ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

Investigation of Temperature inside the Cylinder in 2-Stroke SI Engine Using CFD

R. G. Telrandhe, D. R. Ikhar, A. N. Ingale

***Abstract*— The main objective of the present work is to make a computational study of scavenging system in two-stroke medium capacity engines to reduce or to curb the emissions from the two-stroke engines. The 2-D flows within the cylinder are simulated using computational fluid dynamics using Fluent 6. The total pressure map from computation provided comprehensive information on the scavenging phenomenon. So a proper scavenging process will be carried out with better results to be seen. An unstructured cell is adopted for meshing the geometry created in GAMBIT software Turbulence is modeled by high Reynolds number version k-ε model. Good agreement is observed between predicted result and experimental data. These investigations have revealed a significant improvement in turbulence.**

***Index Terms*— Fluent, Gambit, Scavenging, Turbulence.**

I. INTRODUCTION

In this research ,an investigation of two stroke engine is carried out by using Computational Fluid Dynamics (CFD) analysis software FLUENT 6.3.26 and grid meshing is done with the help of GAMBIT 2.2.30.Then two-dimensional mesh is imported to FLUENT. The inlet and outlet boundary conditions were obtained from a gas dynamic calculation. The standard k-e turbulence model is used with no slip on the wall. The FLUENT dynamic mesh model feature is used to model the motion of two stroke engine during a typical cycle. The dynamic mesh model requires an input an initial mesh along with a specification of the motion of the moving parts. In case of IC engine the prescribed motion of the piston is required. The solver moves the components accordingly, and then automatically reconstructs the mesh, employing one, or a combination of the three available remeshing schemes, dynamics layering, spring smoothing, and local remeshing. To define the mesh motion, periodic rigid body motion is prescribed in the fluid zone above the piston using built-in-user-defined function (UDF), with parameters that are specified by the engine manufacturer. The geometry is meshed with quadrilateral cells. The experimental and computational simulation of two stroke engine is investigated in this research. Both methods of studies, experimental and simulation gave convincing evidence that shown the small discrepancy between experimental and simulation part of this research.

II. PROBLEM SOLVING STEPS

A. Creating Model Geometry Grid Generation in GAMBIT

The mesh was constructed using three separate blocks representing the intake port, cylinder and exhaust port of the engine. Their cell faces at the interface between each other matched exactly setting in the TGRID stage. The overall mesh structure for the piston at the TDC is shown in figure. The volume is divided and the mesh to fulfill the requirement of the dynamic mesh transient analysis. Quadrilateral meshing is used for exhaust port, intake port and cylinder tri. Volume grid generation was established using GAMBIT2.16 pre-processor. The lowest layer of vertices in the cylinder mesh was always maintained to be at piston crown level. Additional layers of cells within the cylinder were activated until achieved at the BDC. A layer of vertices was also required to be moved in the scavenged port to ensure that the cell faces of the cylinder and scavenged matched at the port also exhaust itself. In each case the datum mesh was read in at the start of each time step and mesh motion program modified the datum mesh to that required at the end of the time step. In this case, that setting of time step mesh motion start at TDC and until to BDC for one and half cycle.

Using FLUENT, it is provided a built-in function to calculate the piston location as a function of crank angle. Using the function it is needed to specify the piston stroke and connecting rod length. The piston location is calculated using the following expansion. $Ps= L+A/2(1-\cos(\theta c)-(L2-A2/4(\sin 2(\theta c))$ ½ Where, ps is the piston position (0 at TDC) and A is at BDC, L is connecting rod length, A is piston stroke and θc is current crank angle position. The current crank angle is calculated as below:

$\theta c = \theta s + t\Omega$ shaft

Where, θs is starting crank angle and Ωshaft is the crankshaft speed. The three mesh update procedures available

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

are dynamic layering, local remeshing and spring smoothing. In this case, only dynamic layering method is based on quadrilateral element. Stationary zones were maintained intact for update.

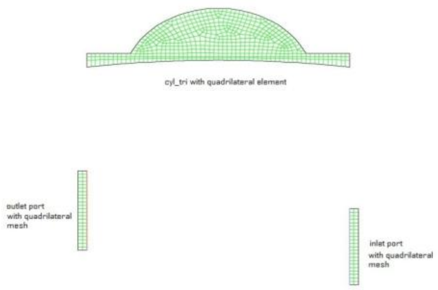

Fig.1. Mesh Geometry

III. CFD MODELING AND SOLUTION SETUP USING FLUENT

A. Setup Solver

A pressure-based, or segregated, solver was employed to decrease the computational demands in terms of required memory. Furthermore, the Semi-Implicit Method for Pressure-Linked Equations (SIMPLE) pressure-velocity coupling was employed for stability reasons. A predictor-corrector approach is used wherein a predicted pressure field is used to solve the momentum equations. A correction pressure is found as the difference between the correct pressure field and the predicted pressure field and is used to solve for the corrected components of velocity. Subsequently, the energy and turbulence transport equations are solved. This process is repeated iteratively until the desired convergence criteria are achieved.

B. Setup Physical Model

1 Standard k-ε Model: The standard k-ε model is a semi-empirical model based on model transport equations for the turbulence kinetic energy (k) and its dissipation rate (ε). The model transport equation for k is derived from the exact equation, while the model transport equation for ε was obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. In the derivation of the k-ε model, it was assumed that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-ε model is therefore valid only for fully turbulent flows. The simplest "complete models" of turbulence are two-equation models in which the solution of two separate transport equations allows the turbulent velocity and length scales to be independently determined. The standard k-ε model in FLUENT falls within this class of turbulence model and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding. Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations. It is a semi-empirical model, and the derivation of the model equations relies on phenomenological considerations and empiricism. As the strengths and weaknesses of the standard k-ε model have become known, improvements have been made to the model to improve its performance.

2 Discrete Phase Model: In addition to solving transport equations for the continuous phase, FLUENT allows to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. FLUENT computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included. Modeling capabilities allow FLUENT to simulate a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, and bubble stirring of liquids, liquid fuel combustion, and coal combustion.

3 Species Transport Equation: To solve conservation equations for chemical species, FLUENT predicts the local mass fraction of each species, Yi , through the solution of a convection-diffusion equation for the ith species. This conservation equation takes the following general form:

$$\partial/\partial t(\rho Y_i) + \nabla.(\rho\, v\, Y_i) = -\nabla.J_i + R_i + S_i$$

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012
Where, Ri is the net rate of production of species i by chemical reaction and Si is the rate of creation by addition from the dispersed phase plus any user-defined sources. An equation of this form will be solved for N-1 species where N is the total number of fluid phase chemical species present in the system. Since the mass fraction of the species must sum to unity, the N th mass fraction is determined as one minus the sum of the N-1 solved mass fractions. To minimize numerical error, The Nth species should be selected as that species with the overall largest mass fraction, such as N2 when the oxidizer is air.

IV. TURBULENCE MODELING

A. Turbulence Intensity

The turbulence intensity, I, is defined as the ratio of the root-mean-square of the velocity Fluctuations, u' to the mean flow velocity, Uavg. A turbulence intensity of 1% or less is generally considered low and turbulence intensities Greater than 10% are considered high. At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

I = u'/uavg = 0:16(ReDH) - 1/8

B .Turbulence Length Scale and Hydraulic Diameter

The turbulence length scale is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows. In fully-developed duct flows, is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between l and the physical size of the duct is l = 0:07L Where L is the relevant dimension of the duct. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe.

C. Convergence Criteria

The convergence criteria originally considered for this study are the scaled residuals; these represent the average residual sum of the conserved quantities considered in the RANS-type equations. The continuity, momentum, k and ε equations were solved until their scaled residuals reached 10-4 and subsequently 10-5. The two velocity profiles of 10-4 and 10-5 do not match very closely for z/D > 2.5. A convergence criteria based on the scaled residuals was not adopted since their magnitudes are based on the relative errors and thus do not provide an absolute measure of error. A criteria based on the error between two states in the solution differing by 1000 computational iterations was used for subsequent simulations. The criterion is based on a maximum relative error of 5% between the velocity profiles at the aforementioned locations. It is clear the agreement is quite good between the solutions and a maximum error of 5% between the two convergence times indicates that the solution is near convergence and will vary very little until it reaches the final RANS solution. This convergence criterion of a 5% error over 1000 iterations was adopted for all steady simulations.

V. EXPERIMENTAL SET UP AND OBSERVATION

Fig 2. Experimental Set up

From the experiment done with the help of compression test the contour of temperature at various RPM is shown in the following figures

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

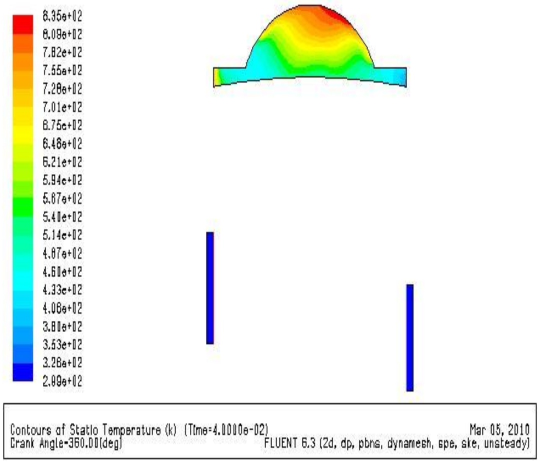

Fig 3 at 1000 RPM

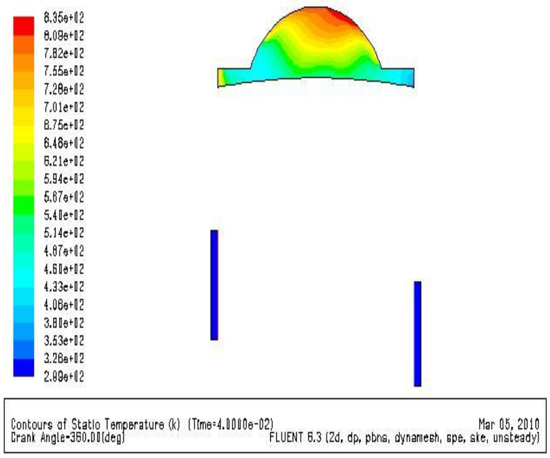

Fig 4 at 1500 RPM

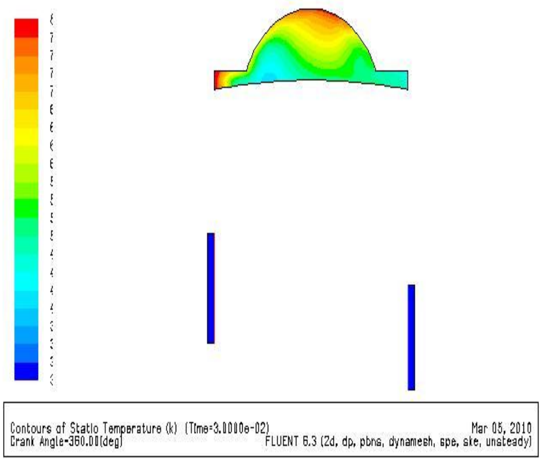

Fig 5 at 2000 RPM

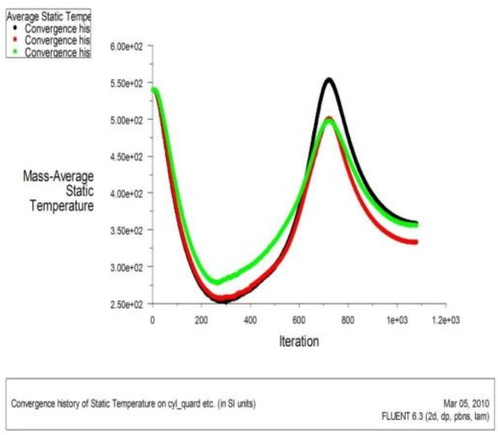

Fig 6 Effect of Engine Speed on Static Temperature

VI. RESULTS AND DISCUSSION

The research describes the CFD analysis technique to predict in- cylinder cold flow simulation of SI engine. All suction, compression expansion and exhaust strokes are simulated. The transient analysis of dynamic mesh was successfully investigated using CFD. The operating and boundary conditions are taken from experimental study. This is significantly important to understand the design and characteristics on cold flow process, short-circuiting on cylinder and exhaust port Form this pressure and temperature profile can be concluded that the higher speed can produce maximum pressure and temperature compare the lower speed of engine. The result shows that the higher rpm mode produces the higher cylinder temperature than lower rpm. Also the cylinder temperature is a function of crank angle for different speed. The results shown that, using the dynamic mesh approaches give confident results compared with experimental.

VII. CONCLUSION

This paper describes the CFD analysis technique to predict in cylinder cold flow simulation of SI engine. All suction, compression expansion and exhaust strokes are simulated. The transient analysis of dynamic mesh was successfully investigated using CFD. The operating and boundary conditions are taken from experimental study. This is significantly important to understand the design and characteristics on cold flow process, short-circuiting on cylinder and exhaust port. The result shows that the higher RPM mode produces the higher cylinder pressure than lower RPM. The results have shown that the dynamic mesh approaches give confident results compared with experimental.

ISSN: 2319-5967
ISO 9001:2008 Certified
International Journal of Engineering Science and Innovative Technology (IJESIT)
Volume 1, Issue 2, November 2012

REFERENCES

[1] Simone BARSOTTI, Luca CARMIGNANI, Luigi MATTEUCCI,"Optimization of a Two Stroke Engine Scavenging Process by a CFD Analysis in order to reduce the Raw Pollutant Emissions"SAE20056556, 2005-32-0113.

[2] N ABANI1, S BAKSHI2 and R V RAVIKRISHNA3, "Multi-dimensional modeling of spray, in-cylinder air motion and fuel–air mixing in a direct-injection engine" sadhana vol. 32, part 5, October 2007, pp. 597-617.

[3] J. B. Heywood, Internal Combustion Engine Fundamentals, McGraw-Hill, Inc., 1988.

[4] Barsotti, S., Carmignani, L. and Matteucci, L., "Optimization of a Two Stroke Engine Scavenging Process by a CFD analysis in order to reduce the Raw Pollutant

[5] Emissions", SAE Paper 2005-32-0113, 2005.

[6] J. Anderson, "Computational fluid dynamics ", McGraw-hill international editions ISBN 0-07-113210-4, 1995.

AUTHOR BIOGRAPHY

R. G. Telarandhe Working as Assistant Professor in Datta Meghe Institute of Engineering, Technology and Research, Sawangi (Meghe), Wardha (Maharashtra, INDIA), **Publications -** Papers in International Conferences – 2

D. R. Ikhar Working as Assistant Professor in Datta Meghe Institute of Engineering, Technology and Research, Sawangi (Meghe), Wardha (Maharashtra, INDIA), **Publications** - International Publications - 9, National Publications - 6

A. N. Ingale Working as Assistant Professor in Datta Meghe Institute of Engineering, Technology and Research, Sawangi (Meghe), Wardha (Maharashtra, INDIA), **Publications** - International Publications - 4, National Publications – 2