CFD Analysis of Petrol Internal Combustion Engine

Mahmoud A. Mashkour   Mustafa Hadi Ibraheem

Mechanical engineering Department University of Technology, Baghdad-Iraq

MahmoodMashkoor@hotmail.com   abc_logo@yahoo.com

Submission date:- 23/7/2018	Acceptance date:- 16/9/2018	Publication date:- 14/10/2018

Abstract

Optimizing operation for internal combustion engines requires the application of advanced essential technique. Moreover, an experimental investigation, numerical 3D CFD simulation, is needed in order to obtain and investigate a vision into the complex phenomena's within cylinder. In this paper, fluid flow inside a single cylinder of spark ignition engine (SI) Hyundai type was modeled depending on the numerical simulation using ANSYS V15.0/ICE CODE, with dynamic mesh technique to study and estimate the characteristics flow under normal operation of octane fuel with respect to crank angle at a constant r.p.m. The engine model was done by SolidWorks environment. This work focused on the simulation of the intake, compression, expansion and exhaust stroke, including cold and combustion simulation, solving the governing equations (continuity, Renolds Average Navier Stoke, and energy equation). The code was validated against published data for present case, and the comparison showed a close agreement between the results and the maximum discrepancy was 17 %**.**

Key words: SI Engine, 3D CFD, Flow characteristics, ICE code, Dynamic mesh.

Nomenclature

SI	Spark Ignition Engine.
r.p.m	Revolution per Minutes.
GDI	Gasoline direct injection.
SR	Swirl ratio.
TDC	Top Dead Center.
BDC	Bottom Dead Center.
ATDC	After Top Dead Center.
BTDC	Before Top Dead Center.
θ_c	Current crank angle
θ_s	Start angle
t	Time (sec.)
Ω	Shaft speed (rpm)
θ event	Event crank angle
n	Integer
θ_{period}	Crank angle period
P_s	Piston Position
A	Piston Stroke (mm)
L	Connecting Rod Length (mm)
ρ	Total mass density (kg m^{-3})
u	Velocity (m/sec.)
τ	Stress tensor
φ	Viscous dissipation function
S	Source term
P	Pressure (Pa)
k	Thermal conductivity (W/m.k)
h	Heat transfer coefficient (W/m^2.k)

1. Introduction

The internal combustion engine (ICE) is considered more important machine that used to provide mechanical power by converting the fuel power, this takes place with many complex processes. The modeling of internal combustion engine (ICE) represents one of the highest level of complexity and a challenging task. This is due to more parameters that play a key role of operation such as, fuel injection, flame propagation, ignition process, and variety of fluid properties due to unsteady state. Therefore, the study and development of internal combustion engines need to characterize deeply what happens from the complex physical phenomenon within engine, especially with the advancement of technology.

The flow characteristic plays one of vital roles in the engine performance. So, several researchers have been involved in the study of ICE, Heywood [1] mentioned that the air turbulence is better for air - fuel mixture. Kuo [2] predicted the gas pressure changes within the cylinder by a simple thermodynamic principle model. Most significant weakness of this model is that it does not take the changes in fuel reactivity during combustion into account. Movva [3] simulated the fluid flow in combustion chamber of ICE by wave action simulation, with FORTRAN code, and obtained the pressure distribution, and velocity at variant boundary condition. Omaraa [4] studied the cold flow in a dual engine, used fluent 2 diamensional transient simulation, and obtained pressure, velocity, and temperature with crank angle by segment analysis into specific angles. Morauszki et.al [5] investigated the fluid properties in GDI engine with numerical computation using ANSYS and shows flow pattern, pressure during compression and power stroke. Ravichandra et.al [6] converted CI into SI by using dynamic mesh technique with fluent 16.0 for engine numerically and experimentally, recording the temperature and pressure inside combustion chamber. Abdul Rahiman et.al [7] studied an intake manifold by CFD and showed the variation of Swirl Ratio (SR) inside the cylinder with respect to crank angle for different manifolds.

This paper represents a modern method in terms of modeling of geometry and 3D simulation by using the finite volume method. The objective of this research is to develop simulation to calculate the flow characteristic in combustion chamber of spark ignition engine by using dynamic mesh technique to visualize the flow within combustion chamber by using ICE code with engine speed 2500 r.p.m. The velocity, pressure and temperature contours will be studied with respect to different crank angles.

2. Methodology

2.1 CFD Tool

The design and manufacture of internal combustion engines are under significant pressure for improvement. The generation of engines requires being light, reliable, robust, flexible, and powerful. Innovative engine designs will be required for satisfying these requirements. The ability to accurately the performance of multiple engine designs is too much crucial and critical, because IC engines consist of complex fluid dynamic interactions between air flow, fuel injection, moving parts, and combustion. Using CFD results, the flow phenomena can be visualized on a 3D geometry and analyzed numerically, providing tremendous insight into the complex interactions that occur inside the engine. CFD simulation is used as a part of the design process in automotive engineering, especially with the rise of modern technology.

2.2 Modeling Geometry

The methodology of modeling geometry was done by selecting the actual model of Hyundai four cylinder - 4 stroke spark ignition engine type as shown in figure (1) and then the geometric model of engine was created by SolidWorks. In order to reduce the simulation, one cylinder of engine was carried out as shown in figure (2). More information about the engine specification listed in Table (1).

Table (1): Engine Specifications

Bore	71 mm
Stroke	60mm
Connecting rod length	126 mm
Compression ratio	8.2
Intake valve diameter	32 mm
Maximum intake valve lift	6.9 mm at 104 deg. ATDC
Intake valve opening	41 deg. BTDC
Intake valve closing	84 deg. ABDC
Exhaust valve diameter	26mm
Maximum valve lift	9.6 mm at 64 deg. ABDC
Exhaust valve opening	66 deg. BBDC
Exhaust valve closing	16 deg. ATDC

Figure (1): 4 - Line cylinder head

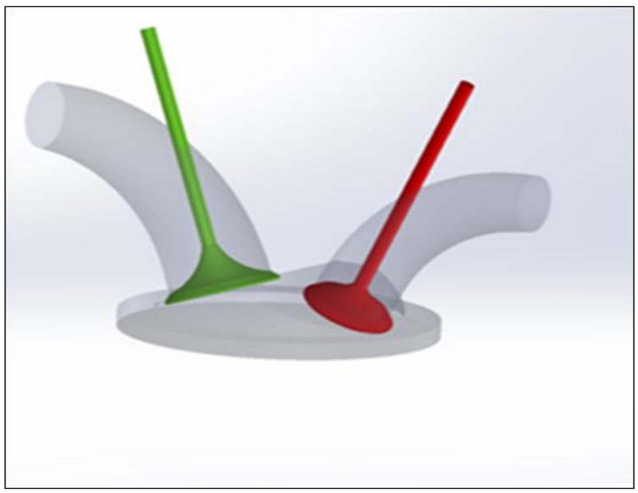


Figure (2): One cylinder model

2.3 ICE Code

This context was focused on the procedures of ICE step. The geometry was conducted by SolidWorks. The model was sub-divided into different zones according to the decomposition technique [8] as shown in figures (3, 4, and 5) for compatibility with mesh of system. The moving mesh was controlled by dynamic mesh events in equations (1), (2). The moving of the parts (piston and valves) with respect to the crank angle was governed in equation (3) and valve timing [8]. The computational software was responsible for grid generation and moving mesh through automatic dynamic addition and removal of cells, which facilitates control over mesh resolution and distortion in moving-boundary problem. In this work the mesh is deforming respect with time and crank angle. Therefore the mesh is stretch, breaks up and re-meshes. The number of cell is about 407.000.

The code uses the finite volume method to solve the discretized Navier- Stokes equations; it is based on the pressure-correction method by selected the second order upwind difference scheme.

$$\theta_C = \theta_S + t\,\Omega_{shaft} \tag{1}$$

$$\theta_{event} = \theta_c + n\theta_{period} \tag{2}$$

The events are specified for one complete engine cycle, according to the engine operation

$$P_S = L + \frac{A}{2}(1 - COS\theta_C) - \sqrt{L^2 - \frac{A^2}{4} sin^2\theta_c} \qquad (3)$$

P_s is equal to zero at TDC and equal to A at BDC.

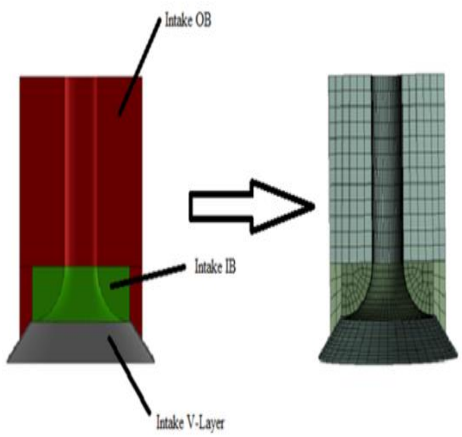


Figure (3): Decomposition of valves and meshed valve

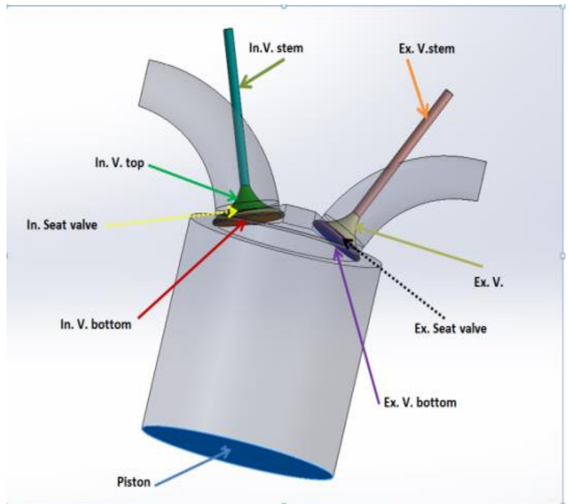


Figure (4): In–cylinder

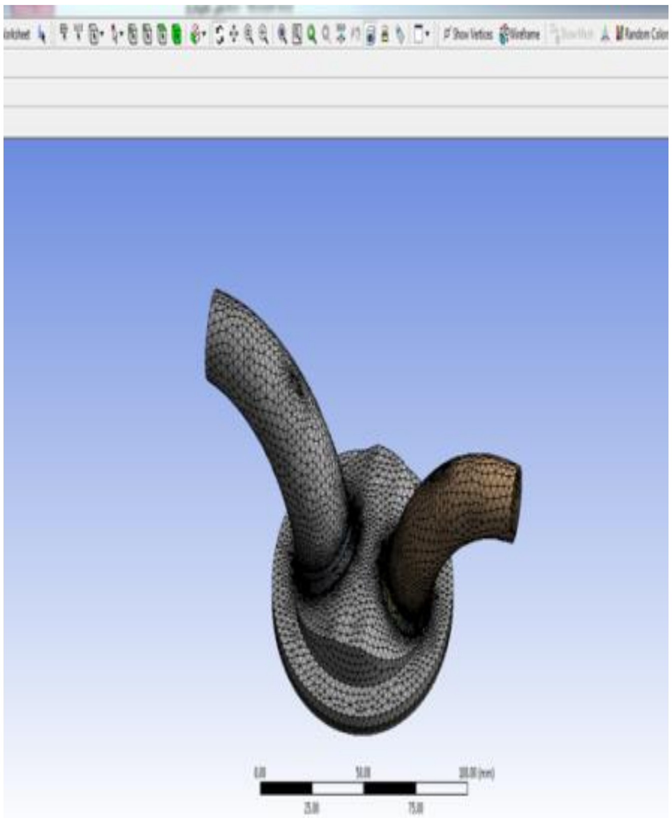


Figure (5): Mesh of model

2.4 Governing Equations

CFD is basically depending on the governing equations of fluid dynamics, the continuity equation for transient state, and k-epsilon turbulence 2-equations model were solved for the present investigation [6] as shown below. The strategy of solver that used in this paper shows in figure (6).

- **Continuity Equation**

$$\frac{\partial \rho}{\partial t} + div.(\rho u) = 0 \tag{3}$$

- **Momentum Equation**

$$\rho \frac{Du}{DT} = \frac{\partial(-\rho + \tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \tag{4}$$

- **Energy Equation**

$$\rho \left| \frac{\partial h}{\partial t} + div.(hV) \right| = -\frac{Dp}{Dt} + div(kgrdT) + \varphi \tag{5}$$

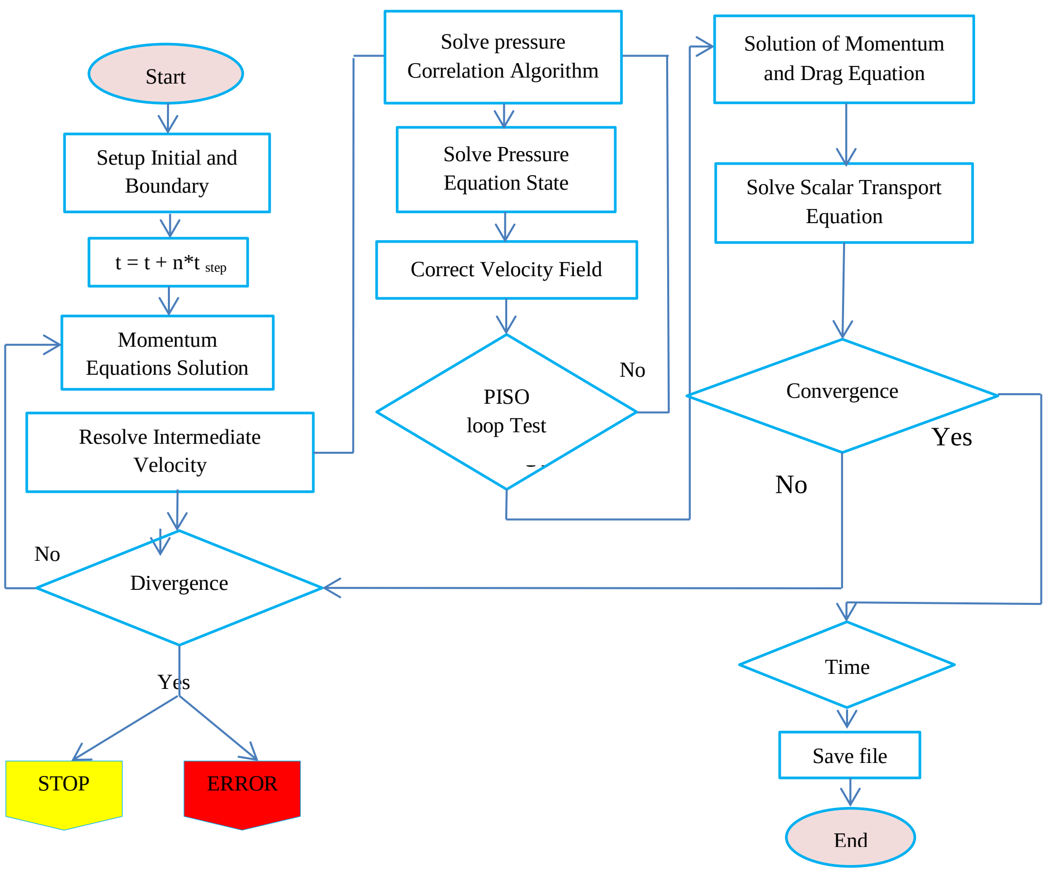


Figure (6): CFD solver flow chart

2.5 Boundary Conditions

The flow domain for this simulation is the combustion chamber,including the inlet valve,exhaust valve,piston ,also intake and exhaust port . The boundary conditions have been used refers to the pressure inlet boundary conditions , the pressure outlet boundary condition for the exit from the exhaust port. The intial condition are given in Table (2).

Table (2): boundary and initial conditions

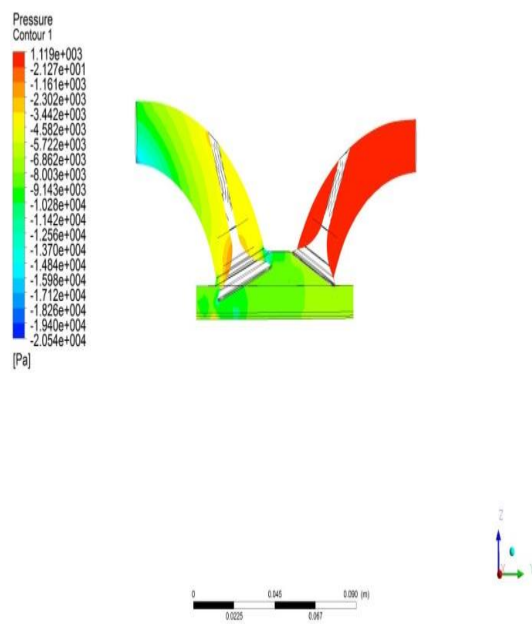
Variable (unit)	Initial value
Pressure (Pa)	Atmospheric pressure
Turbulent kinetic energy (m^2/s^2)	0.02
Turbulent dissipation rate (m^2/s^3)	0.02
Temperature (k)	300
Inlet pressure (Pa)	Atmospheric pressure
Exhaust pressure (Pa)	Atmospheric pressure
Cylinder wall temperature (k)	300
Piston wall temperature (k)	300

3. Results and Discussions

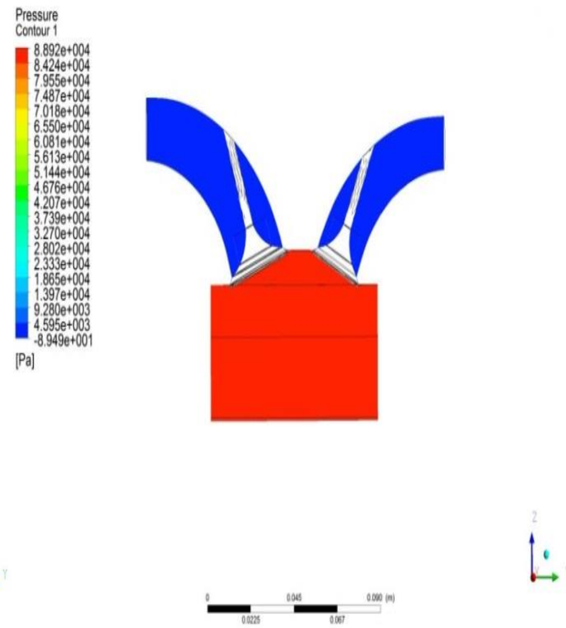
3.1 Cold

The numerical study has been achieved to identify the flow within the engine, so the simulation provided the pressure data for cold flow during the four strokes, as shown in figure (7) and (8).

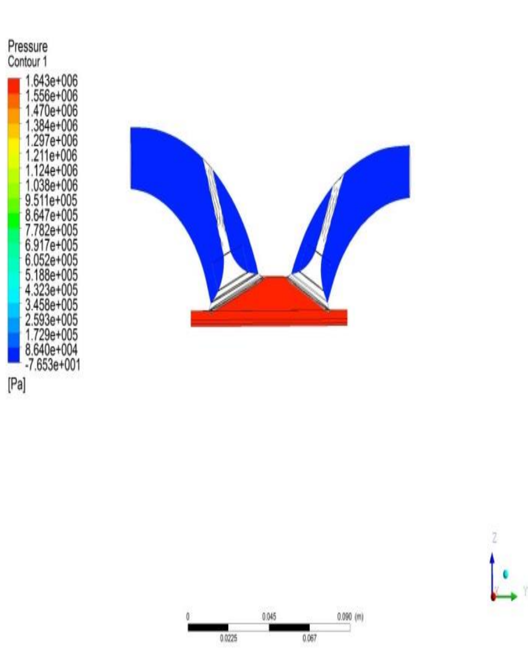
The result shows that the pressure is vacuum at intake stroke because the intake valve starts open at 360 degree, the piston moves from TDC toward down so the volume of combustion chamber is increased, and then pressure reduced. This seems obvious at 390 CA. Then when the piston moves up from BDC to TDC, the volume is decreased so the pressure increased, and then gradually increasing at the 2nd stroke reaching to maximum at the 3rd stroke at 720CA when piston is at TDC; it approximates 16 bar because the volume of combustion chamber is too much small and IVC also, EVC are closed at this moment. During the exhaust stroke, the code shows a decrease in the value of pressure according to the expanding at the end of the third stroke because piston moves towards the BDC, the flow blow down exhaust system.



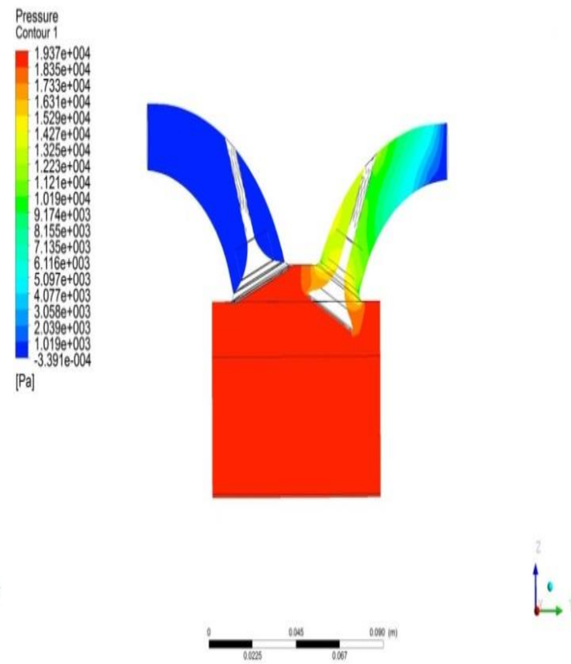
Pressure at 390 deg. Crank angle



Pressure at 630 deg. Crank angle



Pressure at 720 deg. Crank angle



Pressure at 990 deg. Crank angle

Figure (7): Pressure contours of cold flow

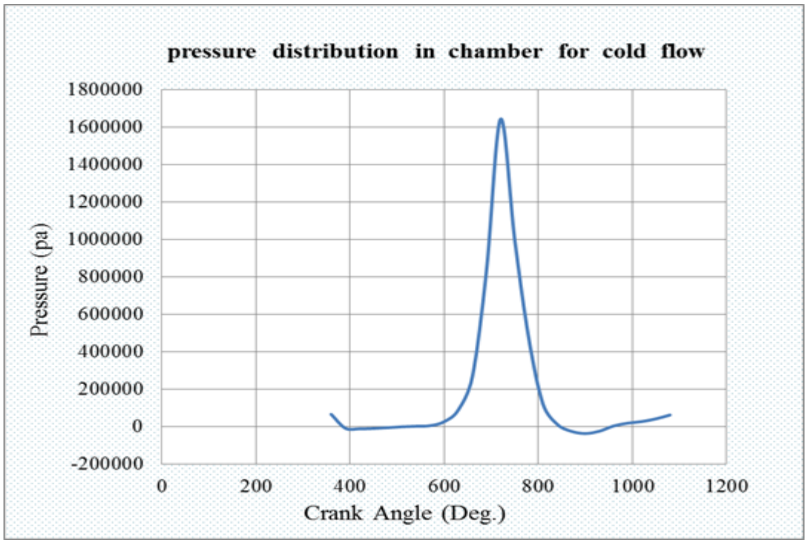


Figure (8): In –cylinder pressure trend of cold

In conjunction with pressure behavior during the engine operation, the temperature takes a similar behavior to the pressure, so the figure (9) and (10) shows that the temperature is reduced in the intake port because of the pressure vacuums in the intake port at suction stroke at 390,420,450 CA. The maximum temperature at 720 CA is 705K and then reduces to approximately 411K at 900 CA where exhaust valve is partially opened.

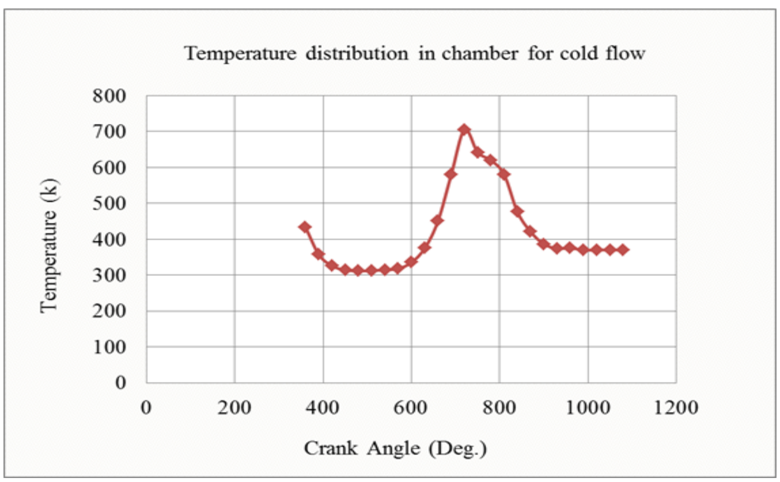
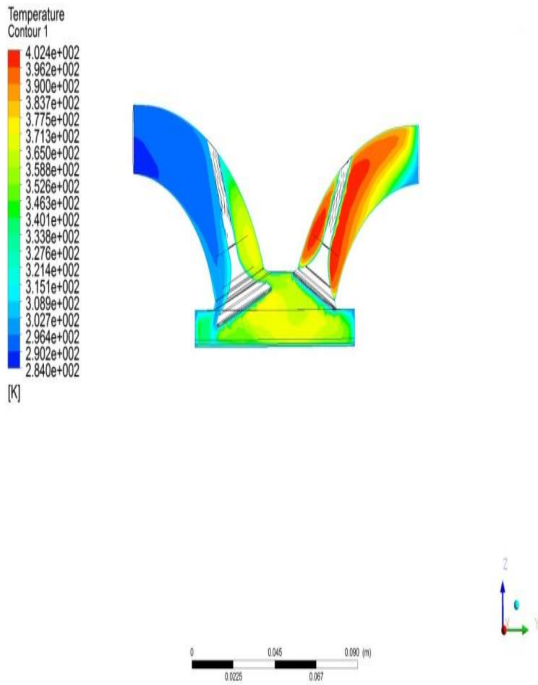
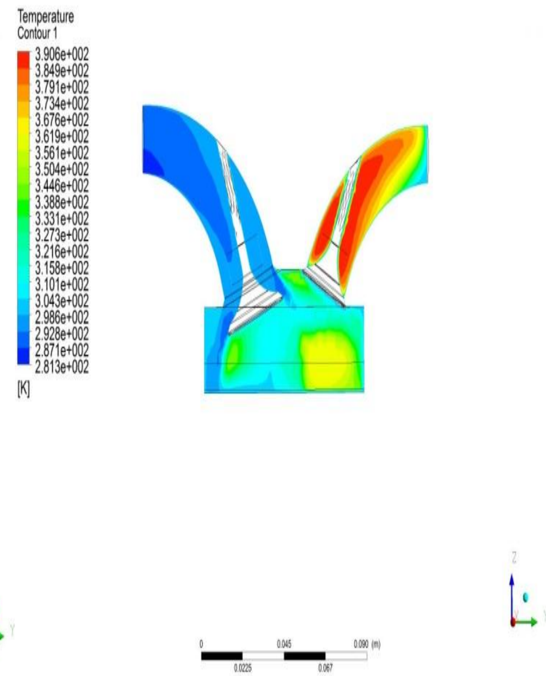


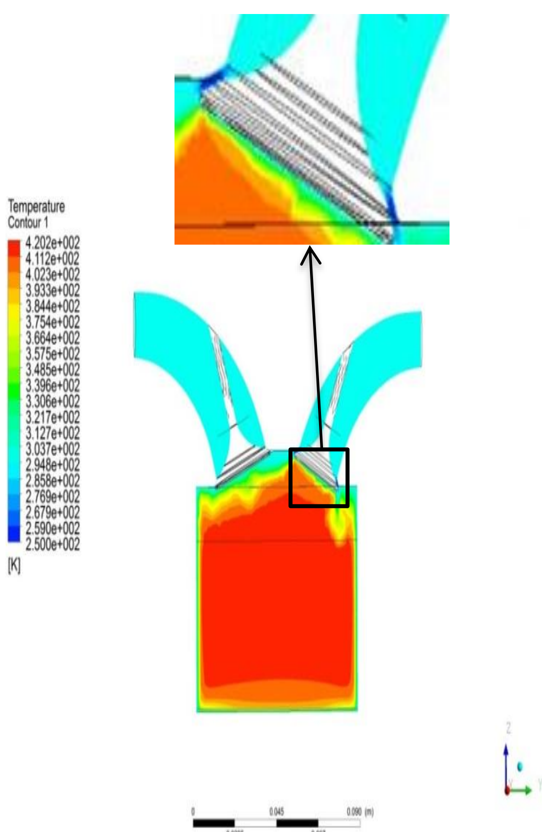
Figure (9): In –cylinder temperature trend of cold



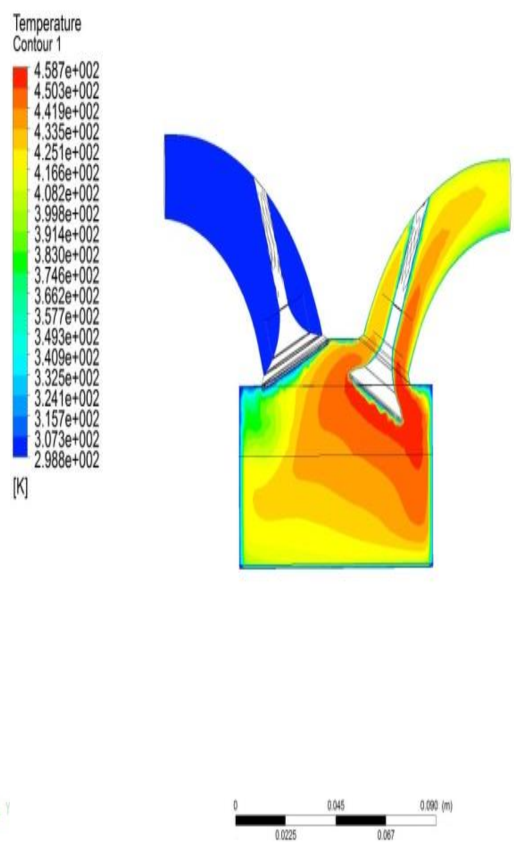
Temperature at 390 deg. Crank angle



Temperature at 420 deg. Crank angle



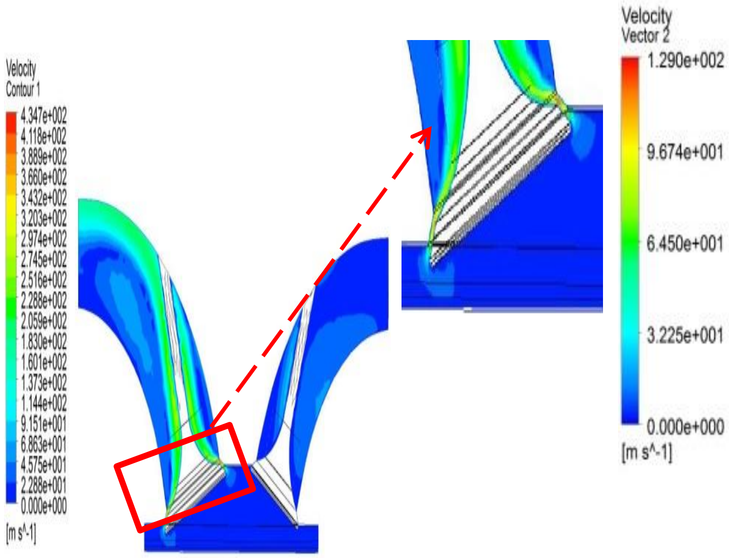
Temperature at 900 deg. Crank angle



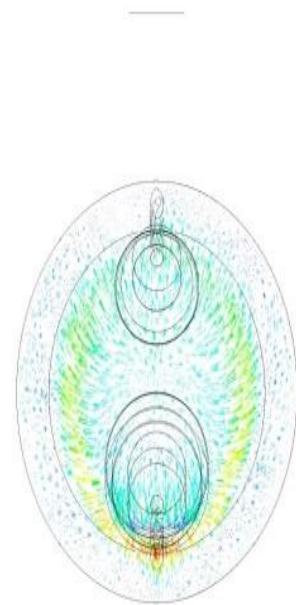
Temperature at 1020 deg. Crank

Figure (10): Temperature contours of cold flow

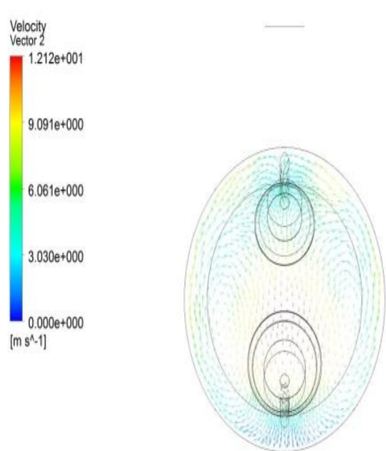
The investigation of velocity field in cold flow was showed in figure (11)and (12), that the velocity at 360 CA about 11m/s in chamber and its maximum at the same angle in the end of intake port , due to partial opening of intake valve (11-a). It is increases in the chamber until reach to maximum value at 450 CA (11-b), then reduces during 2nd and 3th as shown in (11-c, d). With coincidence with exhaust valve movement at 870 CA where the valve starts to open partially, according to valve profile the velocity inside the combustion chamber increases. And then the trend is referring to decreasing value of velocity during 4th stroke.



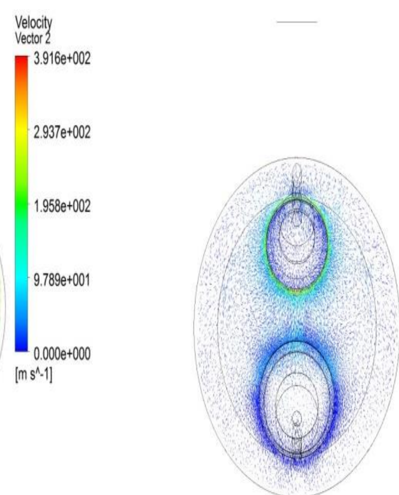
(a) Velocity of cold flow at 360 CA



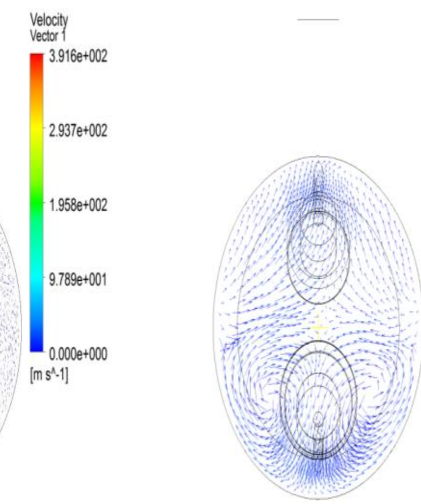
(b) Velocity of cold flow at 450 CA



(c) Velocity of cold flow at 630 CA



(d) Velocity of cold flow at 780



(e) Velocity of cold flow at 990 CA

Figure (11): Velocity contours of cold flow

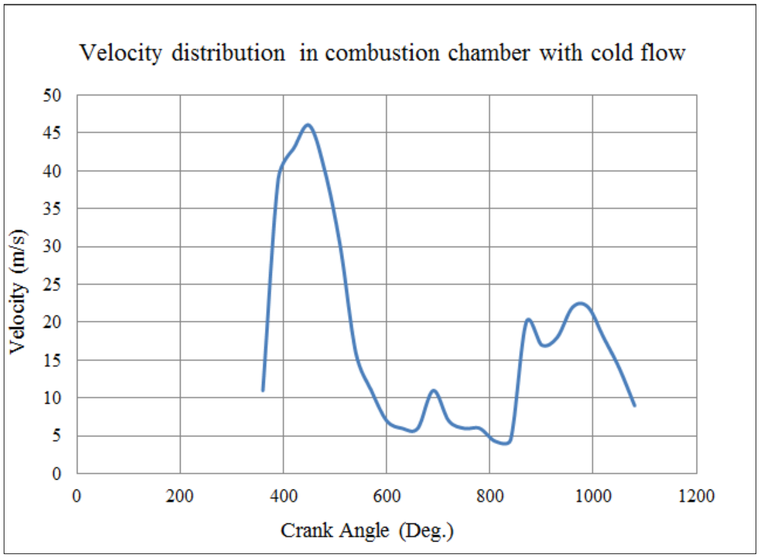


Figure (12): In –cylinder velocity trend of cold

3.2 Combustion

According to the valve timing, the fuel injection at the end of 2nd stroke, into chamber, prepares the mixture for ignition by spark. Then combustion occurs and the results shown in figure (13) and (14) reveals that the pressure increases to approximately to 77 bar at 720 CA.And, it reduces gradually due to the conversion the fuel power to mechanical power at power stroke. At 900 CA the exhaust valve is open to deliver the flow gasses to exhaust system, the pressure is 1.47 bar.

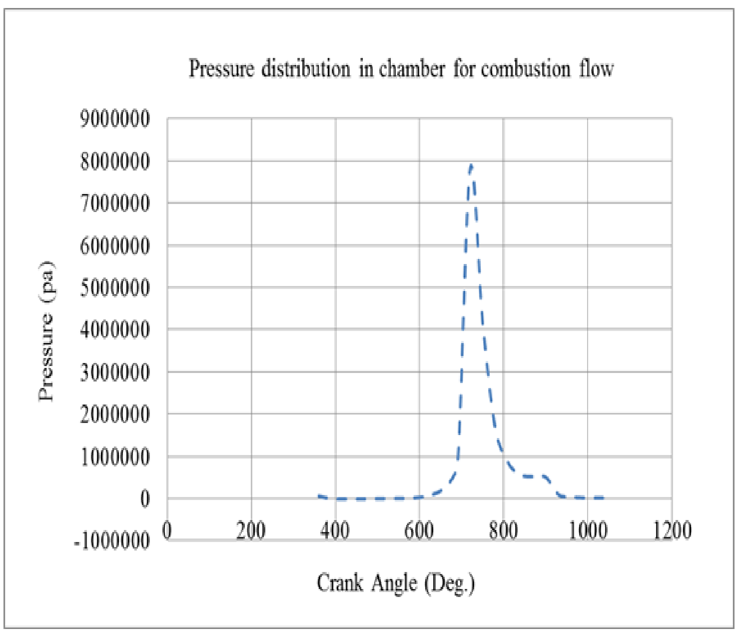
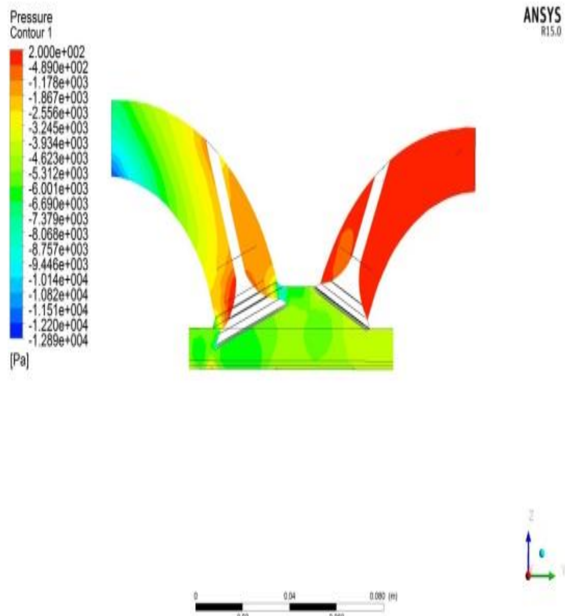
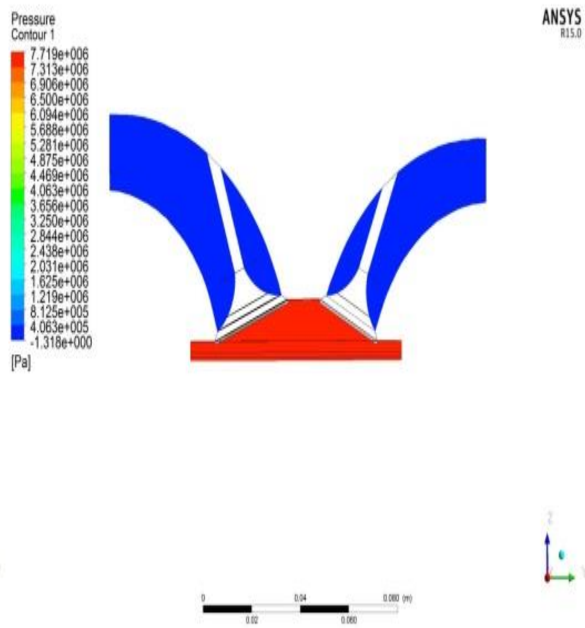


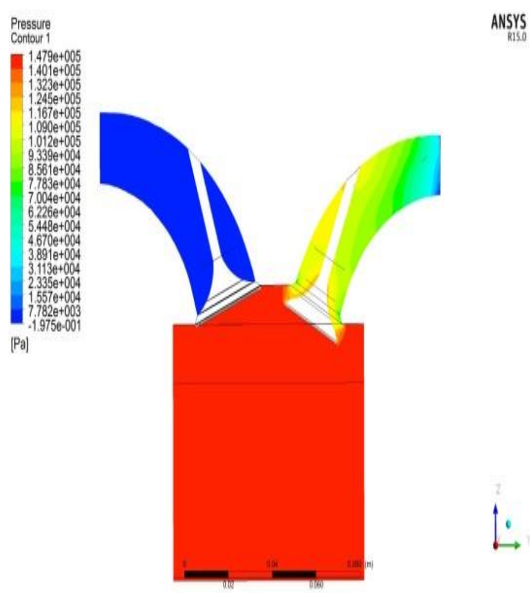
Figure (13): In –cylinder pressure trend at combustion simulation



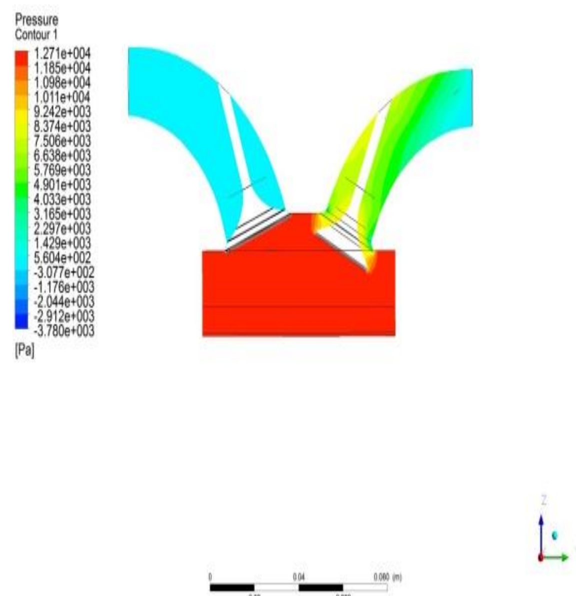
Pressure at 390 CA



Pressure at 720 CA



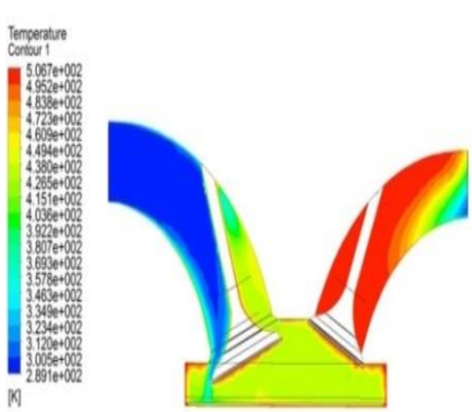
Pressure at 900 CA



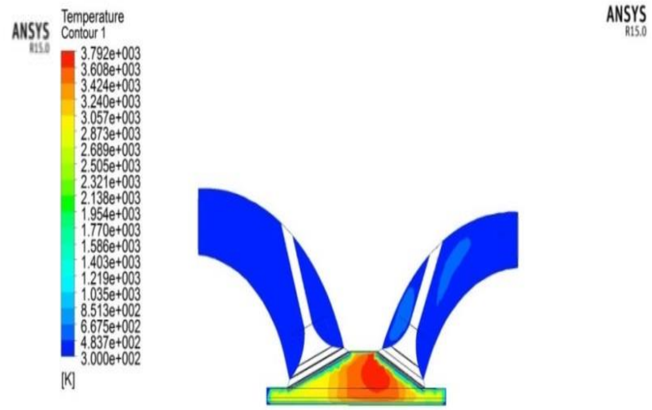
Pressure at 1020 CA

Figure (14): Pressure contours at various crank angles for combustion

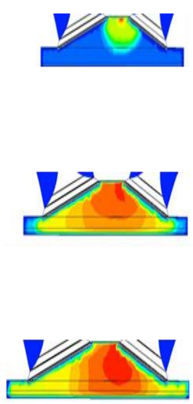
Figure (15) shows the temperature distribution at different crank angles, the temperature increases and the maximum value during the flame propagation as shown in the figure below due to combustion. Then, it reduces when volume of combustion chamber is increased due to expansion during moving the piston to BDC.



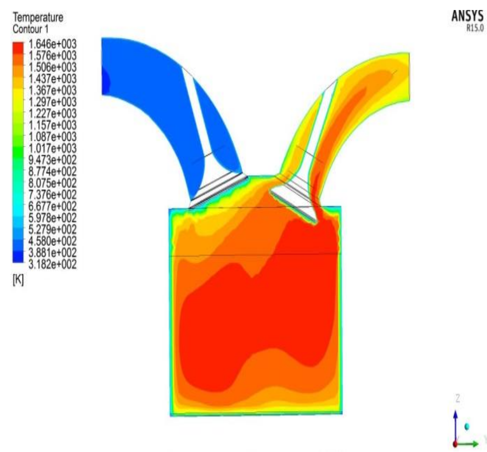
Temperature at 390 CA



Temperature at 720 CA



Flame propagation in combustion chamber



Temperature at 900 CA

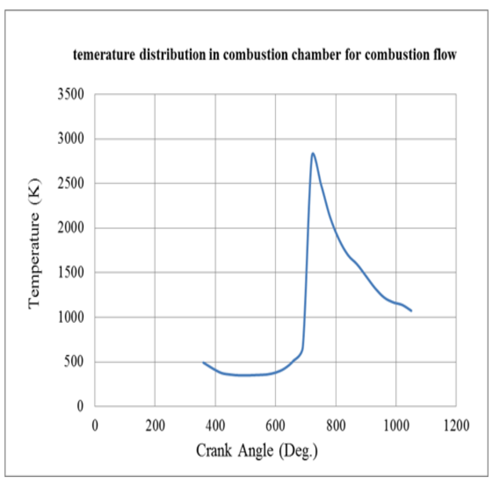
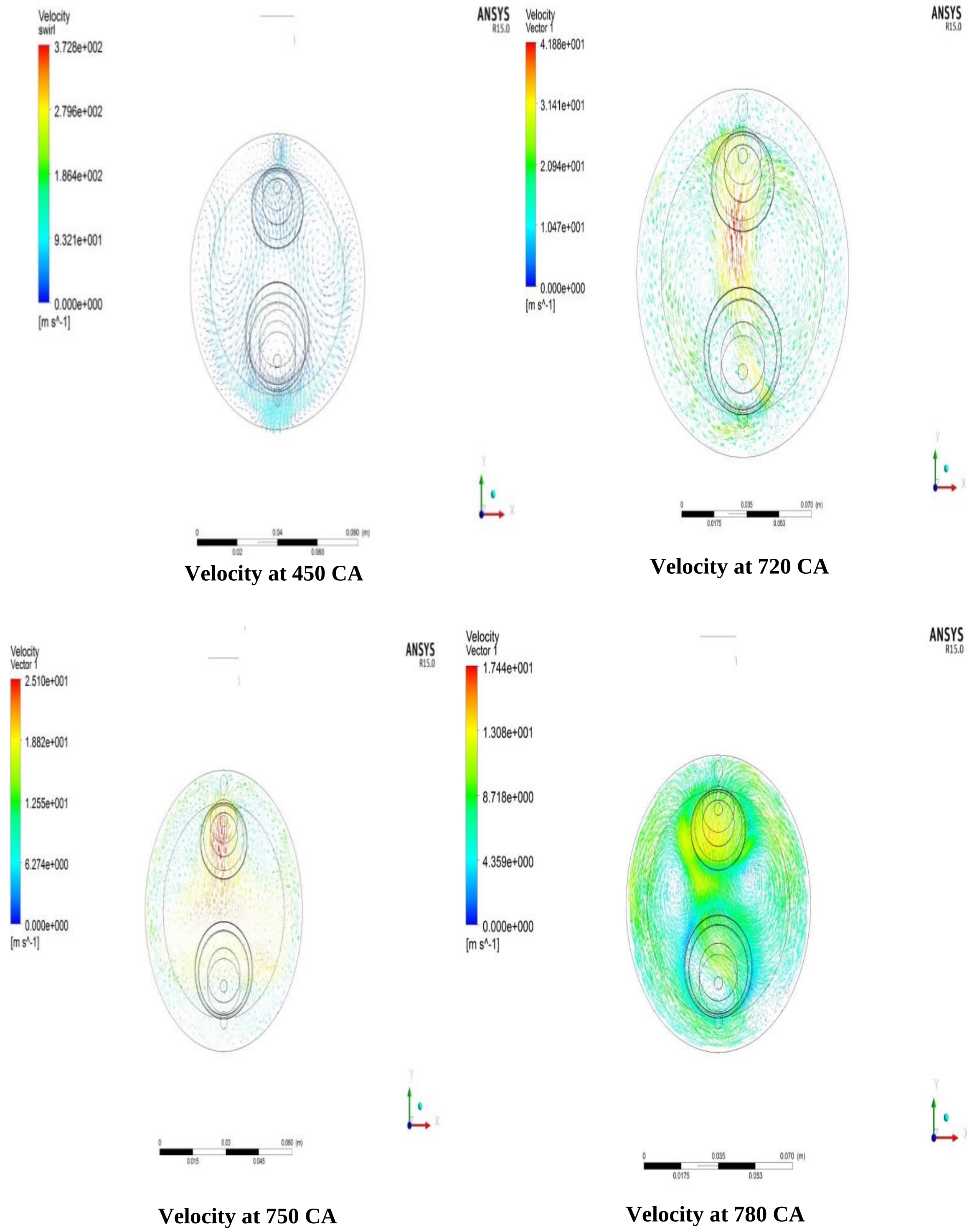


Figure (15): Temperature contours at various crank angles for combustion

The figure (16) and (17) refers to velocity distribution in combustion chamber with combustion simulation the velocity is maximum, approximately 70 m/s at 450 CA. The velocity reduces respectively with combustion propagate (720,750,780 CA).



Velocity at 450 CA

Velocity at 720 CA

Velocity at 750 CA

Velocity at 780 CA

Figure (16): Velocity contours for combustion simulation

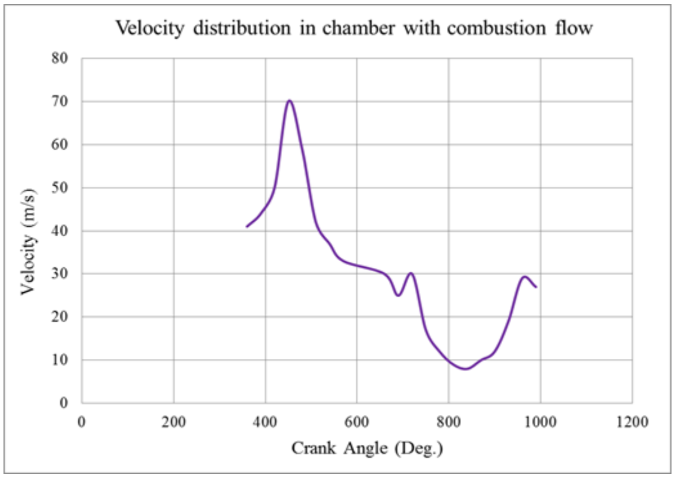
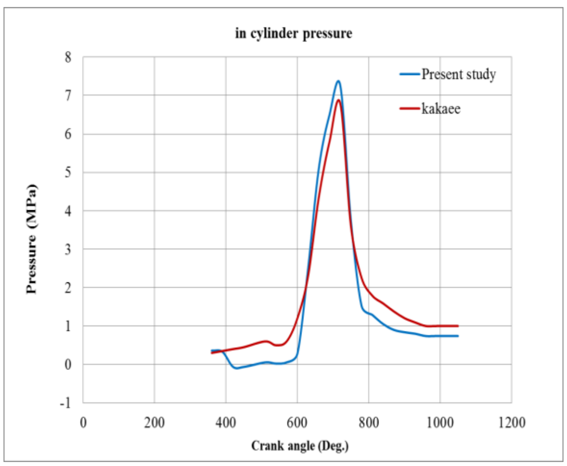
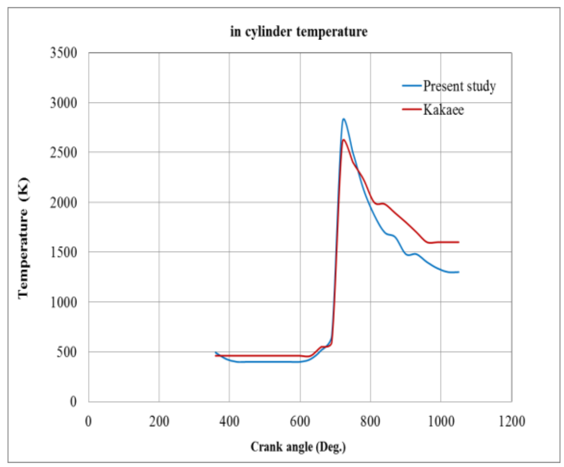


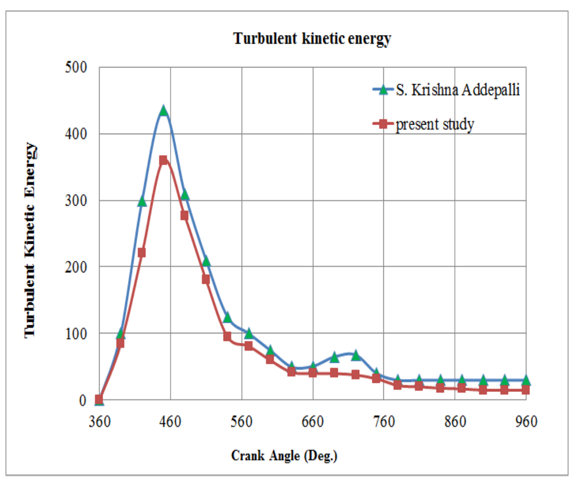
Figure (17): velocity in-cylinder trend for combustion

4. Validation

The validity of this work of CFD analysis has been examined by comparing the results with experimental and the analytic solution of [9], and it is validated with the convergence code [10].







5. Conclusion

The flow characteristics within SI engine were investigated utilizing CFD ICE CODE. As a result, the following conclusions were summarized:

- The adopted approach of the modeling of combustion simulation via ANSYS ICE CODE with dynamic mesh technique can be used to develop of internal combustion engine.
- The two equations standard k-ε model showed acceptable model to simulate the motion of flow within engine.
- Through this work, the in-cylinder flow characteristics were obtained as expected trends.
- The temperature, pressure, and velocity were plotted with respect to the crank angle.
- The value of temperature and pressure for firing simulation were bit higher than researches experimental due the fact this simplified CFD simulation doesn't include friction losses induced interactions of engine components.

CONFLICT OF INTERESTS.

- There are no conflicts of interest.

References

[1] HEYWOOD, J.B., *Internal Combustion Engine Fundamentals*. New York: McGraw- Hill, 1988.

[2] Paulina. S. Kuo. "Cylinder Pressure in a Spark-Ignition Engine a Computational Model ".Engineering Sciences, J. Undergrad. Sci. 3: pp141-145 fall 1996.

[3] Venkata Suman Movva. "Simulation Of Fluid Flow In Internal Combustion Engines Using Wave Action Simulation". M.Sc. thesis.2004.

[4] Ehsan Omaraa "Analysis of Flow Characteristics in Inlet and Exhaust Manifolds of Internal Combustion Engine". M.Sc. Thesis.2009.

[5] T.Morauszki, P.Mandli, Z.Horvath, and M.R.Dreyer. "Simulation of Fluid Flow Combustion and Heat Transfer in Internal Combustion Engine". Hungarian Journal of Industrial Chemistry Vol.39 (1) pp. 27-30-2011.

[6] K. M. Ravichandra, D. Manikanta, M. Kotresh. "CFD Simulation of an Engine by Producer of Gas". International Journal of Civil Engineering and Technology (IJCIET), Volume 8, Issue 10, October 2017.

[7] Abdul Rahiman, Abdul Razak R. K., Mohammad Samee A. D., Ramis M. K. "CFD Analysis of Flow Field Development in a Direct Injection Diesel Engine With Different Manifolds". American Journal of Fluid Dynamics, 4(3): 102-113, 2014.

[8] ANSYS / Work Bench / V15.0. ICE Guide.

[9] Kakaee, Gharloghi, Foroughfar, and Khanlari. "Thermo-mechanical analysis of an SI Engine Piston Using Different Boundary Condition Treatments" J. Cent. South Univ. 22: 3817−3829 DOI: 10.1007/s11771-015-2926-7 Springer, 2015.

[10]S. Krishna Addepalli, Mallikarjuna J.M. "Parametric Analysis of 4-Stroke GDI Using CFD".Alexandria Engineering Journal.2016.

محمود عطا الله مشكور **مصطفى هادي أبراهيم**

قسم الهندسة الميكانيكية، جامعة التكنولوجيا، بغداد– العراق

MahmoodMashkoor@hotmail.com abc_logo@yahoo.com

الخلاصه

ان التشغيل الامثل لمحركات الاحتراق الداخلي يتطلب تطبيقات لتقنيات ضرورية متقدمة بالاضافة الى طرق عملية، ان استخدام التحليل العددي ثلاثي الابعاد يوفر امكانية الحصول على رؤية ثاقبه للظواهر الفيزيائية المعقده داخل المحرك. في هذا البحث، تم نمذجة جريان المائع داخل محرك احتراق بالشرر رباعي الاشواط احادي الاسطوانه نوع هونداي بالأعتماد على التحليل العددي بأستخدام كود ANSYS /ICE، مع تقنية الشبكة الديناميكية لدراسة وتخمين خواص الجريان لوقود الاوكتان عند ظروف التشغيل الطبيعيه وفقاً لزوايا عمود المرفق عند سرعة دوران ثابته. تم أنجاز موديل المحرك بأستخدام بيئة SolidWorks. ركز هذا العمل على تحليل الأشواط الاربعة للمحرك في حالة التشغيل (البارد والاحتراق) متضمناً معادلات الاستمرارية ورينولدز ونافير ستوك ومعادلة حفظ الطاقه. قورن هذا البحث مع الابحاث المنشورة وأظهرت المقارنه مطابقة النتائج،وكان الحد الاقصى للتناقض17 %.

الكلمات الداله:محرك احتراق بالشرر، تحليل عددي ثلاثي الابعاد، خواص الجريان، كودICE، الشبكة الديناميكيه.