International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 05, May-2015

" Compitational Fluid Flow Dynamic Analysis on I.C Engine using ANSYS"

Syed Saleem Pasha
Assistant Professor
Department of Mechanical Engineering
Ghousia College of Engineering-Ramanagaram-562159
Banglore, Karnataka, India

Mohammed Imran,
Assistant Professor
Department of Mechanical Engineering
Ghousia College of Engineering-Ramanagaram-562159
Banglore, Karnataka, India

***Abstract*— CFD analysis is plays a important role in an structure safety of the component in automobile and aerospace applications. Simulation of IC engine is the most important engineering problems in the computational fluid dynamics field. In the present work, simulation of inlet, compression, expansion and exhaust processes will be carried out without considering fuel combustion when a piston moves from TDC and also size of the domain vary during the valve opening and closing, then the mesh well be provided for optimization in a structure to simulate in a stable dynamic conditions throughout a complete cycle. For 3-D analysis in CFD industrial standard codes are used called RANS. The turbulence model used in the analysis is k-ε which is commonly used in a industries that in turns the best model between precision and computational time. The geometric model and mesh (grid) is developed by using ANSYS with the help of ICEM-CFD command. This CFD domain shows that the whole experimental setup is on acoustic phenomena.**

***Keywords*—CFD; Turbulence model k-ε; ICEM-CFD; ANSYS; I.C engine.**

I. INTRODUCTION

CFD analysis is plays a important role in an structure safety of the component in automobile and aerospace applications. Simulation of IC engine is the most important engineering problems in the computational fluid dynamics field. In the present work, simulation of inlet, compression, expansion and exhaust processes will be carried out without considering fuel combustion when a piston moves from TDC and also size of the domain vary during the valve opening and closing, then the mesh well be provided for optimization in a structure to simulate in a stable dynamic conditions throughout a complete cycle.

The details of three-dimensional modeling on reciprocating engine geometry of flat cylinder head and a bowl-in-piston combustion chamber, simulating on non-firing conditions [1]. They were compared on flow characteristics inside the engine cylinder with different piston configurations; the bowl shaped piston plays a significant role near TDC [2]. They were studied the swirl motion in the cylinder during the intake and compression strokes [3]. Performance of full intake and compression processes and presented some comparisons with experimental data and theoretical Data, their results are predicted the turbulence velocity is differing with standard k–ε model [4]-[5]. Review on computations based and LES, then concludes that this method has largest potential [6]**.** Method for calculating the 3D flow-fields in reciprocating I.C engines, as a function of space and time, throughout the complete four stroke cycle by using governing equations [7].

II. PROBLEM DEFINATION

A. *Objectives*

- Simulation of inlet, compression, expansion and exhaust processes will be carried out without considering fuel combustion.
  - When a piston moves from TDC during the valve opening and closing,
  - Size of the domain vary during the valve opening and closing,
- Then the mesh well be provided for optimization in a structure to simulate in a stable dynamic conditions throughout a complete cycle.
- The turbulence model used in the analysis is k-ε which is commonly used in a industries that in turns the best model between precision and computational time.
- Distribution of temperature, pressure and velocity will be analyzed with varying crank angle.

B. *Methodology*

- Problem is defined through literature survey
- Solid modeling and meshing of inlet manifold and geometry of cylinder with one valve using ANSYS.
- Generating grids, with approximate boundary conditions i.e., industrial standard codes are used as RANS and K - ε model.
- Flow takes place in intake valve and engine cylinder will be simulated to study the variation of pressure velocity, temperature, at different crank angle using ICEM-CFD**.**
- Finally simulated results were compared with the actual results.

C. *Assumption*

- Ideal gas is used as working fluid.
- The simulation model is considered as flat piston.
- In a simulation only one valve is used for inlet and exhaust process.

- Assume that the valve will be start to open at 0^0 and closes at 180^0 during suction stroke and also valve will be start to open during exhaust stroke at 540^0 to 720^0.
- Adiabatic and isentropic process is considered for compression and expansion.

D. Model

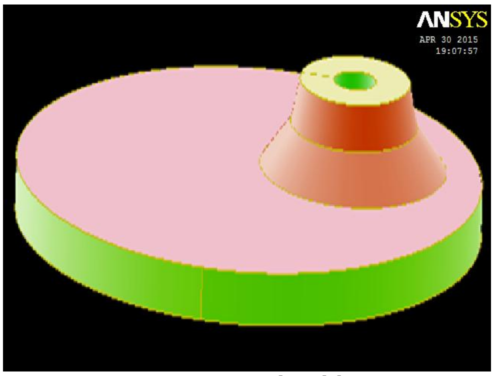
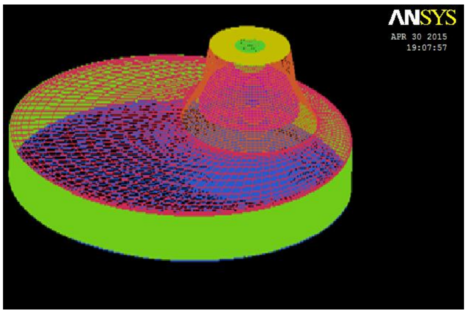


Fig. 1. Geometrical model

Fig. 1. shows Geometrical model developed using ICEM-CFD with manifold for both suction and exhaust by considering only one valve, with top surface as cylinder head and the thickness is clearance volume and bottom surface is piston.

E. Mesh Plot



\

Fig. 2. Computational Meshed Model

Fig.2. shows the computational meshed model, with 6 lakhs grid numbers with Hexahedral cells (structural mesh) is taken for the grid generation,

F. Bundary Conditions

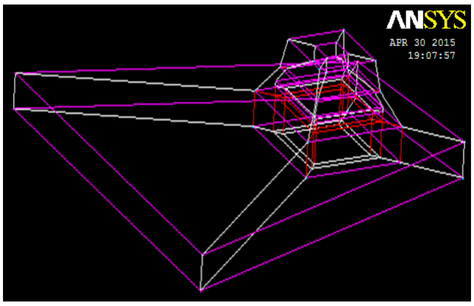


Fig.3. Blocking Approach

Fig. 3. shows the split blocks, discard unused blocks are created to capture underlying shape and the Associate edges to curves so that it capture hard features, vertices is move to position block corners on geometry.

Further boundary conditions are applied based on valve position as follows

- Inlet BC (subsonic, Mach number less than 1)
- Outlet BC (supersonic, Mach number greater than 1)
- Wall BC (adiabatic)

TABLE I. WALL BOUNDARY CONDITION

Parts	Motion of Mesh
Cylinder head, Manifold.	Stationary
Cylinder wall, Valve rod.	Unspecified
Piston, Valve bottom.	Specified

Boundary condition applied for I.C engine from [4] at initially

- Cartesian Velocity : 0.001 m/s
- Static Pressure : 0.081 N/mm^2 (MPa)
- Temperature :300 K
- K-ε value Model :0.001 m^2/s^2

G. Specifications

- Engine Speed : 1200 RPM
- Bore :100 mm
- Stroke :106 mm
- Compression Ratio :18
- Piston time approximated 't' : 0.08 sec
- For medium size engine, (crank radius)R :4.
- Intake Valve Opens :24^0 BTC
  Closes :62^0 ABC
- Exhaust valve Opens :59^0 BBC
  Closes :33^0 ATC

H. Formulas are for Theoretical Calculation

- Compression Ratio (r_c or C_R)

$$r_c = \frac{total\ volume}{clearance\ \ volume} \quad --(1)$$

- Crank Radius $a = L/2$ --(2)
- Connecting rod length $(l) = R * a$ ---(3)
- Angular velocity $\omega = \frac{2*\pi*N}{60}$ ---(4)
- Mean Piston Speed $S_p = 2 * L * N$ ---(5)
- Clearance volume $C_V = L/(C_R - 1)$ ---(6)
- Expression for piston movement error

$$Piston = 0 + [l + a - a * \cos(\omega * t)] - [(l^2 - (a^2 * \sin^2(\omega * t)))^{\frac{1}{2}}] - 0 \quad ----(7)$$

III. RESULT AND DISCUSSION

A. Theoritical Calculation

- Compression Ratio [From Eq. (1)]

$$r_c = \frac{total\ volume}{clearance\ volume} = 18:1$$

- Crank radius, [From Eq. (2)]

  a = 106/2 = 53mm
- Connecting rod length [From Eq. (3)]

  l= 4*53 =212mm
- Angular velocity (ω) [From Eq. (4)]

$$\omega = \frac{2 * \pi * 1200}{60}$$
$$=125.6 rad/s$$

- Mean Piston Speed [From Eq. (5)]

  $= 2 * 0.106 * 1200$ = 254.4 m/min = 4.24 m/sec
- Clearance volume(C_V) [From Eq. (6)]

  CV = 106 /(l7-1) = 6.625 mm
- Expression for piston movement error [From Eq. (7)]

Piston movement error = 0+212-211.99-0 =0.001mm

Expression for piston movement error show below graph

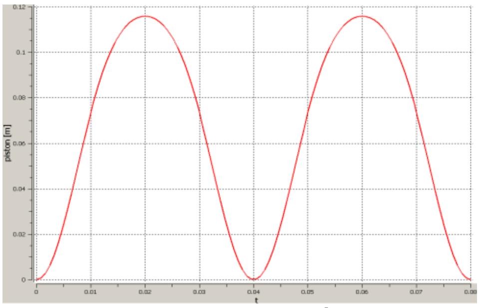


Fig. 4. Piston Movement Position with respect to Time

B. ANSYS Results

a) Velocity distribution

Velocity distribution at different crank angle to find swirl action of mass flow rate in I.C engine Results

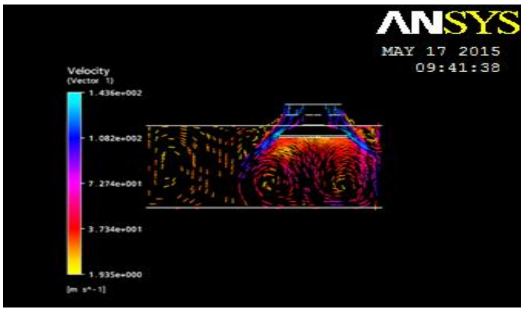


Fig. 5. Velocity distribution of mass flow rate at 60^0 Crank angle.

Fig.5. shows that maximum velocity distribution of mass flow rate is 1.434 x 10^2 m/s at 60^0 Crank angle, it clearly shows that maximum swirl action at the velocity 3.7 x 10^1 m/s (During suction stroke).

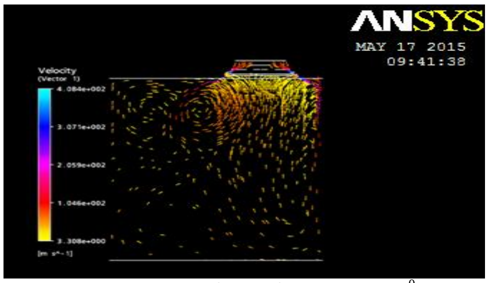


Fig. 6. Velocity distribution of mass flow rate at 180^0 Crank angle.

Fig.6. shows that maximum velocity distribution of mass flow rate is 4.0 x 10^2 m/s at 180^0 Crank angle, it clearly shows that minimum swirl action at the velocity 3.7 m/s (During closing suction stroke).

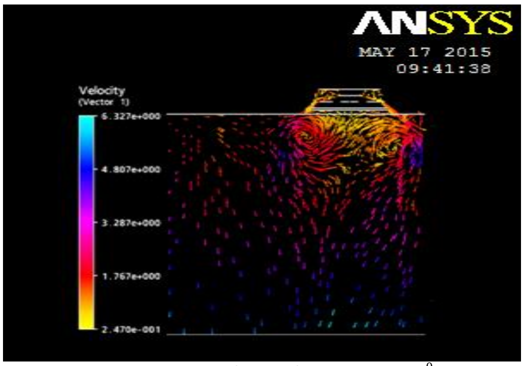


Fig. 7. Velocity distribution of mass flow rate at 540^0 Crank angle.

Fig.7. shows that maximum velocity distribution of mass flow rate is 6.327 m/s at 540^0 Crank angle, it clearly shows that maximum swirl action at the velocity 4.007 m/s (During closing expansion stroke).

b) Temperature Distribution

Temperature distribution at different crank angle to evaluate various ideal combustion of mass flow rate in I.C engine.

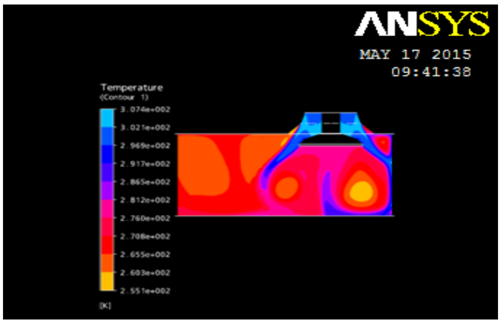


Fig. 8. Temperature distribution at 60^0 Crank angle.

Fig.8. shows that maximum temperature distribution of mass flow rate is 307 K at 60^0 Crank angle (During suction stroke).

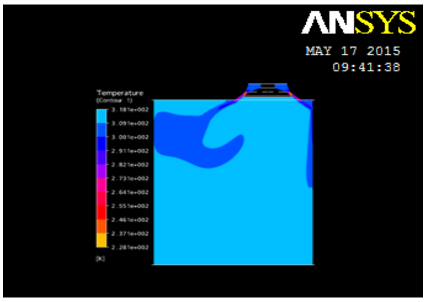


Fig. 9. Temperature distribution at 180^0 Crank angle.

Fig.9. shows that maximum temperature distribution of mass flow rate is 318.1 K at 180^0 Crank angle (During closing of suction stroke).

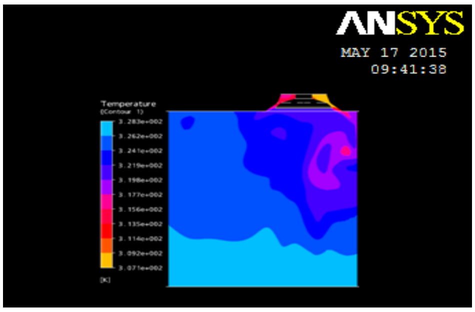


Fig. 10. Temperature distribution at 540^0 Crank angle.

Fig.10. shows that maximum temperature distribution of mass flow rate is 328.3 K at 540^0 crank angle (During Expansion stroke).

c) Pressure Distribution

Pressure distribution at different crank angle to evaluate various ideal combustion pressure in I.C engine.

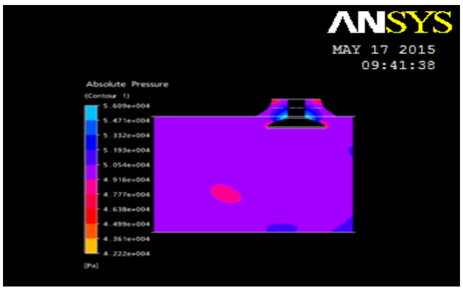


Fig. 11. Pressure distribution at 90^0 Crank angle.

Fig.11. shows that maximum pressure distribution of mass flow rate is 0.0056 MPa at 90^0 Crank angle (During suction stroke).

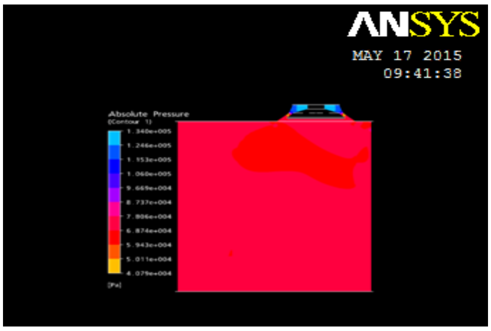


Fig. 12. Pressure distribution at 180^0 Crank angle.

Fig.12. shows that maximum pressure distribution of mass flow rate is 0.13 MPa at 180^0 Crank angle (During closing of suction stroke).

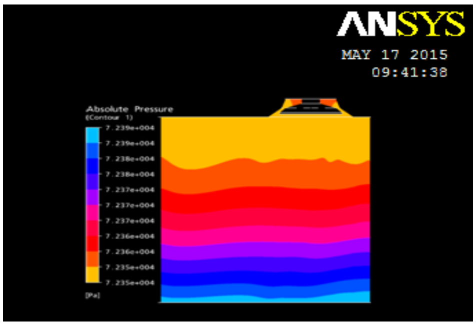


Fig. 13. Pressure distribution at 540^0 Crank angle.

Fig.13. shows that maximum pressure distribution of mass flow rate is 0.00723 MPa at 540^0 Crank angle (During expansion stroke).

C. Discussion

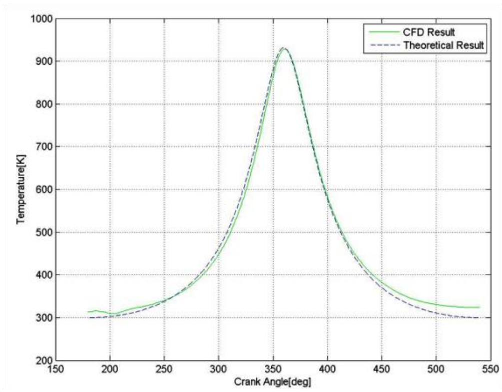


Fig.15 T-θ diagram of CFD and Theoretical Result

Fig.15 Temperature graph evaluates that comparative studies on CFD result and theoretical result are small deviation in initial and final tempetaure of CFD result from theoretical.

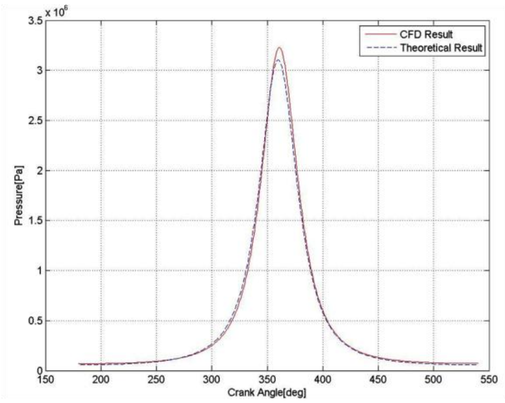


Fig.16 P-θ diagram of CFD and Theoretical Result

Fig.16 Pressure graph evaluates that comparative studies on CFD result and theoretical result are small deviation in peak pressure of CFD result from theoretical this may be due to divergence of residuals during iteration at the end of compression stroke.

IV. CONCLUSION

The present scope of work, development flow visualization setup for overcoming the difficulties fluid flow in I.C engines with different case as like velocity, temperature and pressure boundary conditions in experimental setup and also concerned with studies aimed at understanding type of flows in cylinder. The results obtained from the experiments and theoretical values are found to be in close agreement with the CFD results using k-ε turbulent model. Further, feature scope k-ε model is replaced with k-ω turbulent model validation qualifies the usage of the k-ω model as the effective turbulent scheme in the CFD model for the further heat transfer studies carried out in these geometries.

ACKNOWLEDGMENT

I cordially thanks to my family and my friends for their encouragement and motivation for this work.

REFERENCES

[1] “Simulation of fluid flow in a high compression ratio reciprocating internal combustion engine” by G.Sridhar, P.J Paul, H.S. Mukunda SAE paper 2005-01-1732

[2] CFD modeling of the in-cylinder flow in direct-injection Diesel engines, by F. Payri , J. Benajes, X. Margot , A. Gil, CMT-Motores T_ermicos, Universidad Polit_e cnica de Valencia, Camino de Vera s/n, 46022 Valencia, Spain.

[3] "Numerical simulation of swirling portvalve- cylinder flow in Diesel engine" Aita S, *et al.* . SAE 910263, 1991

[4] "Intake flow predictions of a transparent DI Diesel engine" by Chen A, Veshagh A, Wallace S.SAE 981020, 1998.

[5] "Direct injection Diesel engine simulation: a combined numerical and experimental approach from aerodynamics to combustion" Dillies B, Ducamin A, Lebrere L, Neveu F.SAE 970880, 1997

[6] "Large eddy simulations of in-cylinder turbulence for internal combustion engines" by Celik I, Yavuz I, Smirnov A. A Review. Int J Engine Res 2001;2(2):119–48.

[7] N “Three-Dimensional Computer Analysis of Flow and Combustion in Automotive Internal Combustion Engines” by C. Markatos and T. Mukerjee Mathematics and Computers in Simulation XXIII (1981) 354-366 North-Holland Publishing Company

[8] Heywood J.B. Internal combustion engine fundamentals, 1988, McGraw-Hill publication

[9] IC Engines by V Ganesan-Tata McGraw-Hill Education, 2002

[10] Numerical heat transfer and fluid flow by Suhas V.Patankar, Taylor and Francis 1980.