International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 3, Issue 2, August 2014)

Combustion Analysis of Inverted M Type Piston CI Engine by Using CFD

Dr. Hiregoudar Yerrennagoudaru[1], Manjunatha K[2], Chandragowda M[3], Ravi Kumar K J[4]

[1]*Professor and PG Co-ordinator (Thermal Power Engineering), Mechanical Engineering Department, RYMEC Bellary, Karnataka, India*

[2,3]*Asst Professor,* [4]*M.Tech, (Thermal Power Engineering), Mechanical Engineering Department, RYMEC Bellary, Karnataka, India*

***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 14.5 package.**

***Keywords*: CFD, Combustion modeling, Diesel combustion, Meshing, Simulation.**

I. INTRODUCTION

Reduced fuel consumption and improved efficiency are always the primary area of consideration in internal combustion engine design sector. They were generally done by building prototypes and testing them. But this traditional process always had several limitations. They were time consuming, cost consuming and also a prototype made for a single purpose didn't had the complete versatility of using the same for other purpose. This difficulty can be overcome by using CFD studies. 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.

Development of any internal combustion engine is driven primarily by fuel efficiency and emission requirements. This requires refinement of the in cylinder flow, mixture formation and combustion processes.

The use of Computational Fluid Dynamics (CFD) along with optimization tools can help shorten the design optimization cycle time. Traditional approach of experiments using flow bench testing is very costly as well as time consuming. Moreover CFD allows insight into the minute flow details which otherwise are not capture using flow bench tests.

Understanding the nature of the flows and combustion in internal combustion engines are important for improving engine performance. The flows in IC engines can be characterized by swirl, tumble and compression in the cylinder. This flow motion has a strong influence on the engine combustion process and hence on the engine emission of pollutants. Recently simulation results by Computational Fluid Dynamics codes are used in the development and optimization of new engines by car manufacturers (automotive industry). The in-cylinder fluid motion in internal combustion engines is one of the most important factors controlling the combustion process. Swirl and tumble are well known approaches for in-cylinder flow enhancement. Swirl and tumble are generated in the intake stroke as a result of the inlet port shape and orientations.

II. OBJECTIVE OF THE PROJECT

Modeling the engine with inlet manifold

- Effect of inlet manifold configurations on the in-cylinder flow
- Study the cold simulation flow characteristics inside the IC engine cylinder equipped with piston configurations for complete four stroke using Fluent dynamic mesh model.
- Perform CFD Simulation of the IC engine with inlet valve and intake manifold and piston using dynamic mesh approach.
- Study the effect of piston head configurations on the in-cylinder flow-(only intake and compression stroke)
- Compare effect of different piston head configurations turbulence, swirl and tumble ratio in the engine.

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 3, Issue 2, August 2014)

III. Computational Procedure

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

IV. Cold Flow Simulation

Geometric Model Creation

Geometries can be created top-down or bottom-up. Top-down refers to an approach where the computational domain is created by performing logical operations on primitive shapes such as cylinders, bricks, and spheres. Bottom-up refers to an approach where one first creates vertices (points), connects those to form edges (lines), connects the edges to create faces, and combines the faces to create volumes. Geometries can be created using the same pre-processor software that is used to create the grid, or created using other programs (e.g. CAD, graphics). Geometry files are imported into HM to create computational domain. The Extracted fluid domain of IC engine as shown in fig.

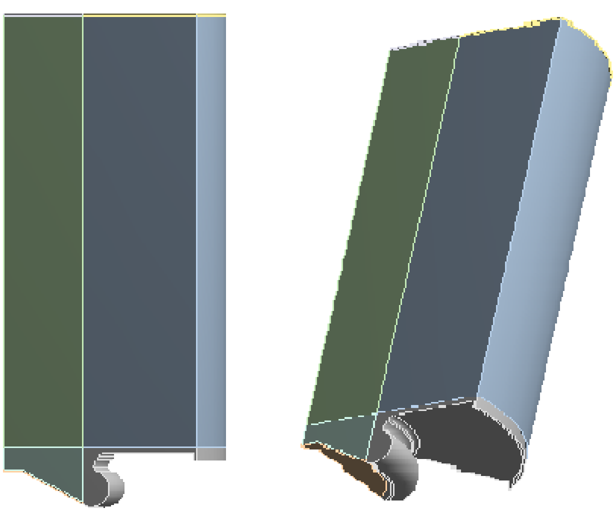

Fig. 1 Combustion Analysis Sector Model

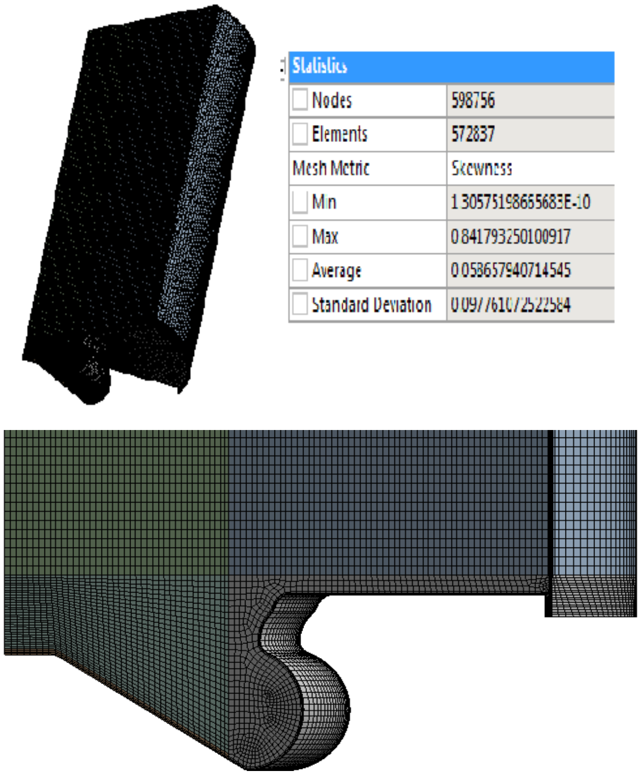

Fig.2. Combustion Simulation Mesh

V. Governing Equations

The basic approach of in-cylinder diesel combustion models are typically compressible turbulent flow. Apart from the complexity of turbulent model, the high pressure spray and resulting spray penetration, evaporation, and involvement of multiphase, multi-component nature only increases its complexity. Even then the nature of fluid is still governed by the basic equations including continuity (mass conservation), momentum (Navier-Stokes equation), energy and turbulence (k-ε model) equations. Of the three combustion modelling (thermodynamic, multidimensional and phenomenological modelling) the software use multidimensional modelling, i.e. CFD modelling

VI. Boundary Conditions

Following are the assumptions incurred on the present analysis:

- Flow is Turbulent
- Flow is Transient and incompressible
- Segregated solver

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 3, Issue 2, August 2014)

The decomposition and zone name matching explained in Fig. 6.8 and Fig. 6.9 contains a sketch of the decomposition and the corresponding zone names. At the inlet, pressure boundary condition is applied. The engine walls are defined as stationary no slip walls.

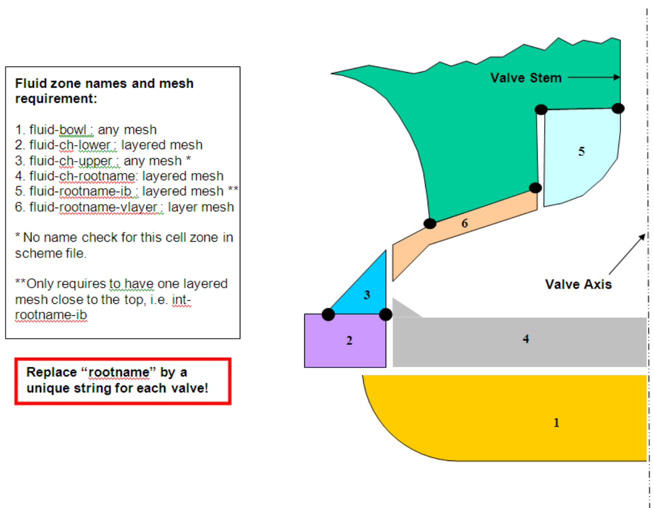

Fig.3. Fluid zone names and mesh requirement

VII. Convergence Criteria

The iterative process is repeated until the change in the variable from one iteration to the next becomes so small that the solution can be considered converged.

At convergence:

- All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance.
- The solution no longer changes with additional iterations.
- Mass, momentum, energy and scalar balances are obtained.

Residuals measure imbalance (or error) in conservation equationsThe convergence of the simulations is said to be achieved when all the residuals reach the required convergence criteria. These convergence criteria are found by monitoring the in the drag. The convergence criterion for the continuity equation is 1E-4 and it is set to 1E-3 for the momentum, k and ω equations. The convergence of the residuals is shown in Fig.

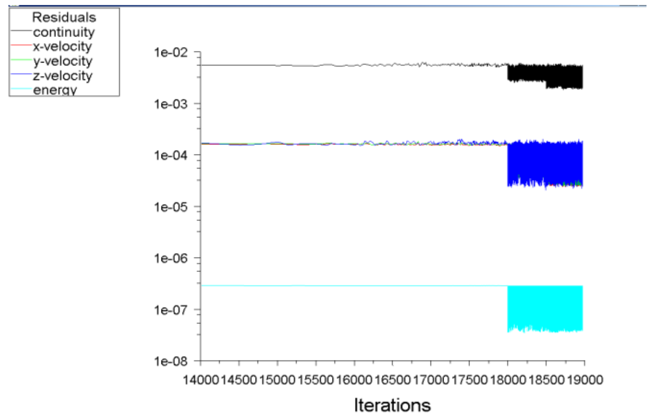

Fig.7. Convergence Criteria

VIII. Combustion Setup

A 60 degree simplified sector model of a4-stroke diesel engine which corresponds to one fuel injector hole is modeled. Since the objective is to model compression and combustion stroke, the actual simulation starts at 233 CA (i.e. IVC). Since the meshed model corresponds to TDC condition, initial part of the tutorial describes how to setup dynamic mesh model and perform mesh motion to get mesh at 233 CA. The simulation is performed only upto 400 CA although the exhaust valve closes much later after this point.

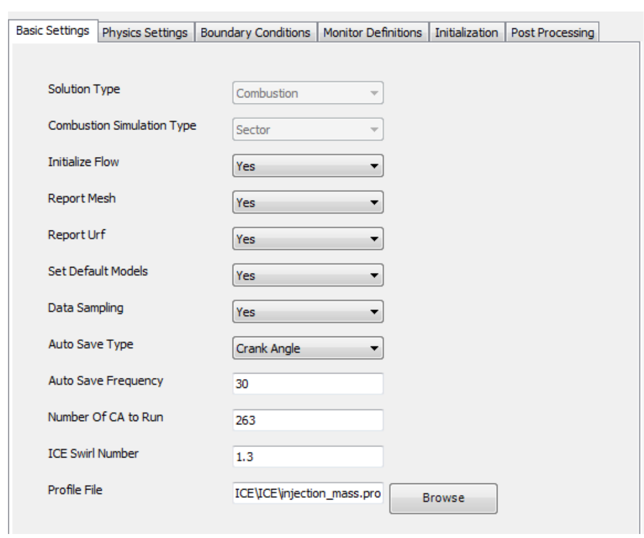

Fig.8. Combustion Setup

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 3, Issue 2, August 2014)

IX. Results And Discussions

The maximum cycle temperature obtained was near 1700K the temperature contours, the particle traces and velocity contour generated by the software as shown in figs, are approximate with the experimental values.

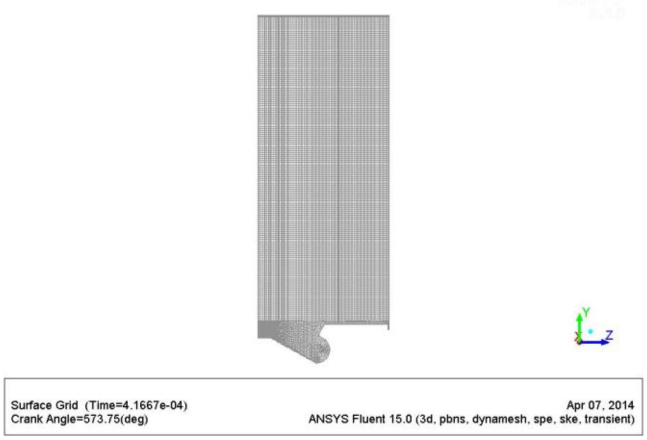

Fig.9. Dynamic Mesh Movement CA 611 degree

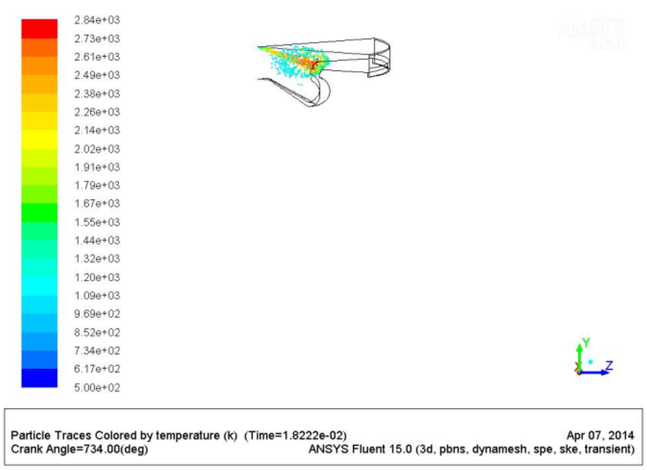

Fig.10. Diesel Particle colored by Temperature at 734 degree

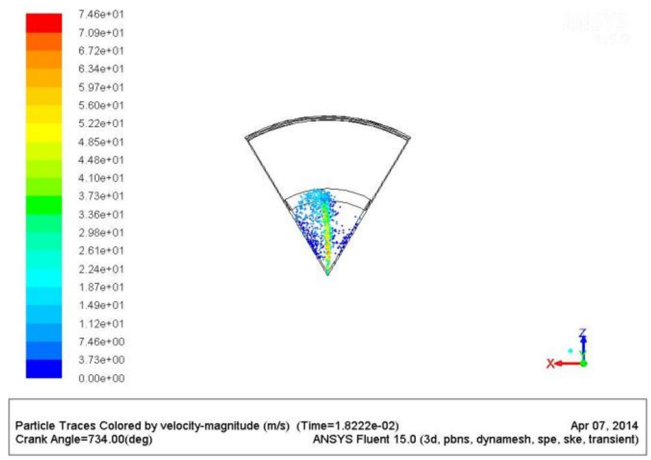

Fig.11. Diesel Particle colored by velocity at 734 degree

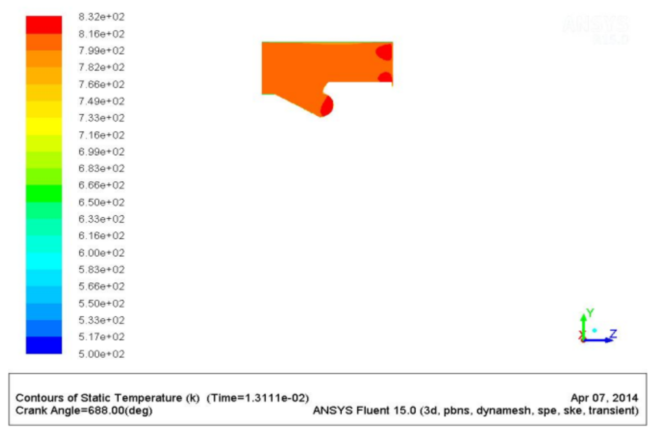

Fig.12. Temperature contours at 688 degree

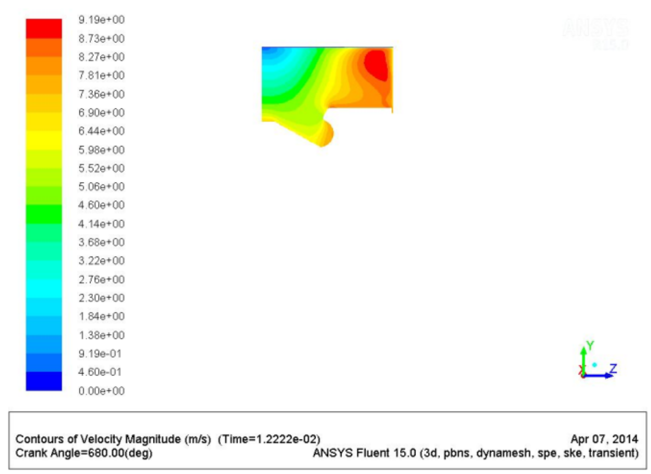

Fig.13.Velocity contours at 676 and 680 degree

International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 3, Issue 2, August 2014)

X. Conclusion

Development of any internal combustion engine is driven primarily by fuel efficiency and emission requirements. This requires refinement of the in-cylinder flow, mixture formation and combustion processes. Design optimization of the intake/exhaust port, valves and piston bowl is essential to realize the above mentioned requirements. The use of Computational Fluid Dynamics (CFD) along with optimization tools can help shorten the design optimization cycle time. Traditional approach of experiments using flow bench testing is very costly as well as time consuming. Moreover CFD allows insight into the minute flow details which otherwise are not capture using flow bench tests. Air motion inside the intake manifold is one of the important factors, which govern the engine performance and Development of any internal combustion engine is driven primarily by fuel efficiency and emission requirements. This requires refinement of the in-cylinder flow, mixture formation and combustion processes. In this simulation IC engine model is created using ANSYS Design modeller and simulation is carried out using ANSYS Fluent. The Discrete Phase Model is used for simulating fuel injection, evaporation, and droplet boiling. The simulation model developed successfully captures the single cylinder diesel engine operating with Hexane as fuel. Result was noted with different crank angles at different flow time. It is found Maximum and Minimum Volume-Average static pressures, Mass-Average static Temperatures at different flow time, Predicted diesel Mass fraction Distribution and Predicted Static Temperature Distribution. The operational range of the model is wide and computational run time is short, thus making the simulation model suitable for use with thermodynamically based cycle simulations in diesel engines running with ignition improver with Hexane fuel. Due to its simplicity, the model can be used for wide range of different fuels to optimize the design.

REFERENCES

[1] Lakshminarayanan, P. A. et. al, Modelling diesel combustion (Springer, 2009)

[2] Dr. Ibtihal Al-Namie et. al., "Study the Effect of Ceramic Coating on the Performance and Emissions of Diesel Engine", Journal of Engineering , August 2012, Volume 18, Number 8,935-942.

[3] Krishnan, D.B. et. al., "Performance of an Al-Si graphite particle composite piston in a diesel engine", Transactions of Wear, 1980,60(2), 205-215.

[4] Murat Ciniviz et.al., " Ceramic Coating Applications and Research Fields for Internal Combustion Engines", Ceramic Coatings - Applications in Engineering, Feb. 2012,195-234.

[5] Alkidas, A.C.," Performance and emissions achievements with an uncooled heavy duty, single cylinder diesel engine", SAE, vol. 890141, 1989.

[6] Uzun, A., Cevik, I. and Akcil, M, "Effects of thermal barrier coating material on a turbocharged diesel engine performance", Surf. Coat. Technology, 1999, 116–119.

[7] Spaniel, M., Macek, J., Divis, M. and Tichanek, R., ''Diesel engine head steady state analysis'', International Journal of Middle European Construction and Design of Cars, 2003,2(3), 34–41.

[8] M. Fadaei a, et. al., "New thermo-mechanical analysis of cylinder heads using a multi-field approach", Scientia Iranica, November 2010, 18 (1), 66–74.

[9] Parlak, A.," Experimental Investigation of Injection Advance and Compression Ratio of a Supercharged Ceramic Coated Diesel Engine", PhD Thesis, Sakarya University, Turkey (in Turkish), 2000.

[10] Gataowski, J.A., 'Evaluation of a Selectively Cooled Single Cylinder 0.5L Diesel Engine", SAE, Paper 900693,USA, 1990.

[11] Afify, E.M. & Klett, D.E. (1996), "The Effect Of Selective Insulation On The Performance, Combustion, And NOx Emissions Of A DI Diesel Engine", International & Congress and Exposition, Detroit, Michigan February 26-29.

[12] Rasihhan, Y. & Wallce F.J., Piston-Liner "Thermal Resistance Model for Diesel Engine Simultion", Proc. Inst. Mech. Engrs., 1991, Vol.205.