International Conference on Green Energy and Environmental Engineering (GEEE-2014)
ISSN: 2356-5608
Sousse, Tunisia - 2014

Numerical Study of Aero-thermal Flow in a Diesel Engine Using a CFD Approach

Noureddine Benyahia [#1], Kamel Talbi [*2]

[1,2]*Laboratory of applied energy and pollution, LEAP*

[#]*Mechanical Engineering Department, University of Constantine1, Algeria*[1, 2]

[1]nrd.benyahia@gmail.com

[2]kam.talbi@gmail.com

***Abstract* - the heat transfer and the fluid flow within the cylinder of an internal combustion engine have a very important role to the efficiency of the engine; the turbulent motion of the compressible fluid is connected directly with the air-fuel mixture and the combustion process. In this field, the experimental studies are always costly, therefore the choice of CFD (computational fluid dynamic) software tool became indispensable for the engine development, the design and the engine emission reduction, and also it allows us to visualize the physical phenomena inside the cylinder in record time.**

In this work, a numerical simulation of the aero-thermal flow is made for a (DI) diesel engine using the (CFD) software fluent to calculate the velocity field, the pressure and temperature distribution within the cylinder. Moreover, a dynamic mesh is used for the moving organs; valves and piston, the complex design of the engine geometry is made in a 3D model. The model of turbulence is chosen as K- epsilon model in this work, the results are used to analyze the physical phenomena inside the cylinder.

***Keywords*- Diesel engine, CFD, heat transfer, numerical simulation, fluent, turbulence, swirl.**

I. INTRODUCTION

In internal combustion engines, the fluid flow and the heat transfer within the cylinder play an important role in the engine efficiency; the compressible flow is unstable and turbulent, so that it is connected directly with the process of the air - fuel mixture and the combustion process. Many researchers have studied this phenomenon, either by experimental studies that are still expensive in this area or numerical studies on using the CFD simulation tools, the use of CFD numerical simulation techniques allows us to visualized the detailed physical phenomena; fluid flow, turbulence, heat transfer inside the cylinder, they also have the advantage of reducing the cost and speed of results. There are two different types of the internal combustion engine, (SI) spark ignition engine and (DE) diesel engine (1). In the diesel engine and, the increase of temperature and pressure of the air during the compression is sufficient to cause spontaneous ignition of the fuel. As Heywood [1] mentioned in his book, the air turbulence generated is better for air -diesel fuel mixture, so a good uniform mixture for combustion. Ideally, the generated turbulent mixing is sufficient to burn the fuel completely. Hence turbulence at the end of the compression stroke must be increased by the fluid velocity and the vortex structures in the cylinder caused by the movement of the piston during the intake and compression phase, and also by the movement produced in some geometries combustion chamber near the TDC of the compression phase.

The analysis of fluid flow and heat transfer in the cylinder is performed with two different methods: experimental methods with application of display technique LDV and PIV with a direct measurement and numerical methods. Among several experimental studies ; Chan and Turner [2] have used LDA technique with an optical access has been provided in the top and side walls of the cylinder study, the experimental data show the complex nature of a three-dimensional flow in the engine cylinder during the intake and compression phase, in a similar study, Kang and Baek [3] studied the characteristics of the turbulence of the vortex flow in a four-valve engine using the method of LDV, the results showed that the crankshaft speed causes the increase of the turbulence intensity and its homogeneous distribution during the compression stroke , Also Reeves et al [4] and Huang [5] performed the change of flow structure for different angles in the crankshaft having clear images for topological flow, swirled structures in the cylinder, PIV using the method using a particle image velocimetry during the intake phase and compression.

In this Centex, many numerical studies on the fluid flow and heat transfer in the cylinder with application of CFD tools, we find several commercial programs, such as (FLUENT, KIVA, and so on) or personal codes in (C++ ,Fortran), using the finite volume method and finite element . Laramée et al [6] has noted that visualization is an important element to analyse and present the results of a CFD simulation, they demonstrates that one can distinguish two types of movements in the diesel engine, swirl motion where the flow rotation axis is substantially coincident with the axis of the cylinder and the tumble motion where the flow rotation axis is perpendicular to the cylinder axis. Akar [7] made a numerical simulation to measure the caractestics flow in the engine with a two-dimensional, three-dimensional and a dynamic model using FLUENT CFD solver, they showed that the structure of the fluid flow is greatly affected by the position and length of the intake valve . Also Song and others [8] in 3D analyze of the fluid flow in a cylinder direct injection diesel engine for different chambers combustion, showed that the geometries of the piston have a significant effect on the swirl ratio and velocity profiles. Milton and others[9] indicate in a numerical simulation that the CFD industrial software fluent give a good accord with experience Hong et al [10] have used CFD simulation code KIVA -3 to analyze the languor scales of turbulence in the cylinder engine, they introduced a (LDV) system, they found that the movement of the fluid is very complex and turbulent, it is subjected to separation and recirculation, in comparison of the results they have observed that the code simulation , KIVA -3 shows a reasonable level and an agreement for such a complex flow field.

To view the actual situation of the engine, there is some turbulence models were used as k-ε, LES, RSM; which are the most used turbulence models for accurate results in simulations of the engine. Sukegawa al [11] have developed a program who treats the turbulent in-stationary flow in detail using DNS Quasi-direct-numerical-simulation, the results obtained gives a good agreement compared with experimental results**.** Wu and Perng [12] and Moureau Angelberger [13] have used LES model for 3D CFD simulation to studied an analyzed the fluid flow and heat transfer in the IC engines. Bilgin [14**]** and Chiavola [15] have used in a numerical study with different multidimensional modeling methods to provide a better understanding of the complex phenomena that occur within the internal combustion engine, they show that the movement of the fluid is cyclic and very complex because of the turbulence moving. Recently**,** Payri and other [16] made a three-dimensional simulation of fluid flow and heat transfer in an direct injection diesel engine with 4 valves, the flow field was calculated for three different combustion chambers and compared with the LDA measurements during the intake and compression phase, the comparison shows that the three-dimensional CFD model with standard k-ε turbulence model is reasonably accurate for crank angles, the results confirmed that the geometry of the piston had little influence on the flow in the cylinder during the intake stroke and the first portion of the compression stroke.

.

In this study, numerical simulation tridimensional was performed to analyze the heat transfer and fluid flow in a cylinder of a diesel engine; the numerical calculations were performed with the CFD commercial code FLUENT 6.3, turbulence model k -ε (RNG) chosen for the resolution of the compressible fluid, a dynamic mesh model was used for the piston stroke in the intake and compression phase.

We interested in this study to visualize the turbulent flow with three engine speed; n=1800, 2300, and 2800 rpm. The velocity profile, the temperature distribution, is represented with different crankshaft angles.

II. Numerical simulation

A. Governing equation

The calculation of the temperature and flow field in a cylinder and combustion chamber of internal combustion engine requires obtaining the solution of the governing equations. The Compressible, unsteady and turbulent in-cylinder flow can be described by differential equations of continuity, momentum, energy, turbulence kinetic energy and its dissipation rate. The Heat transfer regime is accepted as forced convection.

The mass conservation equation can be written as:

$$\frac{\partial \rho}{\partial t} + \rho \frac{\partial u_i}{\partial x_i} = 0 \qquad (1)$$

The momentum equation

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \upsilon \frac{\partial^2 u_i}{\partial x_j^2} \qquad (2)$$

The energy equation written in terms of temperature as:

$$\rho c_P \frac{\partial T}{\partial t} + \rho c_P T \frac{\partial u_i}{\partial x_j} = \lambda \frac{\partial^2 T}{\partial x_j^2} \qquad (3)$$

The RNG *k*-ε turbulence model is derived from the Navier-Stokes equations using a mathematical technique called "renormalization group"(RNG) methods, also additional terms and functions will appear in the transport equations, the turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}\left(\rho k \overline{U_i}\right) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}\right) + P_k - \rho\varepsilon \qquad (4)$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}\left(\rho \varepsilon \overline{U_i}\right) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_k})\frac{\partial \varepsilon}{\partial x_j}\right) + C_{\varepsilon 1}\frac{\varepsilon}{k}P_k - C_{\varepsilon 2}^{*}\rho\frac{\varepsilon^2}{k} \qquad (5)$$

In RNG k–ε model, the turbulent or eddy viscosity concept, and calculation of turbulent viscosity **μt** according to Prandtl-Kolmogorov relation is given as:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \qquad (6)$$

The model constants and the auxiliary relations are:

$$C_\mu = 0.0845 \quad C_{\varepsilon 1} = 1.42\,;\, C_{\varepsilon 2} = 1.68\,;\, \sigma_k = \sigma_\varepsilon = 0.7178\,;\, \beta = 0.012$$

$$C_{\varepsilon 2}^{*} = C_{\varepsilon 2} + \frac{C_\mu \eta^3 \left(1 - \eta/\eta_0\right)}{1 + \beta\eta^3}, \quad \eta = \frac{Sk}{\varepsilon}; \quad and \quad \eta_0 = 4.38$$

And term S is given by:

$$S \equiv \sqrt{2 S_{ij} S_{ij}}\,, \quad S_{ij} \equiv \frac{1}{2}\left(\frac{\partial U_j}{\partial x_i} + \frac{\partial U_i}{\partial x_j}\right)$$

B. The engine model

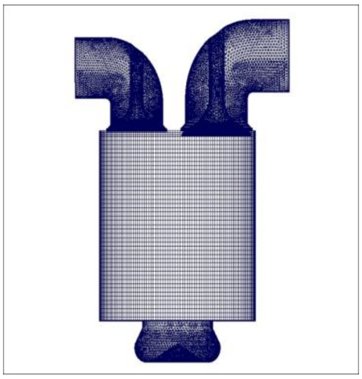


Fig1. Engine geometry in 3D

The complex design of the DI diesel engine is performed in 3D, with of a two-valve and a piston with a bowl ad combustion chamber, the engine characteristics are given in Table I.

Table I
Characteristics of the DI Diesel Engine

Bore	100 mm
Stroke	120mm
Compression ratio	19/1
Inlet valve diameter	45mm
Inlet valve lift	7.85mm
Exhaust valve diameter	38mm
Exhaust valve lift	8.00mm
Engine speed	1800,2300,2800 rpm
Combustion chamber	Piston with bowl

The mesh of the system was performed with the software Gambit as seen in Figure 1, The main problem of this work lies in the opening and closing of the valves movement and the displacement of the piston from the TDC at BDC. A dynamic mesh model was used in the fluent code to handle the complexity of the movement of the 3D mesh topology. The structure model is hybrid mesh configured with displacement of cellules, the total number of cells varies from 461,781 at TDC to 743,064 at BDC.

C. Calculation procedure

FLUENT is a CFD simulation code, using the finite volume method to solve the discretised Navier- Stokes equations, the code are based on the pressure-correction method using the Presto algorithm, the second order upwind difference scheme (UDS) is used to discretize the momentum, energy and turbulence equations. The temporal discretization is implicit on the variable time step depending on the stage of stroke

In each case, the calculation starts from the beginning of the intake phase at 0º Crankshaft TDC **fig-2** and ends at 340 ° which gives 160 ° after BDC in the compression stroke for different speeds. The intensity level of initial turbulence was set at 3%, which sufficient for circulation to consider turbulent. With a fluid inlet temperature T inlet = 300k. Near wall region is treated by using well known wall functions, based on the assumption of logarithmic velocity distribution.
The motion of the valves is linear, and the maximum valve lift is reached at 90 degrees crank angle.The visualization of results were presented as 2D (fig.3) , Plane 0 in 2D along the x axis center, the plans A, B and C are perpendiculars to the cylinder axis, with heights different respectively z = -0.001, -0.005 and0.009 m from the TDC, also a point **M** was chosen for the calculations, located in crossover between the plane 0 and y-axis with a height z = -0.002.

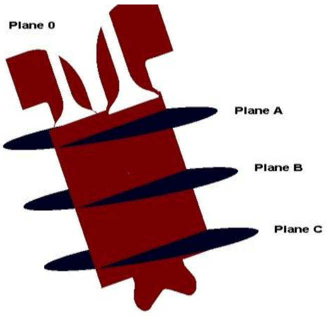


Fig.2. Considerate plans

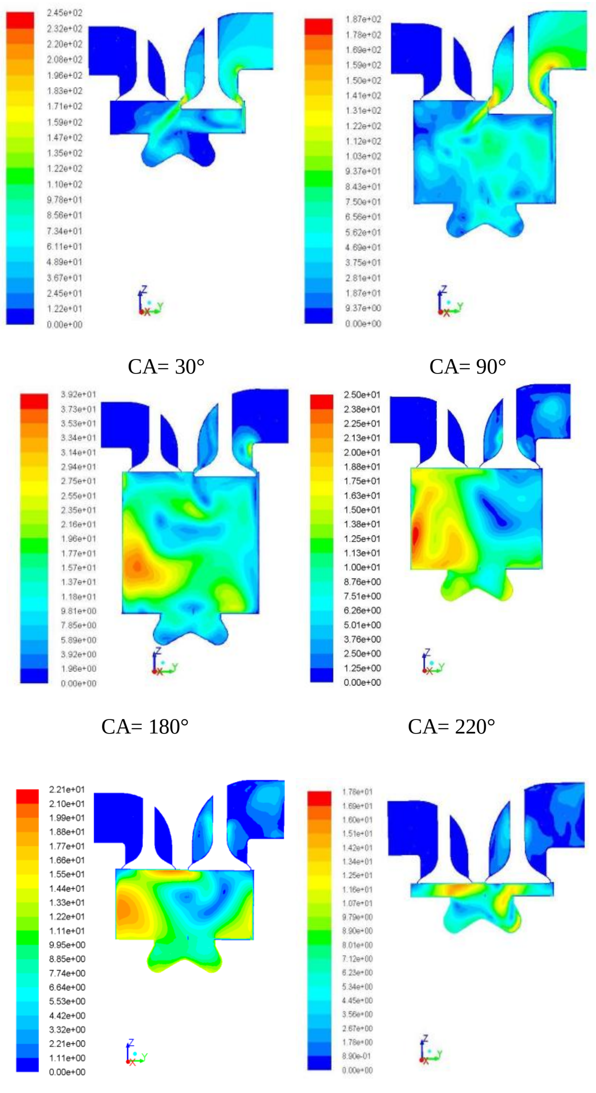
III. Results and discussion

A computational study has been performed in this work for different crank angles and revolution in the cylinder. In this part of the study, the results are present, pressure contours, velocity profiles and temperature contours.

International Conference on Green Energy and Environmental Engineering (GEEE-2014)
ISSN: 2356-5608
Sousse, Tunisia - 2014

The flow characteristics are visualized in 2D with 3 plans: plane A, plan B and C.
To take into consideration the variation of temperature we make a comparison between two cases: the (case 1) cold flow and the flow in motored condition (case 2) where:

- T inlet = 310 k
- T cylinder Wall = T piston = 430 k



CA= 30° CA= 90°

CA= 180° CA= 220°

CA= 290° CA= 340°

Fig.3. velocity field at different crank angle CA, n= 2300 rpm

Fig.3, presents the velocity field for different crank angle, n=2300 rpm, the visualization inside the cylinder is better than other models in literature. As showing us the fig3, the shape of the piston bowl does not play any significant role during the intake stroke, the flow field creates a highly non-homogeneous in the upper region of the cylinder, especially closer to the cylinder head, In this area, the velocity values and turbulence are obtained at the maximum valve lift when CA= 90°.

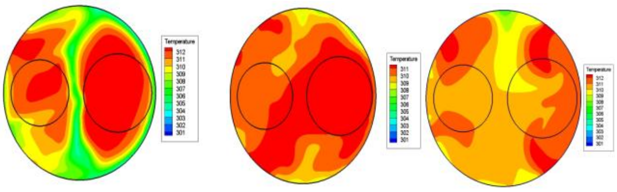


Fig.4. Temperature distribution in plane A, B and C, CA=180° and n=2300 rpm

The fig.4 shows us the temperature distribution inside the cylinder illustrated in plane B and C, the values are very close in the same CA=180° and n=2300 rpm the values are very close in the same CA=180° and n=2300 rpm. The temperature values of air are illustrated in diffident regions (plans) in the cylinder where we observe that the temperature is homogeneous at the beginning of the compression phase.

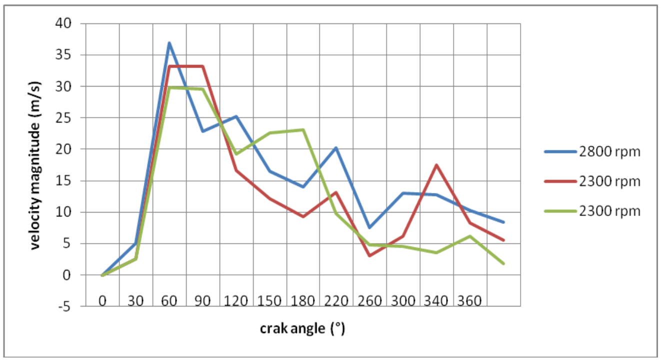


Fig.5. velocity variation with different crank angle in the point (M), n =1800, 2300, 2800 rpm

To study in detail the effect of engine speed during the intake and compression, three engine speeds were considered, the measurement shall be taken at the same point **(M),** the results presented in Figure-5- confirms that the flow field in the cylinder and perfectly adapted to the engine speed. This is especially true up to CA =60 ° of TDC in the intake course, later the rule is invalid contrary, the speed is reduced in the inlet layer from CA =70° to 180°in the intake, as seen in the graph at the top of the compression phase of the fluid velocity increases the speed with more average engine 2300 rpm, this is probably due to the higher kinetic energy due to the increase in engine speed.

The temperature distribution presented in fig.6, for CA=180°. The results are taken along the y-axis and Z=-0.002 from TDC, Temperature distribution is obtained symmetrically for the tow cases, the values of temperature is increase due to heat transfer in motored conditions, it reaches 345 K at the end of the intake stroke.

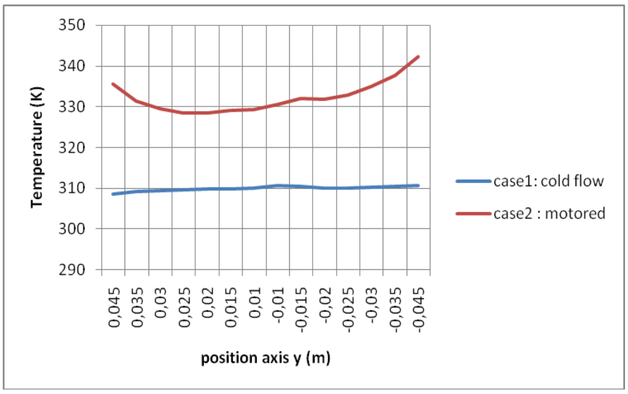


Fig.6. comparison of Temperature distribution between the cold flow and motored conditions flow, along y-axis, with z = -0.002 from TCD

IV. Conclusion

This study show the effect of revolution of engine on the fluid flow and the temperature field inside the cylinder of a diesel engine during the intake and compression strokes, the CFD simulation with the RNG k–ε turbulence model gives an acceptable results of the heat transfer where we find that the temperature of the fluid increases with increasing of crank angle but decreases with engine revolution. The good accurate of phenomenal inside the engine allow us to improving the knowledge on the air flow characteristics, which is linked with the combustion process and ICE development.

Further work

For the future study, we treat the number of swirl and tumble, and heat transfer with different combustion chamber geometry.

References

[1] HEYWOOD, J.B., 1998. Internal Combustion Engine Fundamentals, *McGraw-Hill, New York.*

[2] CHAN, V.S.S. and TURNER, J.T., 2000. "Velocity Measurement Inside a Motored Internal combustion Engine Using Three-Component Laser Doppler Anemometry" *Journal of Optics and Laser Technology, Vol. 32, pp. 557-566.*

[3] KANG, K.Y. and BAEK, H., 1998. "Turbulence Characteristics of Tumble Flow in a Four Valve Engine" *Journal of Experimental Thermal and Fluid Science*, Vol. 18, pp. 231-243.

[4] REEVES, M., TOWERS, D.P., TAVENDER, B. and BUCKBERRY, C.H., 1999. "A High Speed All-Digital Technique for Cycle-Resolved 2-D Flow Measurement and Flow Visualization within SI Engine Cylinders" *Journal of Optics and Laser in Engineering*, Vol. 31, pp. 247-261.

[5] HUANG, R.F., HUANG, C.W., CHANG, S.B., YANG, H.S., LIN, T.W. and HSU, W.Y., 2005."Topological Flow Evolutions in Cylinder of a Motored Engine during Intake and Compression Strokes" *Journal of Fluids and Structures*, Vol. 20, pp. 105-127

[6] LARAMEE, R.S., HAUSER, H., DOLEISCH, H., POST, F.H., 2004. "The State of the Art in Flow Visualization: Dense and Texture-Based Techniques" Computer Graphics Forum, Vol. 23, pp. 203-221.

[7] R. Akar, Combustion chamber design with computational fluid dynamic, Inst. Nat. Appl. Sci., MSc Thesis, 2005

[8] J. Song, C. Yao, Y. Liu, Z. Jiang, Investigation on flow field in simplified piston bowls for DI diesel engine, *Eng. Appl. Comp. Fluid Mech. 2* (2008) 354–365.

[9] MILTON, B.E., BEHNIA, M. and ELLERMAN, D.M., 2001. "Fuel Deposition and Re-Atomization from Fuel/Air Flows Through Engine Inlet Valves" *International Journal of Heat and Fluid Flow*, Vol. 22, pp. 350-357.

[10] HONG, C.W. and TARNG, S.D.1998. "Direct Measurement and Computational Analysis of Turbulence Length Scales of a Motored Engine" Journal of Experimental Thermal and Fluid Science, Vol. 16, pp. 277-285.

[11] SUKEGAWA, Y., NOGI, T. and KIHARA, Y., 2003. "In-Cylinder Airflow of Automotive Engine by Quasi-Direct Numerical Simulation" Journal of JSAE Review, Vol. 24, pp.123-126

[12] WU, H.W. and PERNG, S.W., 2002. "LES Analysis of Turbulent Flow and Heat Transfer in Motored Engines with Various SGS Models" International Journal of Heat and Mass Transfer, Vol. 45, pp. 2315- 2328

[13] V. Moureau, C. Angelberger, Towards Large Eddy Simulation in Internal Combustion Engines: Simulation of a Compressed Tumble Flow, SAE 011995, 2004.

[14] A. Bilgin, Numerical simulation of the cold flow in an ax symmetric non compressing Engine like geometry, *Int. J. Energy Res*. 23 (1999) 899–908.

[15] CHIAVOLA, O. 2002."Multi Dimensional CFD Transmission Matrix Modeling of IC Engine Intake and Exhaust Systems" *Journal of Sound and Vibration*, Vol. 256, pp. 835-848.

[16] PAYRI, F., BENAJES, J., MARGOT, X. and GIL, A., 2004. "CFD Modeling of the In-Cylinder Flow in Direct-Injection Diesel Engines" Journal of Computers and Fluids, Vol. 33, pp. 995-1021.