

# Simulation of engine cylinder liner temperature from coolant side

Babasaheb B.Thete<sup>1</sup>, Prof. V.W.Bhatkar<sup>2</sup>, Prof.M.R.Jagadale<sup>3</sup>

<sup>1,2,3</sup> Department of Mechanical Engineering  
<sup>1,3</sup> G.S.Moze College of Engineering, Balewadi, Pune  
<sup>2</sup> MMCOE, Karvengar, Pune

**Abstract-** Increasing computational power of modern computers, multi-dimensional Computational Fluid Dynamics (CFD) has found more and more applications in diesel engine research, design and development. Various successful applications have proven the reliability of using multi-dimensional CFD tools to assist in diesel engine research, design and development. 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 contours, emission etc. prior to experimental tests to reduce the number of investigated parameters as well as time and thus costs. A multidimensional model was created and analysis of combustion was done using FLUENT, ANSYS 15.0 package.

**Keywords-** CFD, Combustion modelling, Diesel combustion, Meshing, Simulation.

## I. INTRODUCTION

With the increasing advancement in computational power of modern computers, CFD has found its application in diesel combustion. This is now widely used by many automobile industries not only for design and analysis of engine but also for the whole vehicle analysis. Of the many types of models for engine combustion process, multidimensional computational fluid dynamics (CFD) models is gaining momentum due to its capability to predict the gas flow patterns, fuel spray structure etc.

The ANSYS workbench has been used to determine the approximate temperatures of liner from gas side and coolant side. Here only the power stroke is considered for combustion CFD as the engine is four stroke, 4cylinder, diesel engine and temperatures are maximum in power stroke.

## II. OBJECTIVE OF PROJECT

- i) Approximate estimation of the gas temperatures
- ii) Approximate estimation of liner temperatures from gas side and coolant side.
- iii) Estimation of coolant temperature along coolant gallery.

## III. COMPUTATIONAL PROCEDURE

The combustion simulation of compression ignition engine was developed using Fluent software (ANSYS 15.0 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.

## IV. STEPS INVOLVED IN COMBUSTION CFD

### A. ICE-Solver

The inbuilt ICE system of ANSYS workbench has been employed for combustion simulation. The In-Cylinder Combustion Simulation type is used here.

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 modeled as a sector to speed up the calculation.

TABLE I  
 DETAILS OF INPUT GIVEN FOR STARTING  
 SIMULATION TO ICE- SOLVER

Sr. No.	Parameter	Value/Details
1	Crank Shaft speed	1500 RPM
2	Crank rotation period	720 °
3	Crank radius	65mm
4	Connecting rod length	207 mm

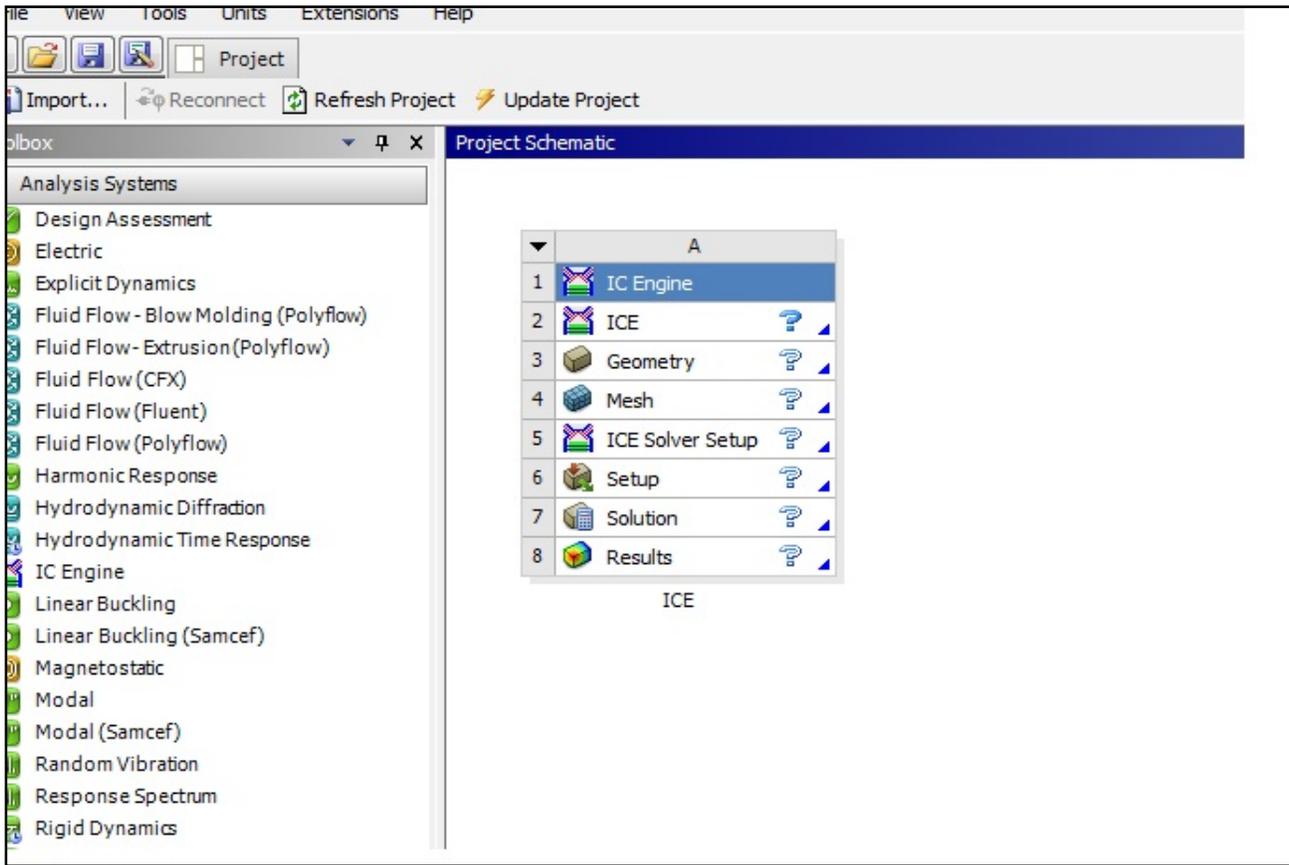


Fig.1. Module selected for simulation, i.e. ICE module.

**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 is drafted. It consists of the hollow space on piston bowl, and the space above it. Included angle of sector is 45° (Fig.2)

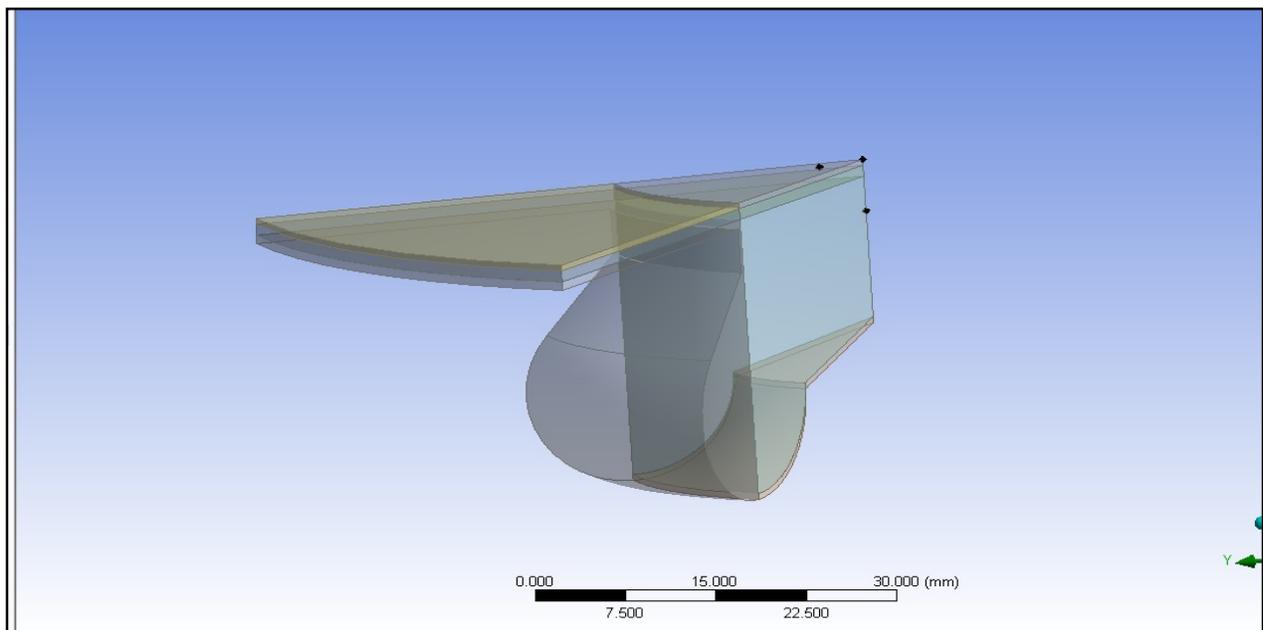


Fig.2 Drafted geometry for combustion simulation

### C. Geometry Clean up

In Fig.2 unwanted surfaces are removed. Total geometry is divided into 7 parts.

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

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)

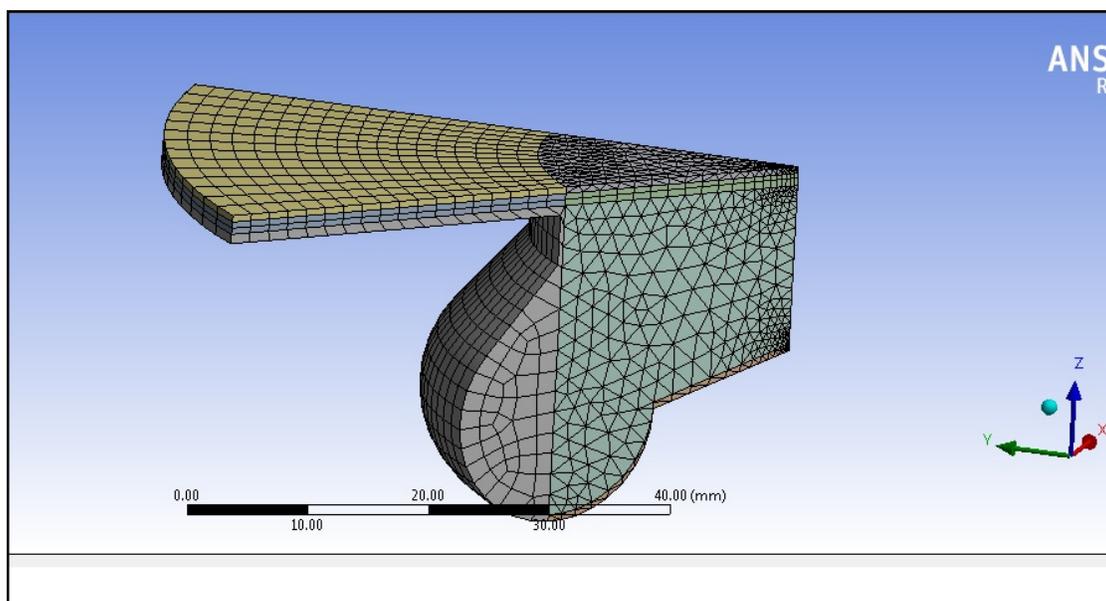


Fig.3 Mesh of combustion space.

### E. ICE Solver set up

It consists of six steps

#### i) Basic setting

In this solution type, combustion solution type is decided. Initialization of flow is done.

Crank angle to run : 180°

Swirl number : 2.1

Other standard or default inputs are given.

#### ii) Physical Set up

Chemical features such as material input is selected from FLUENT (Table 4)

TABLE II  
DETAILS OF INPUT GIVEN FOR PHYSICAL SET UP

Sr. No.	Parameter	Value/Details
1	Pre Combustion mixture	Diesel and air
2	Start of injection	345° crank angle
3	End of injection	368° crank angle
4	Total injection flow rate	1.54x10 <sup>-3</sup> Kg/sec
5	Evaporating species	n-heptane (C <sub>10</sub> H <sub>22</sub> )

iii) Boundary conditions

Boundary conditions such as outlet pressure, inlet pressure, mass flow rate, outlet are decided.

iv) 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.

v) Initialization

All properties are gauge pressure at inlet condition, temperature, turbulent kinetic energy, flow velocity, dissipation rate etc.are given.

vi) Post processing

Here images of various properties at planes are taken.

**F. Set up- FLUENT is used for computation**

It is carried out in following steps. [1 to 9]

- i) Double precision method is used to make results fine. Set up and solution is carried out in FLUENT.
- ii) Absolute velocity formulation is used.
- iii) K- $\epsilon$ MODEL is used (2 equation model).For cylinder wall treatment standard cylinder functions are used.
- iv) Cell zone condition is taken as fluid.
- v) Dynamic mesh has been used.
- vi) Run calculation-No of time steps decided-720, Maximum iterations per time step-5

Calculations are run and results are obtained.

**V. RESULTS AND DISCUSSIONS**

All calculated values are observed in CFD post processing.

**A. Combustion Gas Temperatures**

As maximum heat is released and temperatures of gas are maximum during early part of power stroke. Sample simulated results at crank angles 364°, 372°, 380°, 388°are shown.

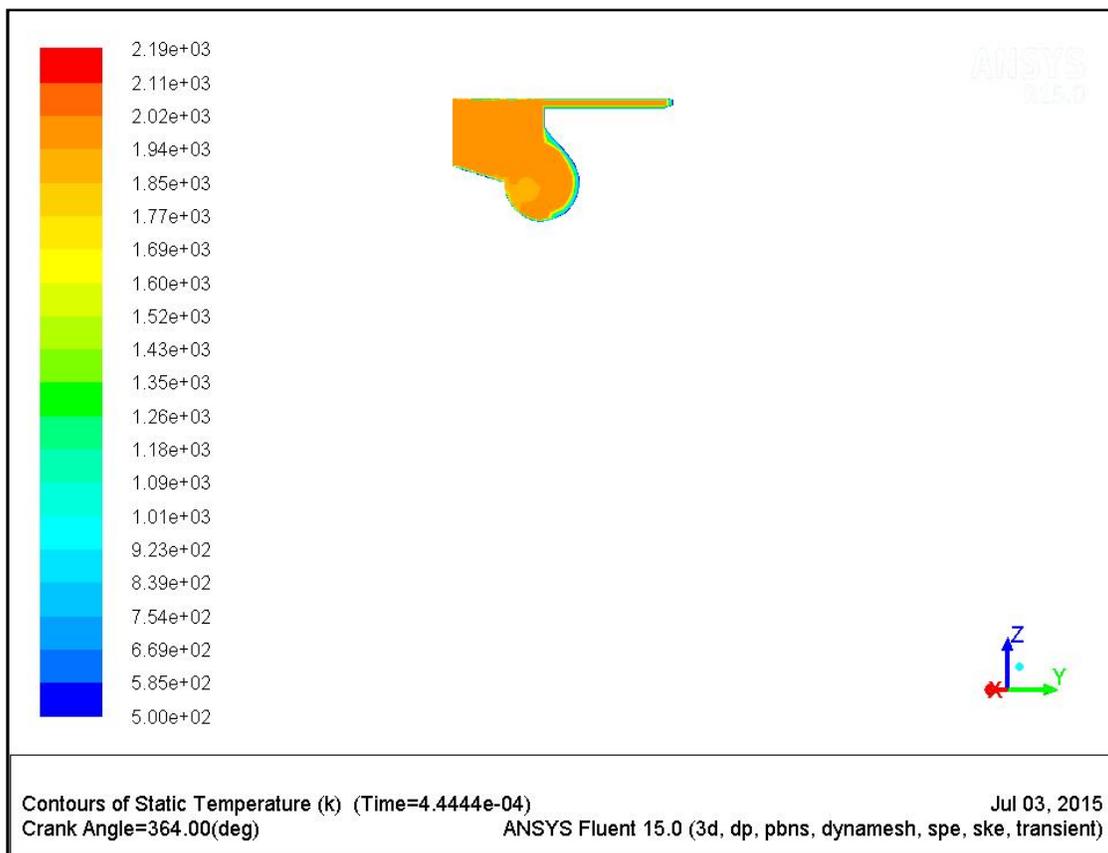


Fig.4 Temperature of gas (°K) from centre of cylinder towards liner at 364°Crank angle.

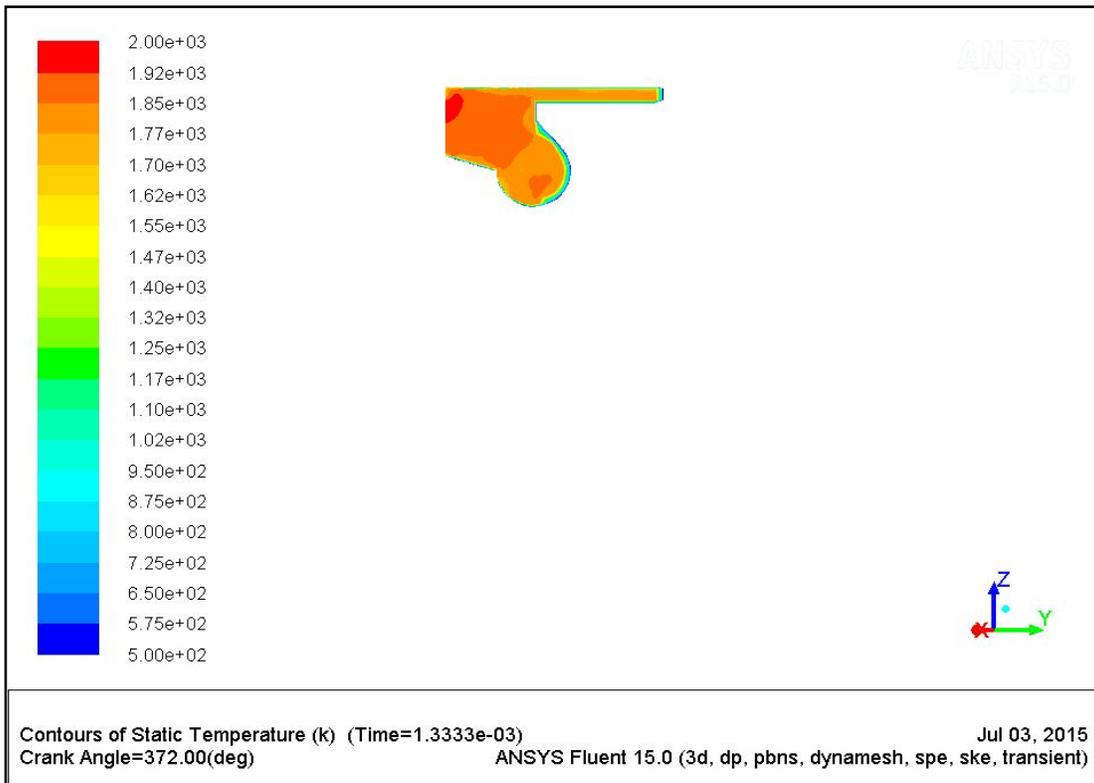


Fig.5 Temperature of gas (°K) from centre of cylinder towards liner at 372°Crank angle.

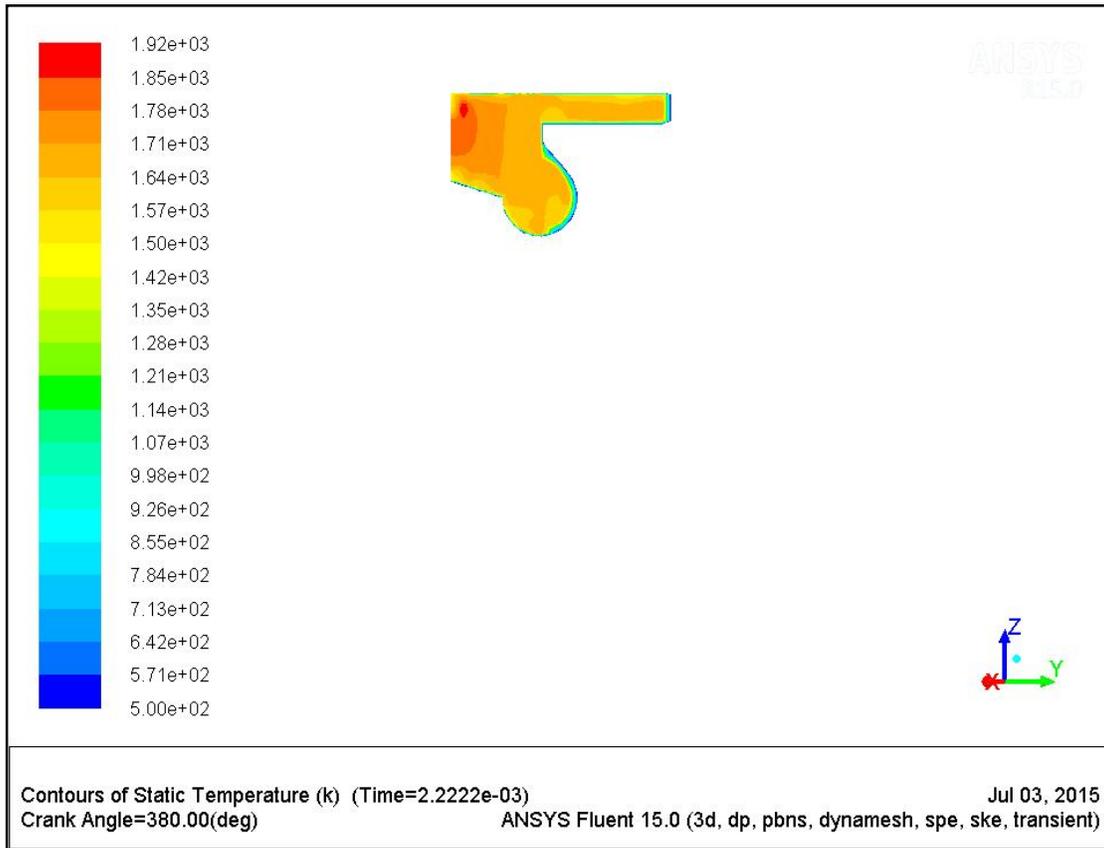


Fig.6 Temperature of gas (°K) from centre of cylinder towards liner at 380°Crank angle.

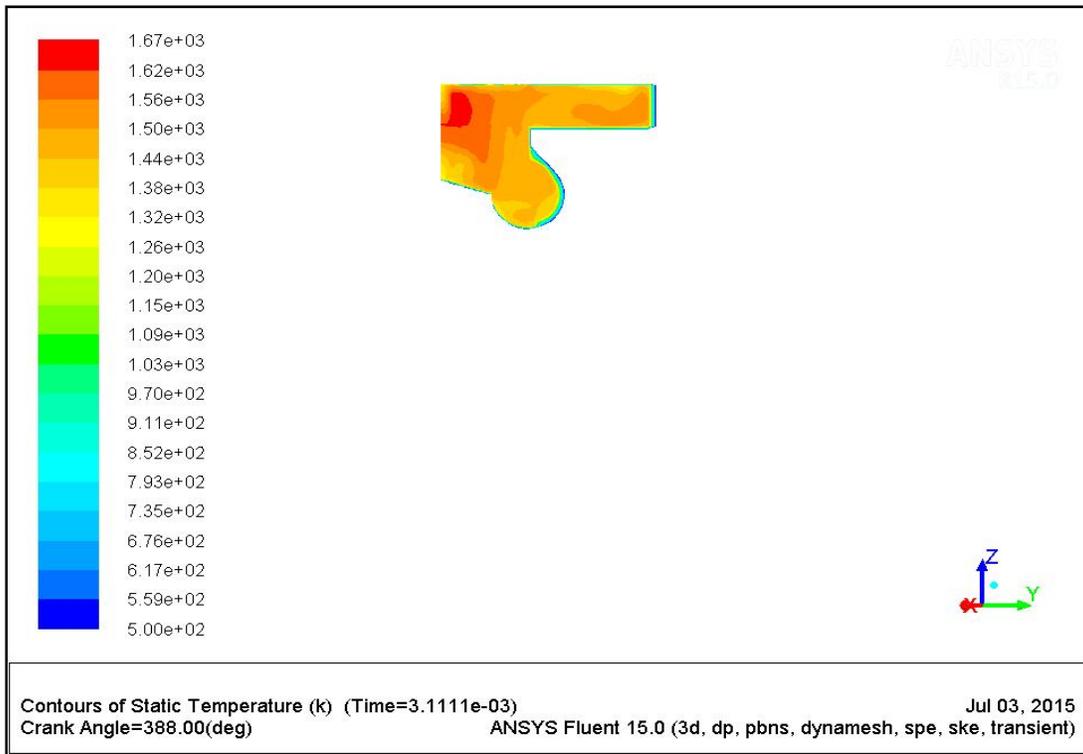


Fig.7 Temperature of gas (°K) from centre of cylinder towards liner at 388°Crank angle.

**B. Cylinder Liner Temperatures from Gas side**

Sample simulated results at crank angles 364°, 372°, 380°, 388°are shown.

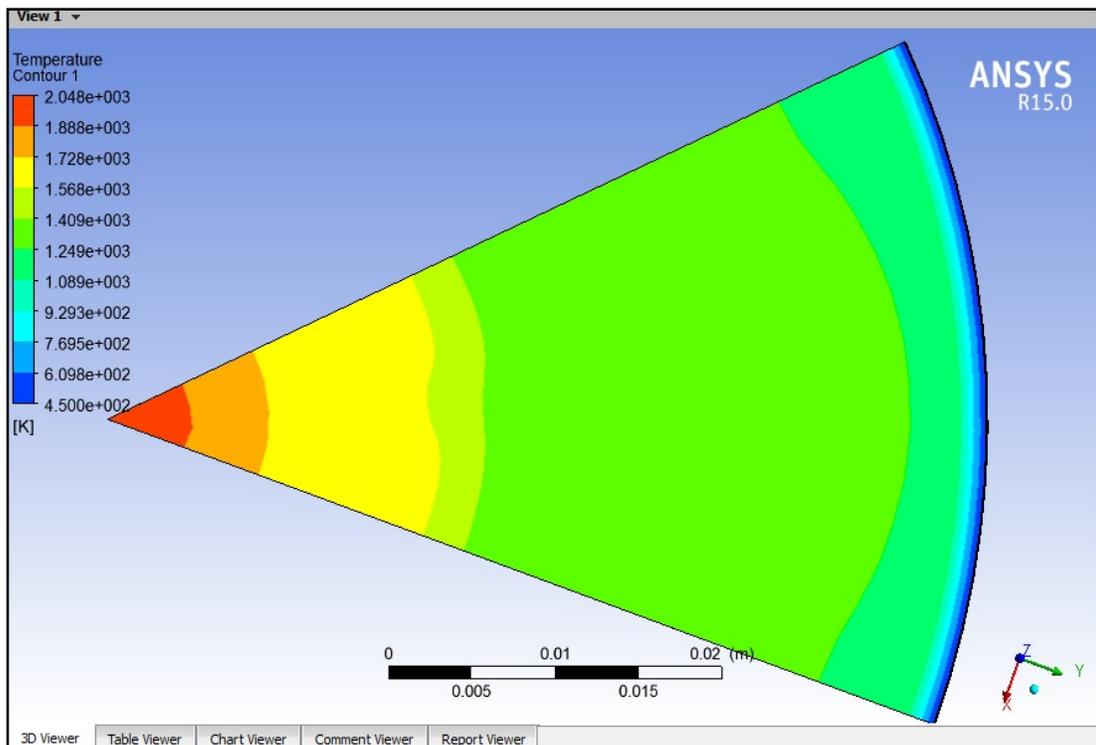


Fig.8 Simulated Temperature from centre of cylinder to liner wall from gas side along horizontal plane at 364°Crank Angle

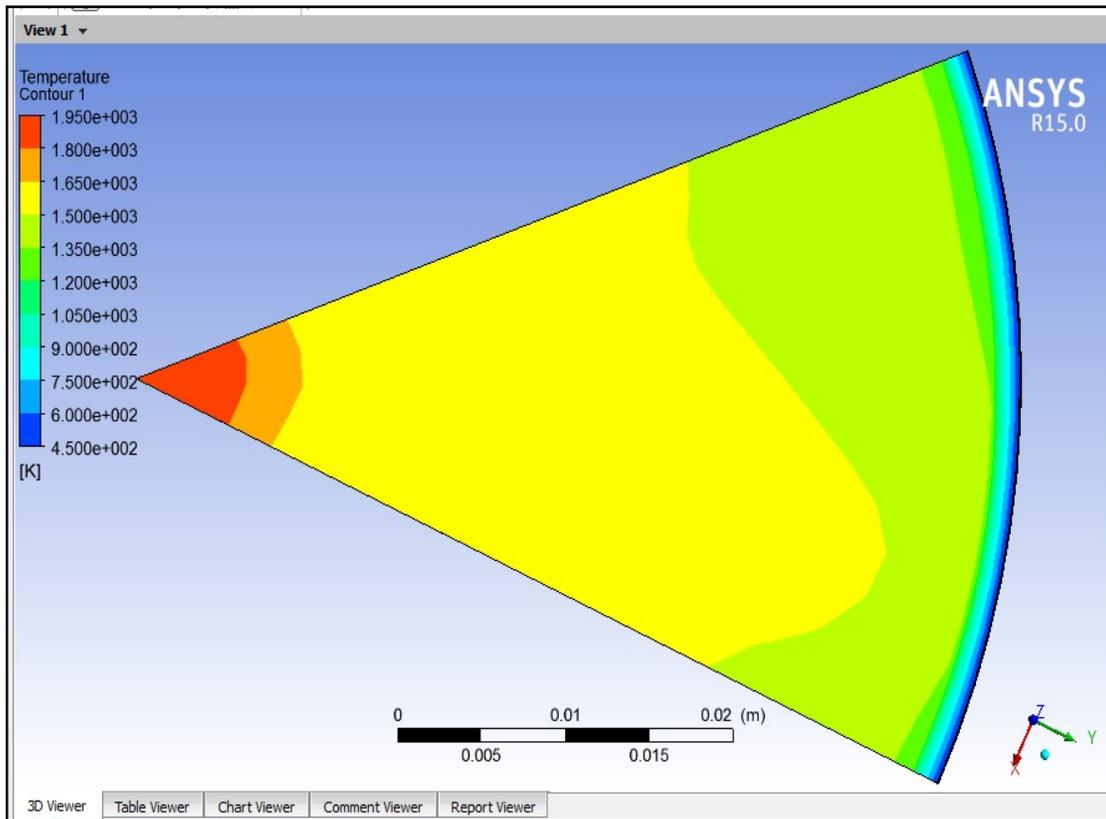


Fig.9 Simulated Temperature from centre of cylinder to liner wall from gas side along horizontal plane at 372°Crank Angle

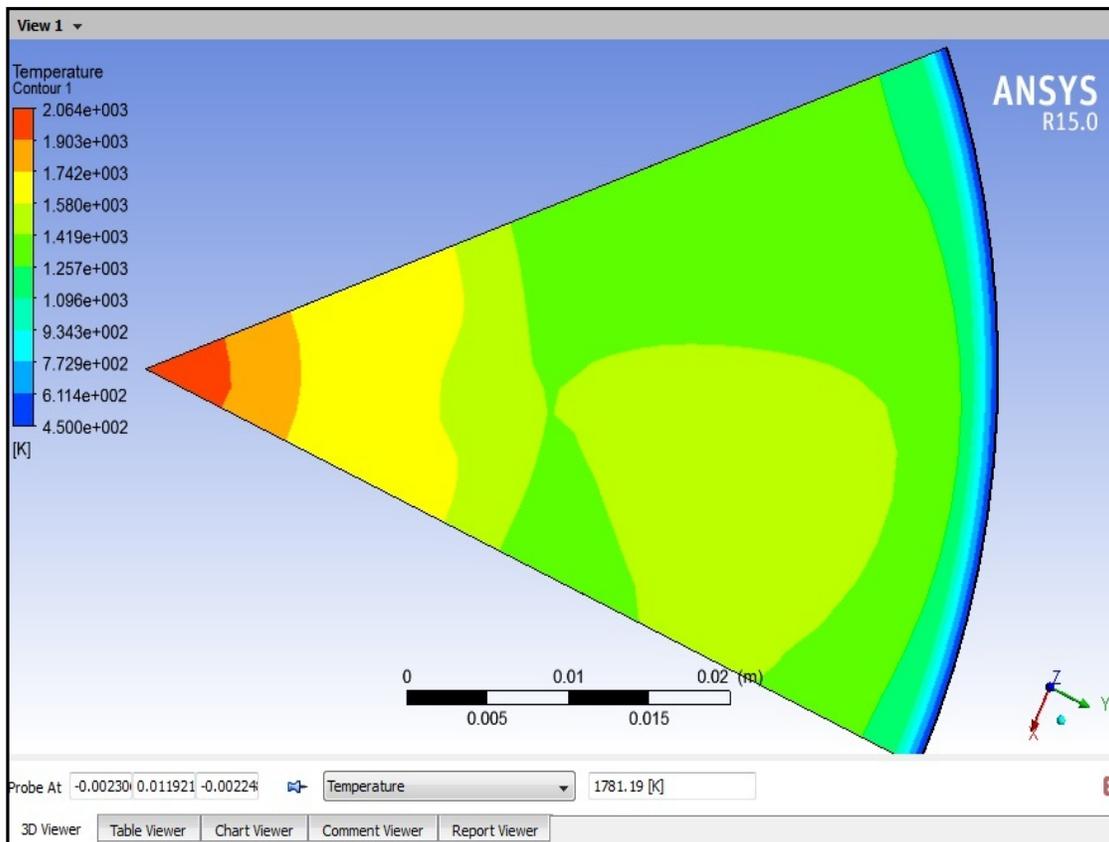


Fig.10 Simulated Temperature from centre of cylinder to liner wall from gas side along horizontal plane at 380°Crank Angle

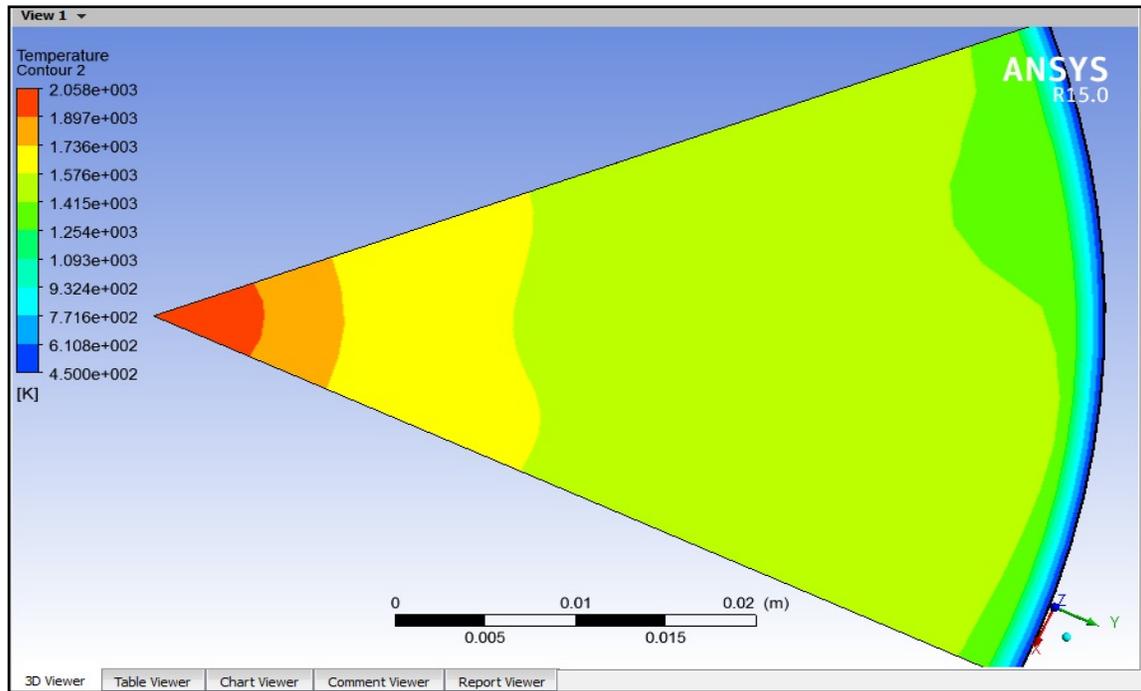


Fig.11 Simulated Temperature from centre of cylinder to liner wall from gas side along horizontal plane at 388°Crank Angle

### C. Cylinder Liner Simulation[1to9]

The approximate estimated temperatures of liner from gas side from prior combustion simulation are given as input for liner simulation. As in prior simulation, the geometry creation, meshing, boundary conditions, solver set up for liner simulation or modeling is done. All related inputs and boundary conditions are given and processed for liner temperatures from coolant side. Also coolant is simulated simultaneously. The metallic liner and fluid coolant are modeled simultaneously. (Fig.12 to Fig.14)

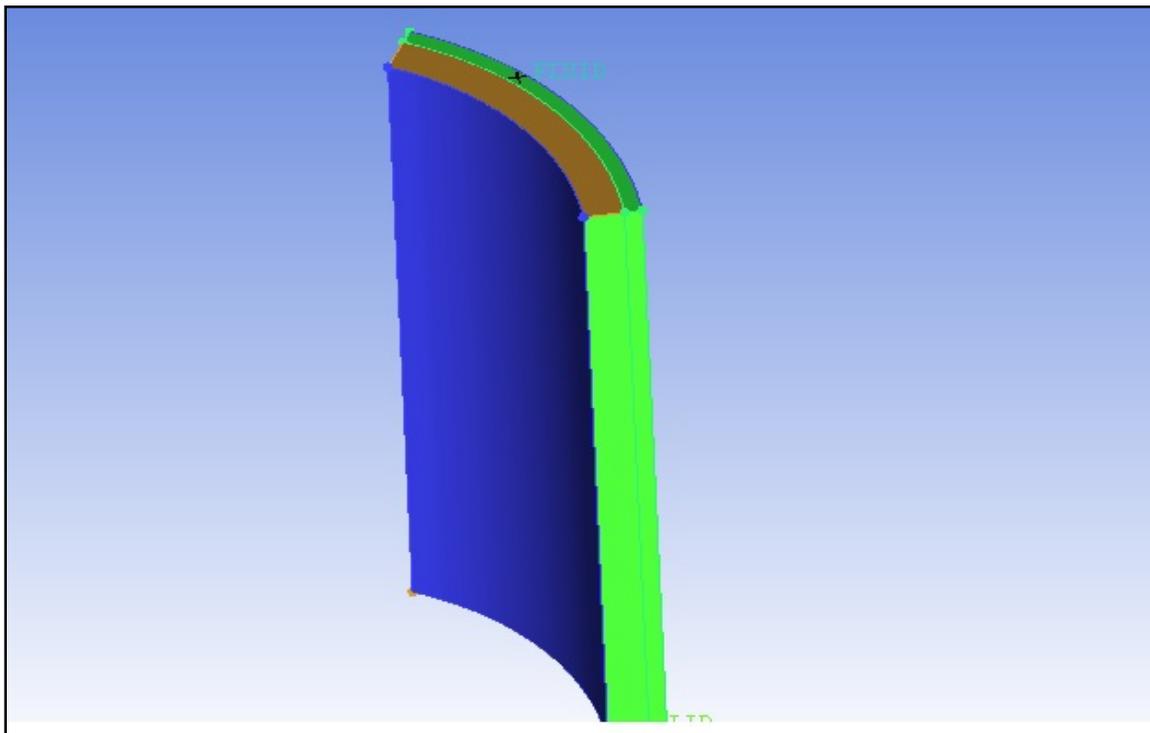


Fig.12 Geometry of liner and coolant modeling

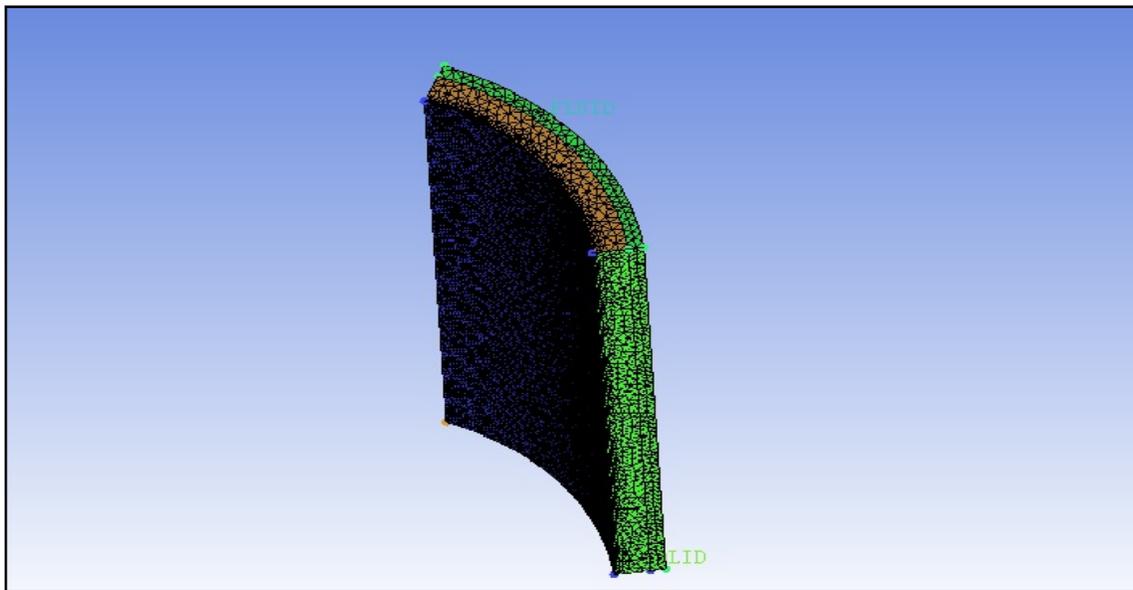


Fig.13 Meshing of liner and coolant modeling

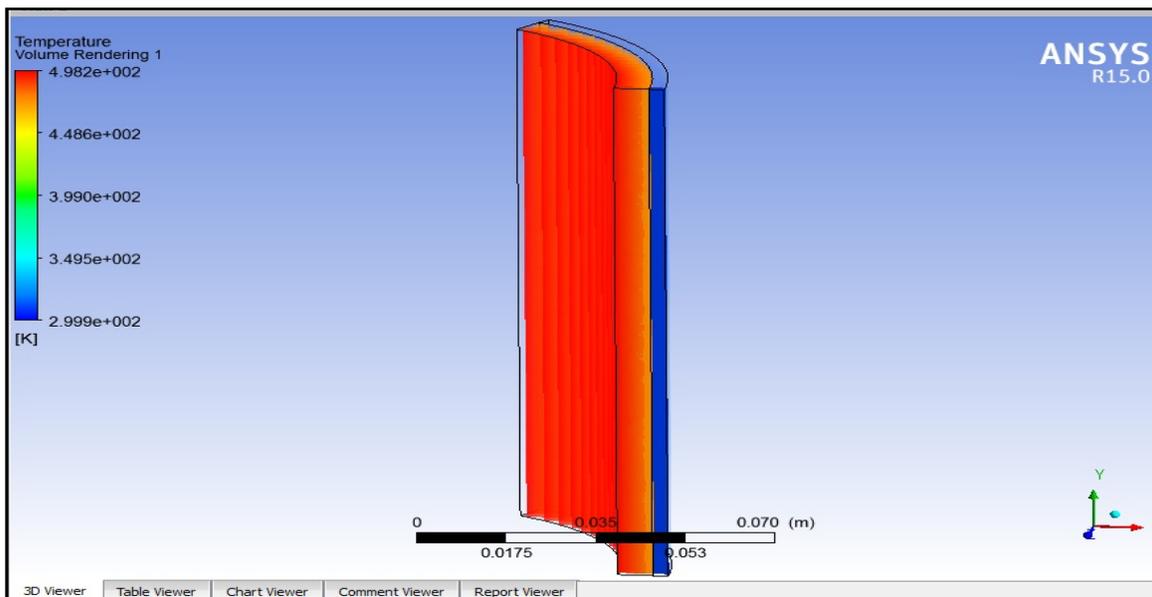


Fig.14 Estimated temperature across liner

## VI. CONCLUSION

- i) The simulation and modeling of combustion of selected diesel engine is done by using ANSYS 15.0 workbench.
- ii) The power stroke is taken for simulation as the maximum temperatures in the cycle occur during power stroke.
- iii) The in cylinder gas temperatures are approximately estimated
- iv) The results of gas temperatures are fed to determine the liner temperatures. The liner temperatures from gas side are also estimated.
- v) Once the gas side temperatures are known, these are given as input to determine the liner temperatures from coolant side. The liner along with coolant gallery is modeled and

- simulated for determination and estimation of various temperatures. The conductive heat transfer model is employed across the liner.
- vi) The temperatures of liner from coolant side are estimated (Fig.14).

## REFERENCES

- [1] Umakant V. Kongre, Vivek 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, Issue No 3, 2010

- [2] M.J.Lewis, J.B.Moss And P.A. Rubini, “CFD Modelling of Combustion and Heat Transfer in Compartment Fires”, School of Mechanical Engineering, Cranfield University, Bedford MK43 OAL, UK,1997
- [3] Tutorial Manual Guide, “Combustion Engine ANSYS Mesh Tutorial
- [4] Rajesh Bisane1, Dhananjay Katpatal, “Experimental Investigation and CFD Analysis of an Single Cylinder Four Stroke C.I. Engine Exhaust System”, International Journal of Research in Engineering and Technology , Volume: 03, Issue: 06 , June 2014
- [5] Pathak Yogesh , Deore Kailas D, Patil Vijayendra, "In Cylinder Cold Flow CFD Simulation of IC Engine Using Hybrid Approach", International Journal of Research in Engineering and Technology , Volume: 03 Issue: 06 , June 2014
- [6] H.Sushma and Jagadeesha.K.B, “CFD Modeling of the in cylinder flow indirect injection diesel engine” International Journal of Scientific and Research Publications, Volume 3, Issue 12, December 2013.
- [7] Divyanshu Purohit, Pragya Mishra, Vishwanath Banskar, “ Flow Simulation of an I.C. Engine in FLUENT, ANSYS 14.0” International Journal of Engineering Research and Applications, International Conference on Emerging Trends in Mechanical and Electrical Engineering (ICETMEE) ,13th-14th March 2014
- [8] J.J.M. Smits,” Modeling of a fluid flow in an internal combustion engine”, Eindhoven University of Technology, Department of Mechanical Engineering, Division Thermo Fluids Engineering, 2006.
- [9] Roberto Castilla López, David Del Campo Sud, “Study of Mesh Deformation Features of an Open Source CFD Package and Application to a Gear Pump Simulation”, 2012