IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
ISSN: 2278-1684, PP: 58-65
www.iosrjournals.org

NUMERICAL SIMULATION OF MULTI-CHAMBER PISTON C.I ENGINE

Mr. ARAHANTH [1], Mr. KARTIKA S.B [2]
[1] *(Department of Mechanical Engineering,S.I.E.T/ V.T.U, India)*
[2] *(Department of Mechanical Engineering,S.I.E.T/ V.T.U, India)*

ABSTRACT: *Due to increase in the demand and the scarcity in the availability of the fossil fuels, it is essential to search for the effective burning of fuel in the existing system. In diesel engines the fluid motion within the engine cylinder is one of the major factors that controls the fuel-air mixing, combustion processes and also has a significant effect on heat transfer. Both the bulk gas motion and the turbulence characteristics of the flow are important for the performance of C.I engine.*

The present work relates to the modification of the C.I engine design for inducing turbulence by squish and tumble flows to improve the combustibility of the combustible mixture. This modification includes the formation of multi-chambers on the piston crown. It consists of three small chambers at 120º apart.

The C.I engine with multi-chamber piston at motoring condition has been analyzed through C.F.D using fluent software and the results obtained are compared with the base C.I engine.

Keywords: *piston crown, performance, squish, turbulence, tumble.*

I.INTRODUCTION

The in-cylinder flows of C.I engine is an important parameter for researchers & scientists. It is due to the fact that the flow structure generated by intake flows is related closely to the design and performance of the C.I engine. The production of high turbulence intensity is one of the most important factors for stabilizing the combustion process. In general two types of vertices are utilized in order to generate and preserve the turbulence flows efficiently. These vertices are usually known as swirl and tumble flows, which are organized rotations in the horizontal and vertical plane of the engine cylinder respectively. They contribute in improving the engine performance by accelerating mixing of fuel and induced air.

A few experimental works in visualizing in-cylinder flows have been conducted to measure velocity fields by using hot wire anemometry or laser Doppler velocimetry. However, it is a really hard task to perform it because the measurements of in-cylinder flows in the reciprocating engine are characterized by highly complex three-dimensionality, turbulence & unsteadiness. A numerical approach could thus be an alternative because of the capability of CFD which has been developed for in-cylinder flow predictions in recent years. The CFD package FLUENT is most efficient for in-cylinder flow analysis. The CFD models should cover the specific problems related to the turbulent flow, high Reynolds number, compressible flow and the complex geometry model. Consequently, the computational times are usually costly and require huge computer memory and even high performance computing (HPC) facilities to reduce the work.

In this work the CFD code of FLUENT with the capacity of moving mesh and boundary algorithms, including the valves and piston movement capability was employed to investigate the effect of multi-swirl chamber piston. The motivation behind this work is this type of modified piston has an influence in air-fuel mixing preparation and generation of turbulence as well as exerts great impact on engine performance. The numerical simulation is performed to obtain the optimum parameters mentioned above for modified engine, which influences to accomplish the better air and fuel mixing for rapid combustion process.

CFD code of Fluent 6.3 is one of the frequently used commercial package programs for in cylinder analysis in the reciprocating engine in the present days. The computer codes are utilized to solve the Navier-stokes (N-S) equation to produce detailed description of the mean velocity and turbulence velocity fields.

The most usual numerical method in Computational Fluid Dynamics (CFD) is finite volume method (FVM). In this investigation, an important fluid flow pattern in CFD simulations, namely, Tumble motion typical in automotive engines is considered and RNG k-e turbulence model were used. The air flow in a two-valve engine cylinder during 720 degree of crank angle was investigated by using a CFD code which is basis on finite volume.

Recently simulation results by Computational Fluid Dynamics codes are used in the development and optimization of new engines by car manufacturers (automotive industry). In order to simulate the mass flow rate and flow pattern of the induction system in an internal combustion engine, a multi-dimensional code was developed by FLUENT. Computed velocities and static pressures obtained from simulations were in good agreement with the experimental data. Computation of the three-dimensional flow in the intake ports of the cylinders of the real engines, including moving valves and piston was carried out by solving the Navier-Stokes equations. No explicit turbulence models were used.

II.ENGINE AND COMPUTATIONAL DETAILS:

The engine specification is same for both the engine except multi-chambers on the piston crown in modified engine. The detailed specification of the base engine selected for the simulation is given in Table 1.
Nowadays simulation tools can provide very useful information for internal combustion engine design and optimization in a relatively short time. In particular, CFD codes give a better insight into the physical processes which take place within the cylinder and some other components of the intake and exhaust systems. Concerning in-cylinder simulations, the Finite Volume Method (FVM) is applied to moving and unstructured meshes, in order to model the behaviour of turbulent, compressible and reactive flows. On one side, it has constantly to deal with new physical and numerical models, and on the other it must be flexible with respect to mesh structure and geometry handling to accommodate moving boundaries.

Table1 IC Engine Specification

SL NO	ENGINE PARAMETERS	SPECIFICATION
01	Number of Cylinders	**Single cylinder**
02	Number of strokes	**Four-stroke**
03	Rated power	**5.2KW(7HP) @ 1500rpm**
04	Bore	**87.5**
05	Stroke	**110mm**
06	Cubic Capacity	**661cc**
07	Compression ratio	**17.5:1**
08	Rated speed	**1500 rpm**
09	Type of cooling	**Water cooling**
10	**Fuel**	**Diesel**

While the motion is solely defined on boundary points, most CFD codes require specifying the position of every vertex for any time-step in the mesh. This can be done in different ways, most commonly using "mesh generation" techniques, like smoothing. In practice this is quite limiting, as it becomes difficult to prescribe solution-dependent motion or perform mesh motion on dynamically adapting meshes.

In the present work an automatic mesh motion algorithm has been adopted. It uses a second-order FEM scheme, in which the user defines how the vertices on the boundary move, using a set of boundary conditions. As nothing is done in advance of the solution, the boundary motion is "built into to code" and can be an arbitrary function of the solution. Then a Laplace equation is solved on the vertices to calculate the motion of all vertices based on the boundary motion. This equation is solved directly on vertices as doing it on volumes is not satisfactory. Additionally, to create a valid mesh, it must be guaranteed that no "crossover" in the mesh is created when executing mesh motion. In this way the position of internal points is determined from the prescribed boundary motion so that the initial mesh remains valid. To preserve the mesh quality during extreme boundary deformations due to piston and valve motion, the number of the cells in the mesh needs to be changed. For this reason a set of “topological changes” has been implemented, which concerns the possibility of attaching or detaching boundaries, adding or removing cell layers and sliding cells interfaces.

Generally an IC engine is a complex geometry. The complexity is further increased by the moving boundaries, after the mesh has been built, it is necessary to prescribe its movement in time to accommodate the movement of the boundaries. Moreover, for good accuracy it is also advisable to modify the topology of the mesh to preserve the sensible aspect ratios of the cell which changes with the boundary movement. For example, a mesh consisting of cubes at BDC may be unacceptable deformed at TDC. Therefore, the number of cell layers between the cylinder head and the piston needs to be changed as the piston moves.

The objective of this work is to perform modelling and analysis at motoring condition in CFD for base and modified engine.

III.PROCEDURE FOR PROBLEM SET UP AND SOLVING BY COMPUTATIONAL METHOD:

1. **Pre-Processing***:* The selected fluid flow problem starts with the construction of geometry, the generation of the mesh to the volumes. The geometry has been created by using CATIA. For creating the mesh Hyper Mesh software has been used. For 3D there are structured meshes of quadrilateral faces and other faces easier to develop like the triangles. Transporting the problem to 3D, hexahedral and pyramidal (tetrahedral) volumes
Can be carried out.

1.1 Geometric Model Creation:
The Geometry of computational domain from the analysis point of view 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, CATIA, etc). Geometry files are imported to HM to create computational domain. Thus the Extracted fluid domain of IC engine as shown in Fig.1

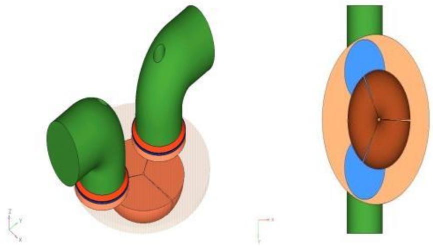
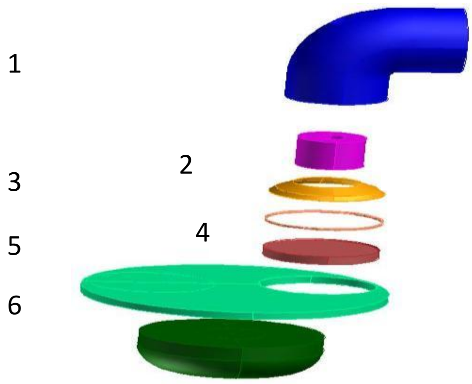


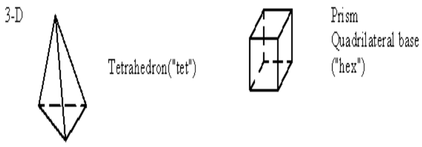
Fig:1 CFD Model of Computational Domain

1.2 Geometry Decomposition:
The Geometry decomposition of the computational domain from the analysis point of view is very important and this has two approaches in Fluent to solve in-cylinder problems, namely, hybrid approach and layering approach. While the hybrid approach is used for engines with canted valves like most SI engines, the layering approach is typically used for engines with vertical valves like most diesel engines.
For either approach mentioned above, in-cylinder problems solved in Fluent consist of three stages. The first stage is to decompose the geometry into different zones and mesh them properly. By breaking up the model into different zones, it is possible to apply different mesh motion strategies to different regions in a single simulation. The second stage is to set up the engine case inside Fluent with the help of a setup journal. The third stage is to perform a transient in-cylinder simulation. The decomposed parts is shown in Fig.2



Sl No	Parts Name
1	Inlet Manifold
2	Crest
3	Valve Seat
4	Connector
5	Valve Bottom
6	Piston Top
7	Piston Bowl

Fig: 2 Geometry Decomposition.



1.3 Mesh Generation:

Fig: 3 Shapes of Meshes Used

For the present work the meshing is done by using Hyper Mesh software. Many different cell/element and grid types are available. They are 2D (Triangle, Quadrilateral), 3D (Tetrahedron, Hexahedron, Pyramid, Wedge…) Choice depends on the problem and the solver capabilities.

In boundary layers, quad, hex, and prism/wedge cells are preferred over tri's, tets, or pyramids. For the same cell count, hexahedral meshes will give more accurate solutions, especially if the grid lines are aligned with the flow. The mesh density should be high enough to capture all relevant flow features. The mesh adjacent to the wall should be fine enough to resolve the boundary layer flow.

Skewness, Smoothness (change in size) and Aspect ratio are the three important measures of mesh quality.The Fig.3 shows the Tetrahedron and Quadrilateral cells. In present problem Tetrahedron, Hexahedron and combination of both (Hybrid) cells are used.

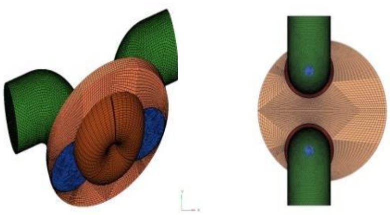


Fig:4 CFD Mesh Domain

1.4Boundary Conditions

The boundary conditions play a very important role in analysis for obtaining good results. Following are the assumptions incurred on the present analysis:

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

The decomposition and the zone name matching is explained in Fig.6 The sketch of the decomposition and the corresponding zone names are given in table.2. At the inlet, pressure boundary condition is applied. The engine wall is defined as stationary without any slip.

Fig:5 Fluid Zone Names

Table: 2 Names and Types of Mesh Requirement

Sl. No	Fluid Zone Names	Fluid Zone Mesh Requirements
1	Fluid-Bowl	Any type Of Mesh
2	Fluid-Chamber- Lower	Layered Mesh
3	Fluid-Chamber- Lower	Any type Of Mesh
4	Fluid-Chamber-Root Name	Layered Mesh
5	Fluid-Root Name-Inner Boundary	Layered Mesh (only requires to have one layered mesh close to the top)
6	Fluid-Root Name-Valve Layer	Layered Mesh

2. Solver setup*:*

Setting up of the solver is very important in any of the fluid flow problem; the solver setting indicates the method and also a procedure for solving (analysis) the problem. In FLUINT the solution can be obtained by many solving methods. All the methods will be working by considering the average of the fluctuation of the flow in-spite of considering the whole path of fluctuation. The transient RANS simulations are conducted on the internal Combustion geometry using Fluent. Segregated solver is used for the computations which employs a cell-centered finite volume method. A second-order upwind discretization is used for the momentum equation and a first order upwind discretization is used for turbulent kinetic energy. The solver settings applied in Fluent for the simulations of IC Engine are tabulated in Table.3. The In cylinder set up given for the present analysis is as shown in the fig.6.

Table:3 Solver Setting Applied For IC Engine

Solver	**Segregated**
Formulation	Implicit
Pressure Discretization	Standard
Momentum Discretization	Second order upwind
Turbulent kinetic energy	First order upwind
Specific dissipation rate (omega)	First order upwind
Pressure-Velocity coupling	SIMPLE

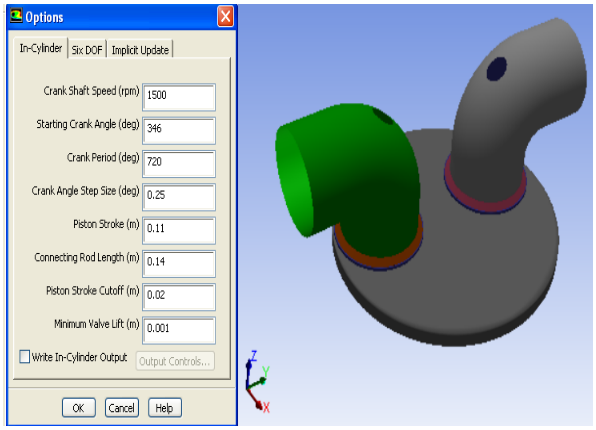


Fig:6 In cylinder Setup for the present problem

3. Post Processing

Post Processing or analysis of the results computed. There are lots of choices: Contours, X-Y plots, velocity vectors, path lines. At this stage we can also take the values in the form of Vector plots, Line and shaded contour plots, 2D and 3D surface plots, Particle tracking, Color post script output and several variables can be analyzed as velocity, pressure, turbulence, forces, density and others.

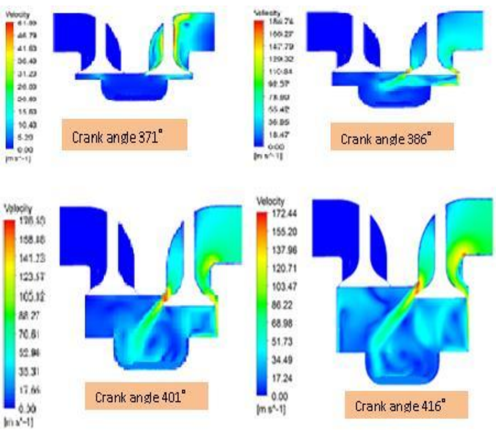
IV.RESULTS AND DISCUSSIONS:

This chapter deals with the results obtained from the CFD analysis for both base and modified engine. The peak pressure, tumble ratio and squish velocity are discussed at different crank angles. The CFD output in terms of contours of pressure, temperature and velocity are obtained as soon as the runs completes. It is very difficult to compare the results with those contours; hence these contours are used to draw the Graphs.

Comparison of Tumble ratio and squish velocity between base and modified engine:

Tumble is a rotational motion about a circumferential axis near the edge of the clearance volume in the piston crown or in the cylinder head, which is caused by squishing of the in-cylinder volume as piston reaches near TDC [6]. Tumble flows are always generated during intake and compression due to high turbulence in the engine cylinder and in power and exhaust stroke its effect will be less.

Squish is the name given to the radially inward or transverse air motion that occurs toward the end of the compression stroke when a portion of the piston face and cylinder head approach each other closely [8]. The CFD output of velocity for base and modified engines are as follows:



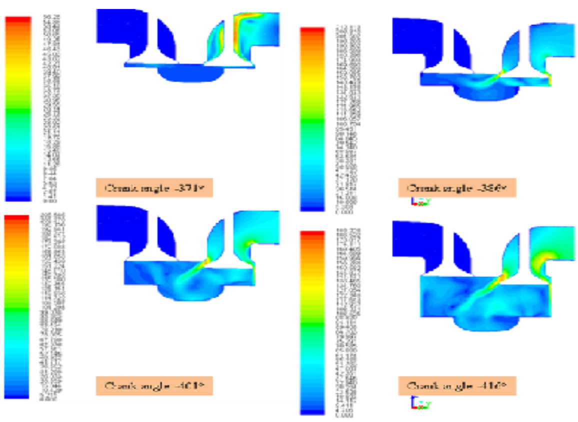


Fig:7 **Fig:8**
Velocity contours at different **Velocity contours at different crank crank angles for base**
Engine **angles for modified Engine**

Fig 7 and 8 shows the variations of fluid velocity at different crank angle for both normal as well as modified engine. During the suction stroke, as the piston moves from TDC to BDC, velocity of the fluid is increasing. When velocity counters of both the engines are compared, it is found that the Tumble (Vortices) has began at the crank angle of 416° in modified engine than the normal engine. As the piston moves from BDC to TDC during the compression stroke velocity is decreasing and there are no noticeable vortices in the normal engine but we can see number of small vortices in the modified engine.

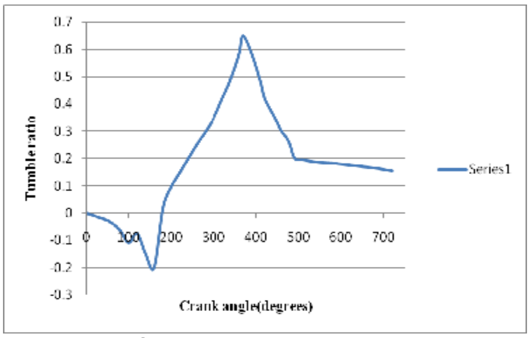


Fig: 9 Tumble ratio v/s crank angle (degrees) for base engine

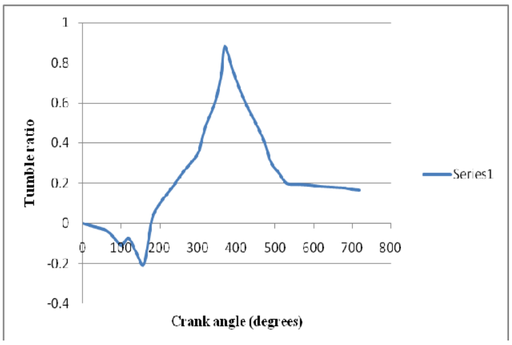


Fig:10Tumble ratio v/s crank angle (degrees) for modified engine

The values of tumble ratio against the crank angle are obtained from the CFD output for base and modified engines are shown in fig 9 and 10 respectively. It can be seen that at the beginning of the suction stroke, tumble is negative due to the geometry of intake manifold [6]. During the compression stroke the tumble ratio gradually increases with the increase in the crank angle. The maximum tumble ratio of the normal engine is 0.65 at crank angle 360º and for the modified engine is 0.88 at the 370° crank-angle. Hence the increase in tumble ratio with the modified piston is around 35%.

From the CFD study, it is clear that the modified piston has the capability of producing higher tumble.

This occurs due to the presence of multi-chambers on the piston crown. It is determined from the CFD analysis that the modified engine has better fluid flow field characteristics and it will improve the combustibility of combustible mixture.

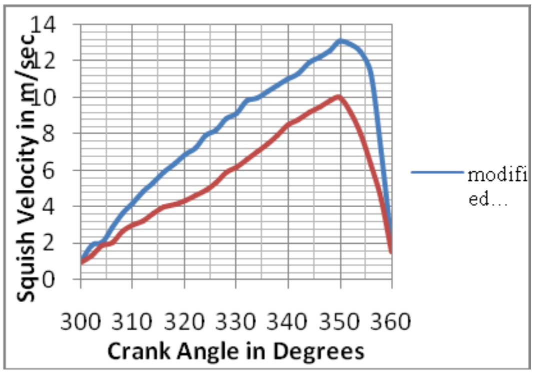


Fig:11 Squish velocity (m/s) v/s crank angle (deg)

The CFD output for squish velocity is plotted against the crank angle is shown in figure11 It is observed that the maximum squish velocity occurs at about 10˚ before top dead centre it’s about 10 m/s for base engine and 13.1 m/s for modified engine, then it decreases. After top dead centre squish velocity would be negative [8]. Hence the increase in squish velocity with the modified piston is around 31%. In the present work squish is generated due to modification in piston design that is by creating multi-chamber on piston crown. From CFD analysis it is determined that increase in squish velocity increases the fluid flow characteristics in modified engine.

V.CONCLUSIONS:

To improve the air flow characteristics of 4 stroke diesel engine, the base engine piston was modified with multi-chamber on the piston crown. The investigation and analysis of air flow characteristics during suction and compression strokes were carried out using CFD code of Fluent 6.3 with dynamic mesh. After the analysis the following results were drawn:

1. The peak tumble ratio of the modified engine was found to be 0.88 in comparison with the base engine of 0.65. Thus an improvement of 35% of tumble ratio is achieved.
2. The peak tumble ratio in modified engine shifts towards TDC by 10° compared to the base engine.
3. The squish velocity of the modified engine was increased by 31% in comparison with the base engine.
4. It is clear from the study that the modified piston has the capability of producing higher tumble and squish which will improve the combustibility of combustible mixture during combustion.

REFERENCES:

[1] *Code coupling to enhance cfd analysis of i.c. Engines by Gino Bella*, Rossella Rotondi*, Gabriel Defranco† y Norberto Nigro†, Mecánica Computational Vol. XXIIM. B. Rosales, V. H. Cortínez y D. V. Bambill (Editores)Bahía Blanca, Argentina, November 2003, p.no. 26-40.*

[2] *Flow field development in a direct injection diesel engine with different manifolds by Benny Paul1*, V. Ganesan2, International Journal of Engineering, Science and Technology Vol. 2, No. 1, 2010, pp. 80-91.*

[3] *Computational Analysis Of Egr Mixing Inside The Intake System & Experimental Investigation On Diesel Engine For Lcv by S Karthikeyan R. Hariganesh, M.sathyanadan S. Krishnan, p. Vadivel, D vamsidhar, International Journal of Engineering Science and Technology (IJEST) Vol. 3 No. 3 Mar 2011 p.no.2350-2357.*

[4] *Turbulence and Heat Transfer Analysis of Intake and Compression Stroke in Automotive 4-stroke Direct Injection Engine by Algerian Journal of Applied Fluid Mechanics | Vol 1 | 2007 p.no.37-50.*

[5] *In An Engine Valve Lift Visualization And Simulation Performance Using Cfd by Semin, Rosli Abu Bakar, Abdul Rahim Ismail and Ismail Ali, Conference on Applications and Design in Mechanical Engineering, 25-26 October 2007, Kangar, Perlis, Malaysia.*