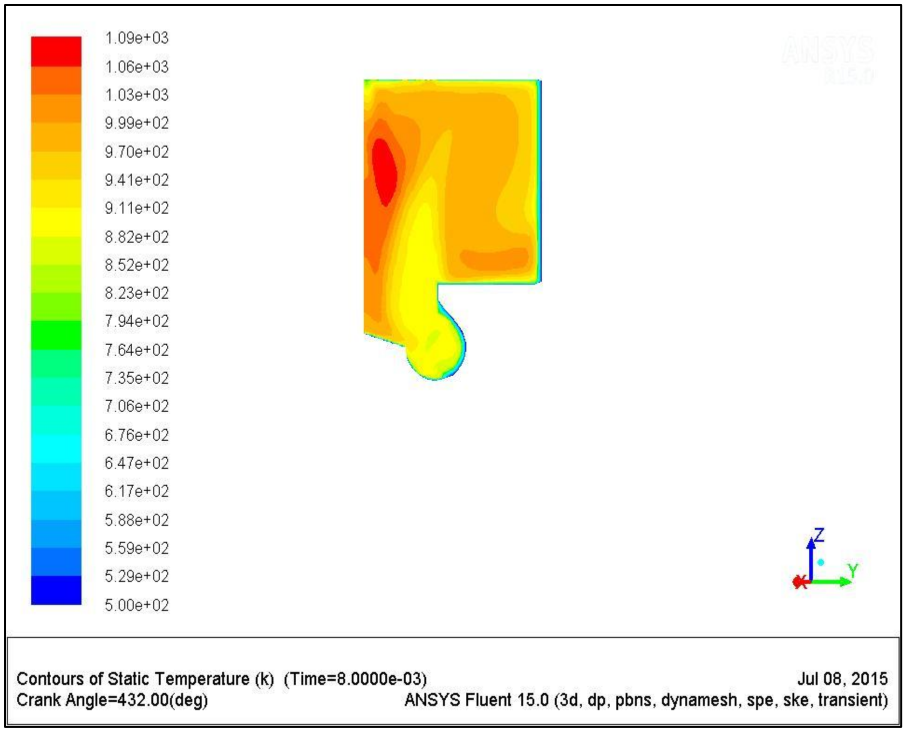


Fig.27 Temperature of gas (K) in space above piston from center of cylinder towards liner at 432.00° crank angle

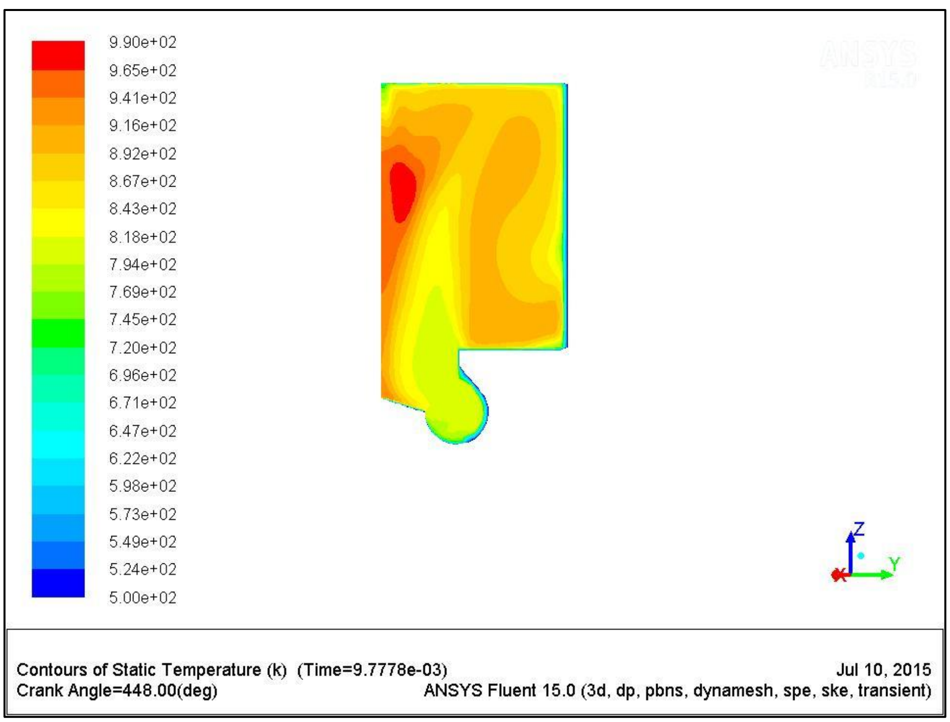


Fig.28 Temperature of gas (K) in space above piston from center of cylinder towards liner at 448° crank angle

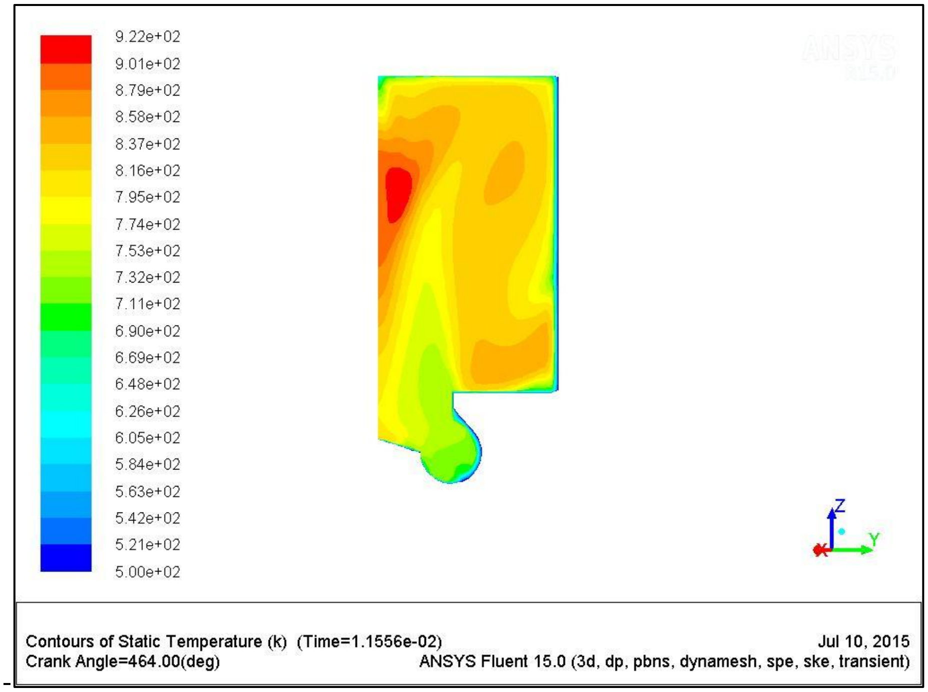


Fig.29 Temperature of gas (K) in space above piston from center of cylinder towards liner at 464° crank angle

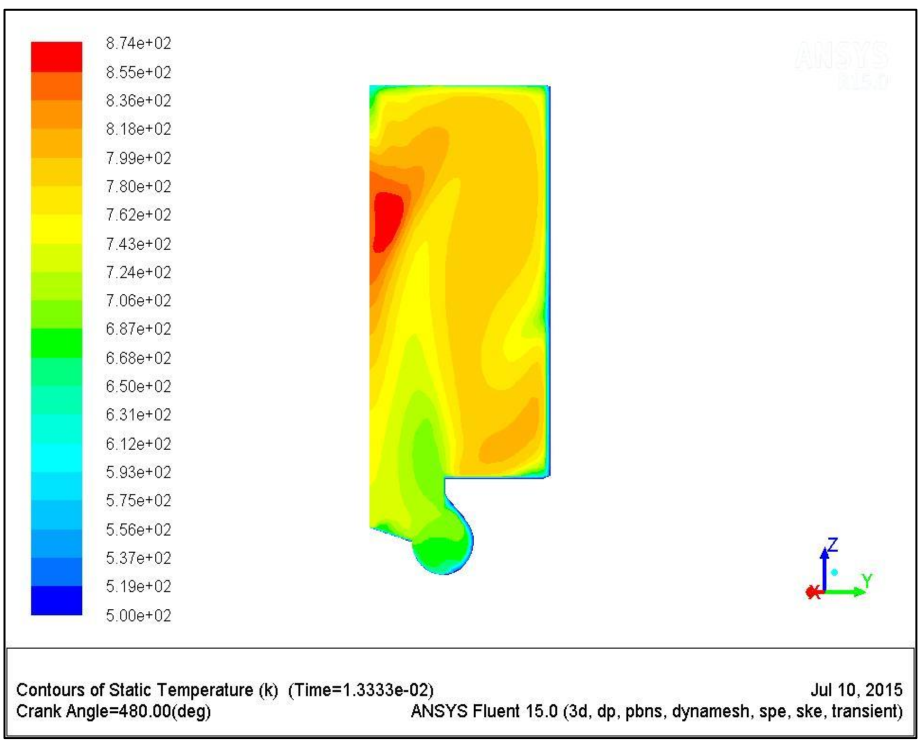


Fig.30 Temperature of gas (K) in space above piston from center of cylinder towards liner at 480° crank angle

From all the above figures, for longitudinal direction along stroke length, the in cylinder temperature reduces from top dead centre position to Bottom dead centre position .

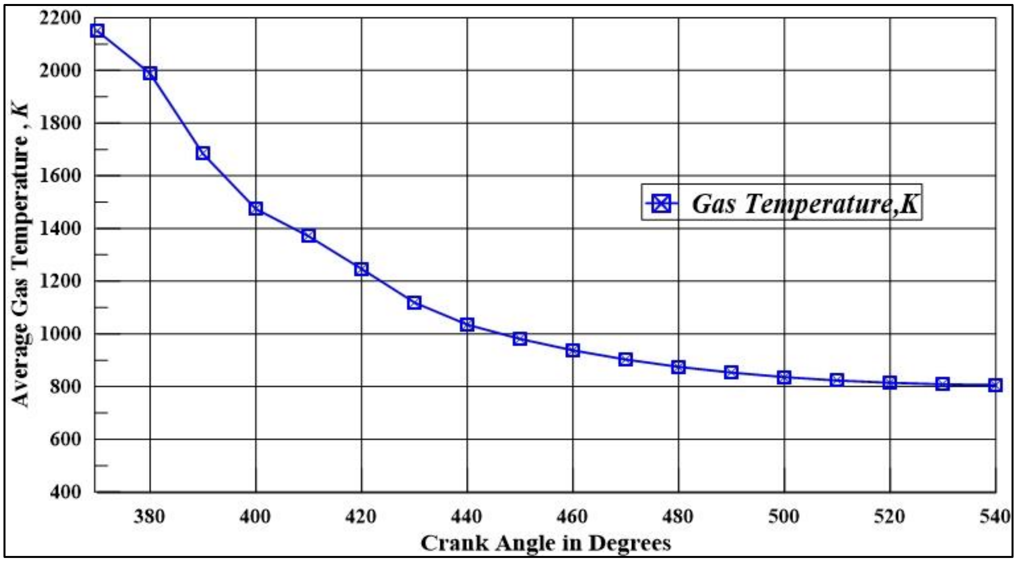


Fig.31 Variation of in- cylinder Average Temperature with Rotational Crank Angle

Above Fig.31 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 are formed, the temperature reaches to 2184 K. During the total expansion stroke when the piston comes to bottom dead center the temperature of gases reaches to 800 K. This provides temperature variation of gas in the longitudinal direction.

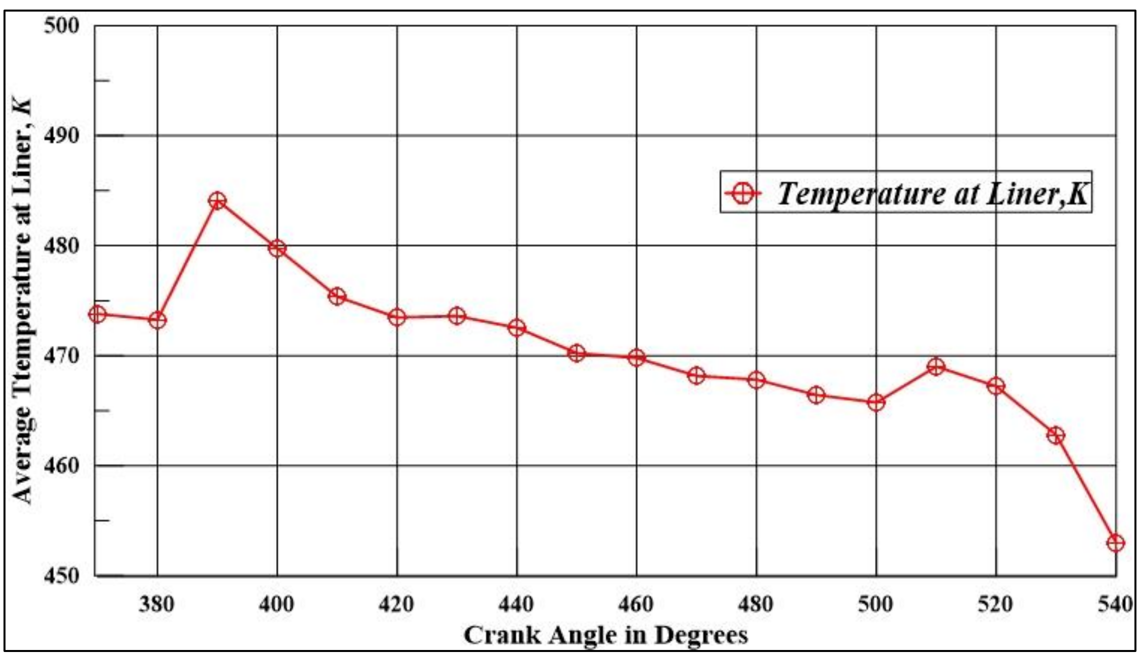


Fig.32 Variation of Liner Temperature with Crank Angle

Figure 32 displays the variation 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.The liner temperature initially increases with increase in gas temperature and eventually reduces as gas temperature also reduces with crank angle .

4. Conclusion

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 ithe same. The temperature of gas interior the cylinder is investigated numerically and further used, to determine the liner temperatures. As the diesel is injected in the combustion chamber during early part of power stroke, the preheated and compressed air gets mixed with diesel and there is combustion and maximum pressure rise is there due vigorous burning of fuel air mixture as the diesel has been already reached its ignition temperature hence, maximum temperature in the cycle is occurring during this instant. This is found to be 2183 K. 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. Due to thermal resistance offered by gas, the temperature of the liner wall further reduces 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 the liner temperatures. The results of estimation of in cylinder temperature during combustion are estimated in transient conditions. The computations, estimation and simulations are done for early part or half of power stroke as almost all heat is liberated during this. The strategy of computation can be extended for remaining part of power stroke.Here the dynamic meshing and simulation of in cylinder gases at particular crank angles is done. This can be more done in fine way in order of more steps of crank angles.

5. REFERENCES

1. A.Broatch,P.OlmedaX,MargotJ.Escalona, “New approach to study the heat transfer in internal combustion engines by 3D modelling”, International Journal of Thermal Sciences Volume 138, April 2019, Pages 405 415,DOI:https://doi.org/10.1016/j.ijthermalsci.2019.01.006

2. SAE Technical Paper, Integration of 3D Combustion Simulations and Conjugate Heat Transfer Analysis to Quantitatively Evaluate Component Temperatures,2003-01-3128, October 27, 2003 by SAE International in United States
DOI: https://doi.org/10.4271/2003-01-3128

3. SAE Technical Paper, A Numerical Investigation on Scalability and Grid Convergence of Internal Combustion Engine Simulations,2013-01-1095, April 08, 2013 by SAE International in United StatesDOI: https://doi.org/10.4271/2013-01-1095

4. Kenji Fukui, Yoshifumi Wakisaka, Kazuaki Nishikawa, Yoshiaki Hattori, Hidemasa Kosaka, Akio Kawaguchi, SAE Technical Paper 2016-01-0675, 2016,Development of Instantaneous Temperature Measurement Technique for Combustion Chamber Surface and Verification of Temperature Swing Concept 2016-01-0675,**DOI:** https://doi.org/10.4271/2016-01-0675

5. A.Torregrosa[a]P.Olmeda[a]B.Degraeuwe[a]M.Reyesb, A concise wall temperature model for DI Diesel engines, . Applied Thermal Engineering,Volume 26, Issues 11–12, August 2006, Pages 1320-1327,DOI:https://doi.org/10.1016/j.applthermaleng.2005.10.021

6. Egel Urip , Ka Heng Liew & S. L. Yang , Modeling IC Engine Conjugate Heat Transfer Using the KIVA Code, Numerical Heat Transfer, Part A: Applications -An International Journal of Computation and Methodology Volume 52, 2007 - Issue 1,1-23,DOI:https://doi.org/10.1080/10407780601112803

7. Yuanhong,LiSong-CharngKong, Coupling conjugate heat transfer with in-cylinder combustion modeling for engine simulation, International Journal of Heat and Mass Transfer,Volume 54, Issues 11–12, May 2011, Pages 2467-2478,DOI:https://doi.org/10.1016/j.ijheatmasstransfer.2011.02.015

8. C D Rakopoulos, E G Giakoumis, D C Rakopoulos, Study of the short-term cylinder wall temperature oscillations during transient operation of a turbo-charged diesel engine with various insulation schemes,2008,Research Article DOI:https://doi.org/10.1243/14680874JER00608
9. Subodh Kumar Sharma, P. K. Saini, N. K. Samria, "Experimental Thermal Analysis of Diesel Engine Piston and Cylinder Wall", *Journal of Engineering*, vol. 2015, Article ID 178652, 10 pages, 2015. https://doi.org/10.1155/2015/178652
10. Haiyan Miao & Brian Milton, Numerical Simulation of the Gas/Diesel Dual-Fuel Engine In-Cylinder Combustion Process, Numerical Heat Transfer, Part A: Applications An International Journal of Computation and Methodology,
Volume 47, Issue 6 ,523-547 ,2006,DOI:https://doi.org/10.1080/10407780590896844

11. Carmen C. Barrios, Aida Domínguez-Sáez, DévoraHormigo, Influence of hydrogen addition on combustion characteristics and particle number and size distribution emissions of a TDI Diesel Engine Fuel, 199 (2017) 162 – 168. DOI: 10.1016/j.fuel.2017.02.089

12. Fanos Christodoulou, Athanasios Megaritis,Taguchi optimization method Materials and Design, 27(2005) 853-17)

13. Fangachi Najoua, Yessari Madiha, Hajjaji Abdelowahed, 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. DOI:https://doi.org/10.1016/j.ijhydene.2013.11.124

14. 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,DOI: 10.1016/j.egypro.2013.05.002

15. Constantin Pana, Christian Nutu, Experimental aspects of the hydrogen use at diesel engine, 10th International Conference Interdisciplinary in Engineering, 2017, 649 – 657. DOI:10.1016/j.proeng.2017.02.446

16. 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. DOI: 10.4271/2012-28-0005

17. 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.

18. 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

19. Rajesh Bisane, DhananjayKatpatal, Experimental Investigation & CFD Analysis of an Single Cylinder Four Stroke C.I. Engine Exhaust System, International Journal of Research in Engineering and Technology3,6(2014), 50 – 55. DOI: 10.15623/ijret.2014.0306009
20. 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.

21. 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.DOI: http://hdl.handle.net/2440/82432