International Refereed Journal of Engineering and Science (IRJES)
ISSN (Online) 2319-183X, (Print) 2319-1821
Volume 5, Issue 8 (August 2016), PP.33-39

Simulation And CFD Analysis of Various Combustion Chamber Geometry of A C.I Engine Using CFX

Dr. Abdul Siddique[1], Shaik Abdul Azeez[2], Raffi Mohammed[2]
[1]*(Mechanical Engineering Department, King Khalid University, Kingdom Of Saudi Arabia)*
[2]*(Department of Mechanical Engineering, NRI Institute of Technology, A.P, INDIA)*

Abstract:- The geometry of the combustion chamber Is one of the factor Effecting the efficiences of C.I Engine (Diesel Engine). There are various engines used for agriculture purpose out of those MINI-PETER Diesel engine is generally used in present days. In this present work the modified geometries of MINI-PETER diesel engine is compared with the baseline data. The effect of geometry on the parameters like CO, CO_2, NO_X, HC and smoke density are studied. From the results it is concluded that the turbulence effect in the modified geometry is higher than the baseline geometry parameters.

Keywords: - CFX, emission analysis of engine, emission analysis of C.I engine, CFD analysis, combustion chamber

I. INTRODUCTION

An I.C Engine is one of the best available reliable sources of energy in the field of agriculture. Major issue arises on performance of diesel engine are enhanced by proper design of combustion chamber. Flow and combustion chemistry which effect swirl induced by re-entrant piston crown on pollution emission from a single cylinder diesel engine. For more efficient in combustion, less emission and soot, High Carbon formation is required. It is observed that from the literature several types studies and methods that have been reported in to increase the performance of engine such as injection pressure, injection timing, exhaust gas recirculation, swirl ratio, multi injection spray angle, nozzle diameter etc,. In this present study concentrates on combustion chamber area of single cylinder diesel engine with specification of 3.93 kW, 1550 rpm .By Using of piston crowns in top portion of piston is most advantages for proper combustion timing and increasing the volumetric efficiency of the engine. CFD and CFX analysis of the C.I engine has been carried out with various piston geometries and validated and compared with experimental data. From the results it is concluded that there is a reasonable agreement between CFD (using Ansys Fluent 14.0) and Experimental results.

II. ROLE OF CFD ANALYSIS IN IC ENGINE

CFD and CFX investigates the Fuel injection in expansion stroke and exhaust strokes, air flow path inlet, chemical reactions, and pollutant control on 4-S diesel cycle. Combustion occurs as a flame front propagating into the unburnt reactants. Direct Injection cycle under the categorized in non-premixed combustion, which is influenced by swirl and turbulence. With the rise of modern technology of software computing power and CAD systems, it has become effective for analysts to perform CFD analysis of internal combustion engine analysis

III. MODELING AND MESHING

The geometry of the C.I engine is modelled in Pro-Engineer software. Mesh creation and specific zone name is done in Gambit 2.4.6 and model is imported into FLUENT 14.0. The mesh created is based on the crank angle specified and the results obtained from the different stroke of cycle environment.

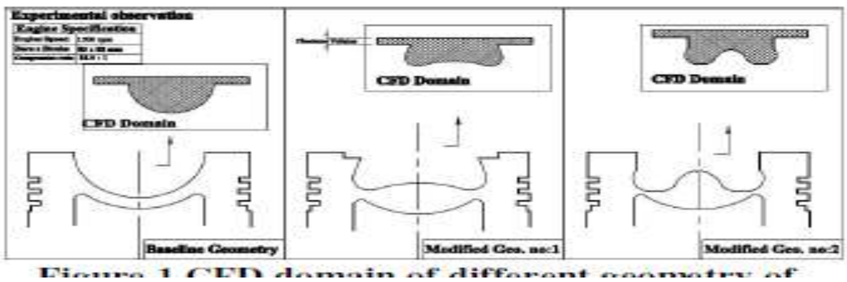


Figure 1 CFD domain of different geometry of combustion chamber show different

Configuration of piston.

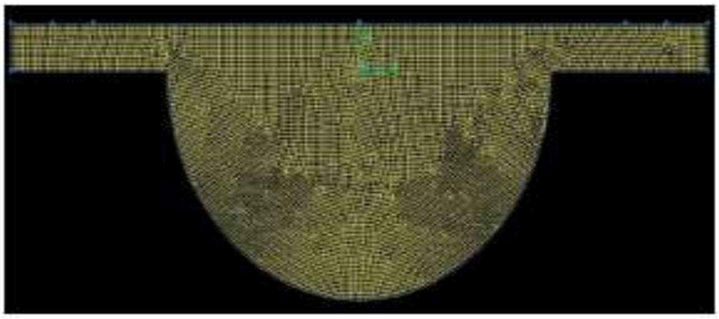


Figure 2 Meshed baseline geometry

3.1 Computational methodology and Boundary condition

For CFD analysis viscous standard k-e RNG standard model is enabled for considering volumetric reaction and eddy dissipation. Domain is subjected to motion of piston suitable boundary condition for piston, cylinder, fluid and cylinder walls. Combustion process in a C.I engine involves the transient injection of finely atomized liquid fuel into the air at high temperature and pressure. Boundary condition location of the injector, size of the injector, injection temperature and pressure, mass flow rate are having significant effect in diesel combustion modeling. The injection mass flow rate parameters and Engine specifications are given below

Table 1 Engine specification

Parameter	Magnitude
Engine Speed	1550 rpm
Mass Flow Rate	0.00111055 kg/s
Spray Cone Angle	55 Deg.
Start Crank Angle	360 Deg.
Stop crank Angle	720 Deg.

3.2 Governing Equation and Scope of study for combustion model

Turbulent flow model is consist 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 of mass conservation including continuity and Navier-Stokes equation momentum, energy and k-e RNG model turbulence equations. When area of combustion chamber increase, volumetric efficiency (1) will increase, that's possible.

$$\eta v = \frac{\text{ma (mass of air inducted in cylinder per cycle)}}{\text{Density of air inlet X Vd (Displaced Volume)}} \quad \text{………… (1)}$$

For this Research study numerator part of equation (1) is to be increase within the constrained domain of engine size and shape for increase the efficiency of the cycle.

3.3 Chemical species involved

Combustion chemical reactions computed in the burnt gases is function of mean local quantities Figure 3 computed in software in that single step oxidation of methane with oxygen to form carbon dioxide and water vapor is considered. The following species has been used for Diesel engine Combustion fuel Diesel (C10H26).

CnHp + O2 _ CO2 + H2O…………. (2)

2 $C_{10}H_{26}$+ 37 O_2 _ 20CO_2 + 34 H_2 ………. (3)

ANSYS FLUENT predicts the local mass fraction of each species through the solution of convection diffusion equation for the ith species. This conservation equation has the form

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho \vec{v} Y_i) = -\nabla.\vec{J}_i + R_i + S_i \qquad (4)$$

Where *i S* = Net rate of production of species ith by chemical Reaction.

R = Rate of Creation by addition from the Dispersed phase + any user defined Source.

3.4 Mesh Independency

Grid independency is checking the result for solution is independent from different mesh types and size, result is only depend the CFD domain's Boundary conditions.

Table 2 Input parameter of CFD domain

Parameter	Magnitude
Crank shaft speed	1550 rpm
Crank radius	56 mm
Bore	85 mm
Stroke	85 mm
Fuel	Diesel $C_{10}H_{26}$

Which become solve main outputs from CFD analysis like velocity, Swirl, pressure drop, mass flow rate.

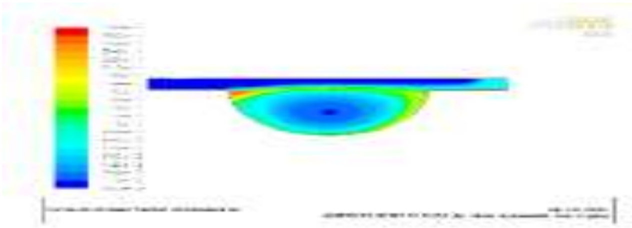


Figure 3 Tri mesh Scheme

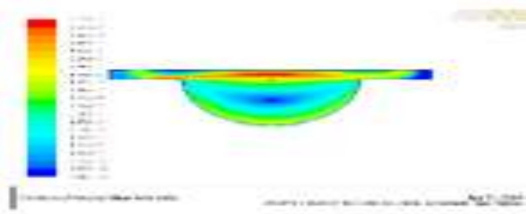


Figure 4 Quad poor mesh Scheme

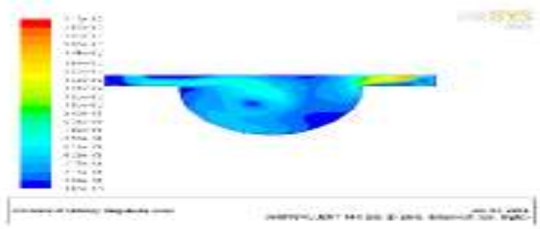


Figure 6 Quad Coarse mesh

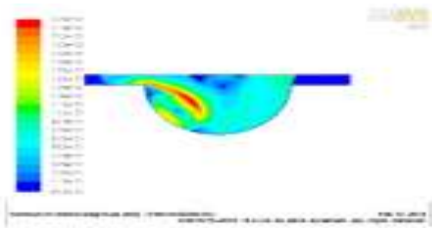
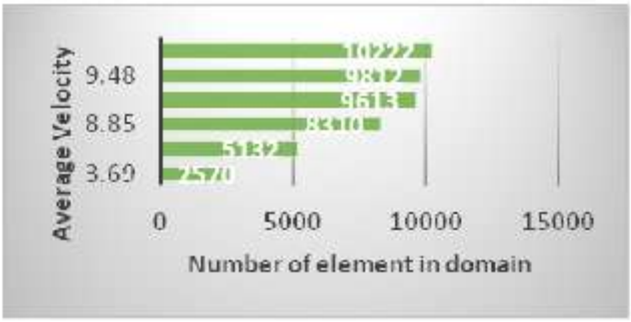


Figure 5 Quad Very fine mesh

Figure 3, 4, 5, and 6, show the independency of mesh size in velocity result, from contour we state that the more or 9613 element in the domain appropriate result.



Graph 1 Average velocity vs. max. Element in domain

Chart 1 shows the result comes after 9613 element in 23.5 mm X 89.80 mm size of domain from velocity remain study for more successive smaller cell size of 9812 and 10222.

IV. NO $_X$ MODEL

Formation and controlling NO x emission is that combustion is highly diverse and transient in C.I engines. While NO and NO2 are suffered together as NO x, there are some distinctive Differences between these two pollutants. NO is a colorless and odorless gas, while NO2 is a reddish brown gas with bitter odor. Both gases are considered as toxic; but NO2 has a Level of toxicity 5 times greater than that of NO. Although NO is largely formed from oxidation of NO, attention has been given on how NO can be controlled before and after combustion. NO is formed during the post flame combustion process in a high temperature region. The principal source of NO formation is the oxidation of the nitrogen present in atmospheric air. The nitric oxide formation chain reactions are initiated by atomic oxygen, which forms from the Dissociation of oxygen molecules at the high temperatures reached during the combustion process. The principal reactions governing the formation of NO from molecular nitrogen are,

$N_2 + O\_ NO + N$,

$N + O2\ \_\ NO + O$,

N + OH _ NO + H.

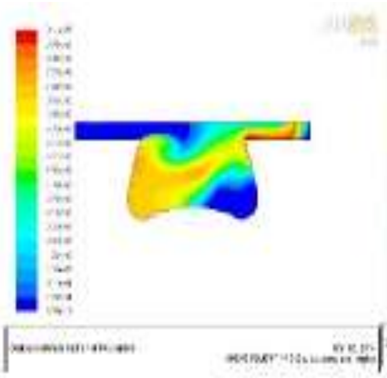
NO formed in the flame zone can be rapidly converted to NO2 via reactions such as,

NO + HO2 _ NO + OH.

Subsequently, conversion of this NO unless the NO2 to NO occurs via

NO_2+ O _ NO + O2

Formed in the flame is quenched by mixing with cooler fluid. This explanation is consistent with the highest NO_2=NO ratio occurring at high load in diesels, when cooler regions which could quench the conversion back to NO are extensive.



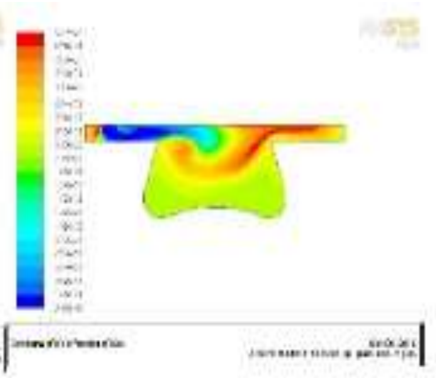
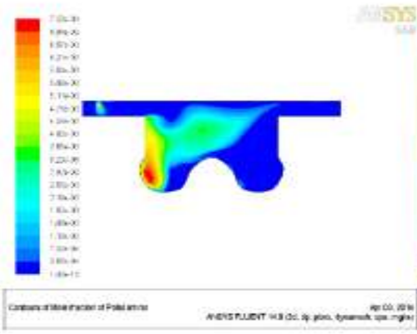


Figure 7 No mole Fraction for modified geo. No 1   **Figure 8** H_2O mole Fraction for modified geo. No 1



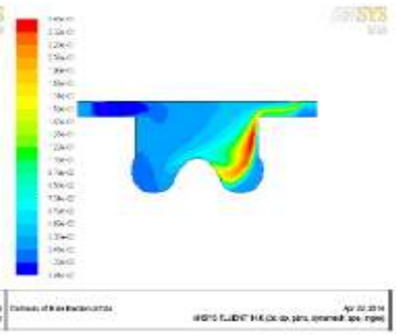


Figure 9 No for modified geo. No 2   **Figure 10** H_2Ofo modified geo. No 2

V. VALIDATION OF CFD RESULT WITH EXPERIMENT RESULT

Using NO x model of ANSYS FLUENT and CFX, applying appropriate input in boundary condition getting output at exhaust magnitude NO x. Software does not give the Direct result of it obtain from mole fraction of NO and H2O using equation (10)

Figure 11 Experimental set up

$$Noxppm = \frac{No\_Mol\_FractionX\ 10^{6}}{1 - H_2O\_Mol\_Fraction} \quad \ldots\ldots\ldots\ldots (10)$$

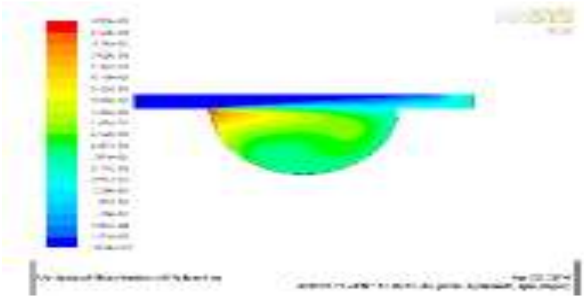


Figure 12 NO mole fraction for Baseline

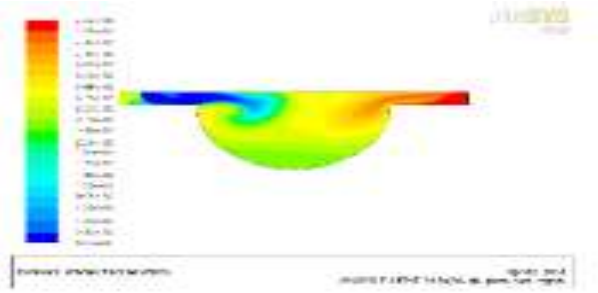


Figure 13 H_2O mole fraction for Baseline geometry

5.1 Comparison with experimental result and CFD result
Figure 12 show the counter value is 4.98×10^{-3} and
Figure13 show counter value is 3.20×10^{-2}
From equation (10) calculating NOx in ppm.

$$Noxppm = \frac{(4.98X10^{-3}X10^{6})}{1-(3.20X10^{-2})}$$

$$NOxppm = 5144\,ppm$$

Table 3 CFD validation with experimental result:

Comparison On Emission Parameter	**E**xperimentally	CFD result % of Error **7.8 %**
NO_x	4213 ppm	5112 ppm

VI. SWIRLING AND TURBULENT KINETIC ENERGY INSIDE THE CYLINDER

Swirling and Turbulent kinetic Energy are generally generated by the angular momentum of the cylinder flow about each of the three orthogonal axes, it depends upon the piston's movement in the cylinder with different specific R.P.M. and the chemical reactions of gases inside the combustion chamber of a C.I engine.

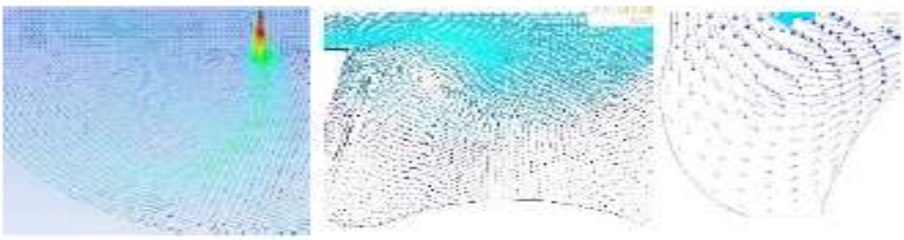
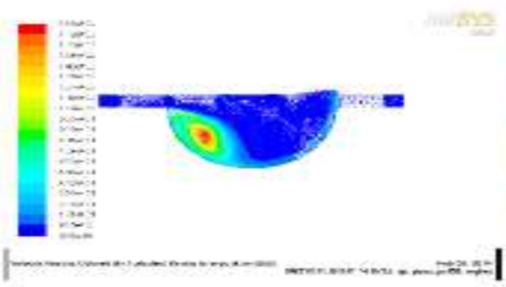


Figure 14 Swirling Counters for Baseline, modified geo. No.1 and Modified geo No.2

Figure 14 shows the different swirling effect in combustion domain at Top Dead Center position, it is observed that the modified geometry no 1 and 2 gives swirl on 3 to 5 mm upper sides of domain due to shape of cavity.

6.1 Turbulent kinetic energy inside the cylinder

The following figure 15 and 16 show the turbulent kinetic energy level during the suction and compression processes at different Crank angles for piston Geometry 1.Baseline 2.Modified Geometry No:1 and 3. Modified Geometry No: 2



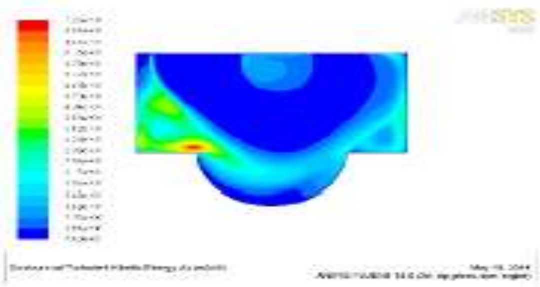
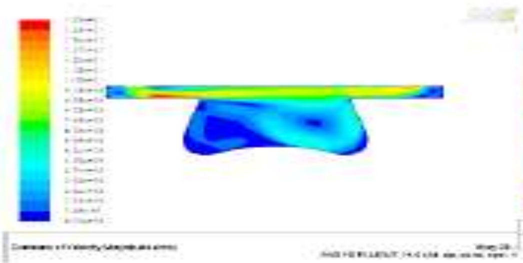


Figure 15 TKE at TDC (12 Degrees) Crank angle **Figure 16** TKE at 90 Degrees Crank angle

Figures 15 and 16 show the turbulent kinetic energy (TKE) is maximum at TDC position so it is predicted that all the calculation of NO x, velocity and TKE at TDC position for piston modified geometry no 1 and No 2 respectively.



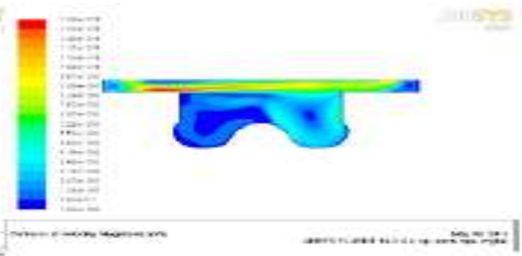


Figure 17 Velocity induced modified geo No.1 **Figure 18** Velocity induced modified geo NO_2

Figures 17 and 18 show velocity induced in baseline Geometry is 9.48 m/s, modified Geometry No 1 is 15 m/s and modified Geometry No 2 is 13.8 m/s at TDC.

VII.RESULT AND DISCUSSION

Modified geometry No 1 increases velocity 2.1 % compare to baseline geometry due to shape, Modified geometry No 2 decreases 13.1 % of NO x formation using proper inducing velocity and TKE For maximum power output. Therefore, it is observed that the geometry No 1 is suited for industrial use but for less formation of the Modified geometry No1 of piston results in higher velocity compared to the baseline and modified geometry No 2. Shapes considered in this study, modified geometry No1 of piston results in higher velocity compared to the baseline and modified geometry no 2. From the emission parameter point of view it is better to use geometry No 2 of piston configuration.

.

VI. CONCLUSION

The Study concludes following particulars: Research and Development in the field of I.C engine mainly emphases the maximum efficiency with minimum pollution emission parameters. This requires refinement of the cylinder internal flow, fuel mixture formation and combustion processes. The loss of energy is involved mainly due to two major factors: Mechanical loss due to friction (62-68 %) ,Thermal losses (27-32%): thermal loss mainly cylinder wall temperature loss due to atmosphere effect (25-34%) and other emission parameter loss (55-68%) due to existing unburned fuel of lean combustion chemistry. The use of modified geometry No 1 and 2 for selective engine combustion chamber result in maximum velocity, maximum volumetric efficiency and for minimum NO x production with improvement of overall efficiency by 2-4.5%.

REFERENCES

[1]. A.M. Indrodia (2014), Investigation of different combustion chambers geometry of diesel engine using CFD modelling of in cylinder flow for improving the performance of engine: (AIMTDR-2014)...
[2]. B.V.V.S.U. Prasad and C.S. Sharma, (2011), High swirl-inducing piston bowls in small diesel engines
[3]. For emission reduction, India Journal Applied Energy 88, 2355–2367, Bangalore 560012, India.
[4]. Charles E. Roberts and chad stvel. (2011), Advancement in Diesel Combustion System Design
[5]. To Improve the Stoke-BSFC Tradeoff. International Journal of Automotive Engineering vol. 2 pg.55 60, Texas78238 United States.
[6]. Ulugbek Azimov, Masahiro Okuno (2011), Multidimensional CFD simulation of syngas combustion in a micro-pilot-ignited dual-fuel engine using a constructed chemical kinetics mechanism.
[7]. International Journal of hydrogen energy, 700-8530, vol.36. Okayama, japan,
[8]. C.D. Rakopoulos and G.M. Kosmadakis, (2010), Investigation of piston bowl geometry and speed
[9]. Effects in a motored HSDI diesel engine using a CFD against a quasi-dimensional model. Energy
[10]. Conversion and Management vol.51, page 470–484. , Athens, Greece.
[11]. Jayashankara B, Ganesan V. (2010), Effect of fuel injection timing and intake pressure on the
[12]. Performance of a DI diesel engine – a parametric study using CFD. Energy Convers Manage
[13]. 51(10):1835–48.
[14]. Payri F, Benajes J, Margeo X, Gil A. (2004), CFD modeling of the in-cylinder flow in direct-injection
[15]. Diesel engine. Computational Fluids 33:995–1021.
[16]. Zhu Y, Zhao H, Melas DA, Ladommatos N. (2004), Computational study of the effects of the re-entrant lip shape and toroidal radii of piston bowl on a HSDI diesel engine's performance and emissions. SAE Paper 2004-01-0118.
[17]. Arturo de Risi. (2003), Optimization of the Combustion Chamber of Direct Injection Diesel Engines. 73100, society of automotive engineers, vol 2003-01-1064., Lecce Italy.
[18]. Wickman DD, Senecal PK, Reitz RD.(2001), Diesel engine combustion chamber optimization using
[19]. Genetic algorithms and multi-dimensional spray and combustion modeling. SAE Paper no. 2001-01-0547.
[20]. De Risi A, Manieri DF, Laforgia DA. (1999), Theoretical investigation on the effects of combustion chamber geometry and engine speed on
[21]. Soot and NOx emissions. In: Proceedings of ASME 1999 fall technical conference, ICE-vol. 33/1; p.51–
[22]. Shyy W. (1994), Computational modeling for fluid flow and interfacial transport. Amsterdam: Elsevier Science.