Empirical simulation of spherical combustion chamber geometry in a DICI engine

S.Arumugam*, K.Pitchandi, P.Maheskumar
Department of Mechanical Engineering,Sri Venkateswara College of Engineering, Pennalur, Irungattukottai, SriperumbudurTaluk,Pin – 602 117, Tamilnadu, India
***Corresponding Author: E.mail:arumugam@svce.ac.in; 9444763422**

ABSTRACT

In-cylinder flow analysis were carried out for the spherical combustion chamber in Ansys 14.5the challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder, and piston.The simulation work is to investigate the effect of spherical combustion chamber in a DICI Engine. The influences of the combustion chamber geometry characteristics on velocity contour, temperature contour, in-cylinder flow direction and swirl motion of the combustion have been investigated. The initial swirl is produced by the inlet valve design and increased by the spherical combustion chamber geometry. Achieve complete combustion of the air and fuel mixture at centre of the combustion chamber due to the tumble motion initiated the intake air and strengthened by high pressure fuel is injected at the centre of the combustion chamber. Temperature contour shows that due to the amount of fuel burnt inside the combustion chamber has to increase temperature of the exhaust gas and velocity.

Keywords: Diesel engine, Direct Injection, Spherical Combustion chamber geometry, Compression ignition, Flow direction, Swirl motion.

INTRODUCTION

The design and manufacture of Direct Injection Compression Ignition (DICI) engines is under significant pressure for improvement. The next generation of engines needs to be compact, light, powerful, and flexible to produce less pollution and use less fuel. Innovative engine designs will be needed to meet these competing requirements. The ability to accurately and rapidly analyze the performance of multiple engine designs is critical (Heywood J B, 1988; Pundir B P, 2010).

When the liquid fuel is injected into the combustion chamber, the spray cone gets disturbed due to the air motion and turbulence inside. The onset of combustion will cause an added turbulence that can be guided by the shape of the combustion chamber. Swirl is defined as organized rotation of charge about the cylinder axis (Asif, 1996; Venkateswaran and Nagarajan, 2010). Swirl is created by bringing an intake flow into the cylinder with an initial angular momentum. Swirl is used in CI engine concepts to promote more rapid mixing between the inducted air charge and the injected fuel. Swirl is also used to speed up the combustion process and in two-stroke engines, it improves scavenging. Squish is the name given to the radially inward or transverse gas motion that occurs towards the end of the compression stroke when apportion of piston face and cylinder head approach each other closely (Prasad, 2011; Dehong, 1996).

This interaction is severe in combustion chamber design for re-entrant and toroidal for high swirl DICI engines, a Re-entrant combustion chamber geometry in which the lip of the combustion chamber protrudes beyond the walls of the bowl shapes provides a substantial improvement in combustion, performance and emissions over the toroidal combustion chamber design (Jaichandar, 2012; Middlemiss, 1978; Carraretto, 2004; Reyes, 2006).

The performance of DICI engine depends upon complex interactions between mechanical, fluid, chemical, and electronic systems. However, the challenge in design is the complex fluid dynamics of turbulent reacting flows with moving parts through the intake/exhaust manifolds, valves, cylinder, and piston (Labeckas, 2006; Tsolakis, 2007). The time scales of the intake air flow, fuel injection, liquid vaporization, turbulent mixing, species transport, chemistry, and pollutant formation all overlap, and need to be considered simultaneously. Computational Fluid Dynamics (CFD) has emerged as a useful tool in understanding the fluid dynamics of DICI engine for design purposes. This is because, unlike analytical, experimental, or lower dimensional computational methods, multidimensional CFD modelling allows designers to simulate and visualize the complex fluid dynamics by solving the governing physical laws for mass, momentum, and energy transport on a 3D geometry, with sub-models for critical phenomena like turbulence and fuel chemistry. Insight provided by CFD analysis helps guide the geometry design of parts, such as ports, valves, and pistons; as well as engine parameters like valve timing and fuel injection (Tsolakis, 2008).

Engine analysis using CFD software has always been troubled due to the inherent complexity in Specifying the motion of the parts. Decomposition of the geometry into a topology that can successfully duplicate that motion is critical. Creating a computational mesh in both the moving and non-moving portions of the domain is quite difficult. Then solving the unsteady equations for flow, turbulence, energy, and chemistry should be verified for its accuracy. Post processing of results and extracting useful information from the very large data sets is crucial part in CFD (Ramadhas, 2005).

MODEL FORDICI ENGINE

Combustion Chamber and Piston Shape: A critical design issue is the size and shape of the combustion chamber, the piston crown shape, and the layout of the valves. Here, the chamber can be flat, a hemispheric dome, or a penta-head, while the piston crown can be flat, domed, or a bowl. The valves can be positioned as "straight", i.e. the valves are aligned with the cylinder axis. Straight Valve Engine, or they can be "canted", i.e. they are at an angle to the cylinder axis and normal to the surfaces of the combustion Chamber. It has been shown that the volumetric efficiency and the amount of air that makes it into the cylinder is dependent on the ratio of the intake valve area relative to the cross section area of the cylinder. Hence it is desirable that the intake valves be as large as possible relative to the bore. However, if the combustion chamber is flat, it limits the surface area available for the valve layout to just the cross section. If the combustion chamber is hemispheric or penta-headed, it opens up more surface area for the intake and exhaust valves, allowing them to be larger and more efficient. However, this means that the combustion chamber has a larger volume and surface area, which implies that the flame front for combustion has a longer distance to travel, increasing the chance of incomplete combustion. Also the compression ratio will be decreased since there is a larger volume at the top centre. In addition, a larger wall surface area increases the heat losses during combustion. Thus, there is adverse impact on combustion efficiency.

This may be counteracted by changing the piston shape from the flat shape to a domed shape to reduce the volume. But this means that the flame front has to travel around the piston dome to reach all parts of the combustion chamber volume, thus increasing the time taken for complete combustion, raising the possibility of knocking in SI engines. The piston could then be made to have a bowl in the centre, which would reduce the flame travel time, but reduce the compression ratio.

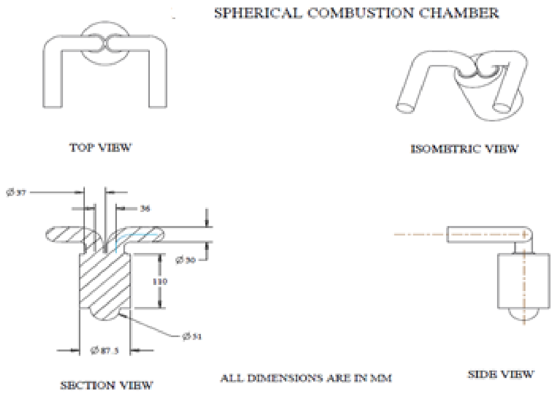

Fig.1. Spherical Combustion chamber

Flow analysis in spherical combustion chamber: By deriving the combustion chamber volume from the modelling software, the same combustion chamber is imported in ANSYS for the In-cylinder Flow Analysis.The following are the images of spherical combustion chamber.

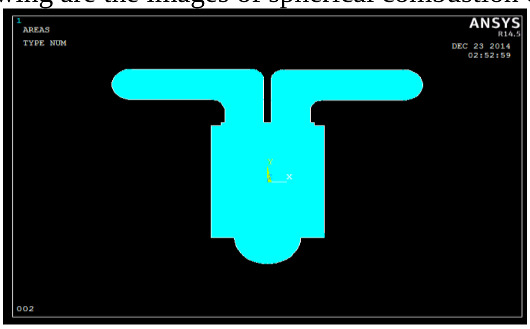

Fig.2. 2-D view for chamber volume

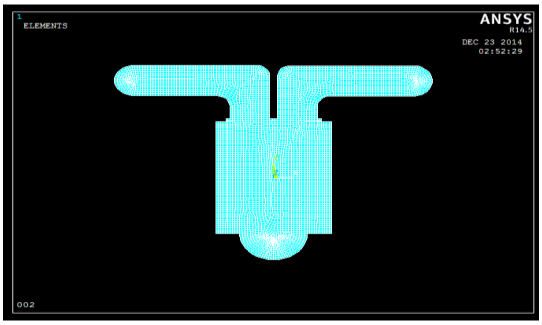

Fig.3. Meshed view for chamber

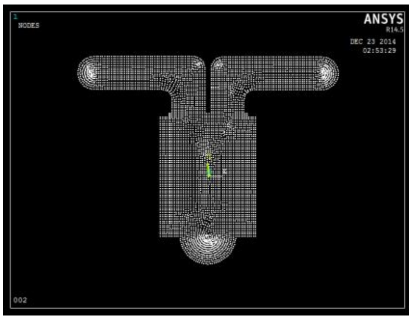

Fig.4. Nodes for spherical chamber

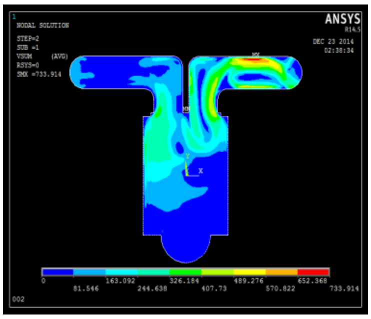

Fig.5. Velocity contour (u-sum) forspherical chamber

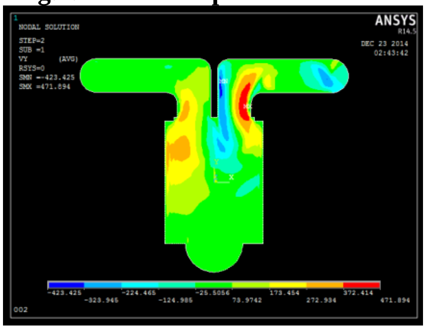

Fig.6. Velocity contour (y-direction)

Fig.7. Temperature contour

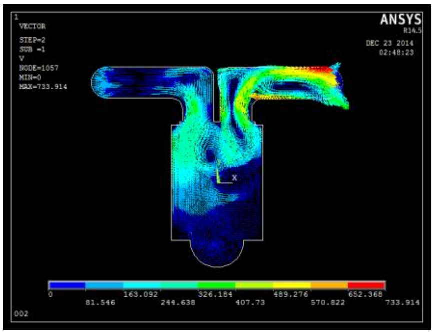

Fig.8. In-cylinder swirl motion

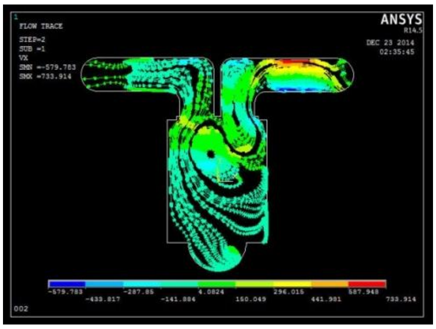

Fig.9. In-cylinder flow direction

CONCLUSION

In-cylinder flow analysis is carried out for the spherical combustion chamber in Ansys 14.5 and the following effects are observed while analysing the various resultant plots. The velocity of the exhaust gas has been rapidly increased after the compression stroke and at the expansion stroke at the same time as simulating the spherical combustion chamber. The overall velocity and directional velocity (y-plot) produces the same increase in the velocity after the combustion process. This leads to the better intake of fresh air (O_2).The swirl increases at the tip of the spherical combustion chamber due to shape of the bowl. The basic swirl flow of the chamber is achieved by the inlet valve design. At the centre of combustion chamber, the tumble flow is achieved due to the high pressure fuel is injected at the centre of combustion chamber and location of the injector is leads to enhance the complete combustion.

The temperature plot shows that, temperature at the piston head or crown is more than in the cylinder bore. This leads to the less fuel impingement in the spherical combustion chamber. The amount of fuel burnt inside the combustion chamber has been increase due to the increase in temperature.

REFERENCES

Asif, F. Christopher, SW. Michael, PW, Air pollution from motor vehicles: Spherical chambers and technologies for controlling emissions. Washington D.C, the World Bank Publications, 1996.

Brandl F, Reverencic I, Cartellieri W and Dent J, Turbulent air flow in the combustion bowl of a D.I. diesel engine and its effect on engine performance, Society of Automotive Engineers, Technical, 1979.

Carraretto C, Macor A, Mirandola A, Stoppato A, Tonon S, Biodiesel as alternative fuel: experimental analysis and energetic evaluations, Energy, 29, 2004, 2195–211.

Dehong, Z. and Hill, P.G, Effect of swirl on combustion in a short cylindrical chamber', Combustion and Flame, 106, 1996, 318–332.

Heywood J B, Internal combustion engine fundamentals, McGraw-Hill, 1988.

Jaichandar S and Annamalai K, Influences of re-entrant combustion chamber geometry on the performance of Pongamiabiodiesel in a DI diesel engine, Energy, 44, 2012, 633-640.

Labeckas G, Slavinskas S, The effect of rapeseed oil methyl ester on direct injection diesel engine performance and exhaust emissions, Energy Conversion and Management, 47, 2006, 1954–67.

Middlemiss I, Characteristics of the Perkins 'Squish Lip' Direct Injection Combustion System, SAE Technical Paper, 780113, 10.4271/780113, 1978.

PasupathyVenkateswaran S. Nagarajan G, Effects of the re-entrant bowl geometry on a DI turbocharged diesel engine performance and emissions- a CFD approach, 132(12), 2010, 803-813.

Prasad BVVSU, Sharma CS,Anand TNC, Ravikrishna RV., High swirl-inducing piston bowls in small diesel engines for emission reduction, Applied Energy, 88(7), 2011, 2355–67.

Pundir B P, I.C.Engines combustion and Emissions, Narosa Publishing House, 2010.

Ramadhas AS, Muraleedharan C, Jayaraj S, Performance and emission evaluation of a diesel engine fueled with methyl esters of rubber seed oil, Renewable Energy, 30, 2005, 1789–800.

Reyes JF, Sepulveda MA. PM-10, Emissions and power of a diesel engine fueled with crude and refined biodiesel from Salmon oil, Fuel, 85, 2006, 1714–1719.

Tsolakis A, Megaritis A, Wyszynski ML, Theinnoi K, Engine performance and emissions of a diesel engine operating on diesel-RME (rapeseed methyl ester) blends with EGR (exhaust gas recirculation), Energy, 32, 2007, 2072–80.

Tsolakis A, Megaritis A, Yap D, Engine performance and emissions of a diesel engine operating on diesel-RME (rapeseed methyl ester) blends with EGR (exhaust gas recirculation), Energy, 33, 2008, 462–70.

Usta N, An experimental study on performance and exhaust emissions of a diesel engine fuelled with tobacco seed oil methyl ester, Energy Conversion and Management, 46, 2005, 2373–86.