Numerical prediction to study the effect of pentroof chamber on the induction swirl intensity

Sadoun Fahad dakhil
Basrah Technical College
ISSN -1817-2695
(Received 22 February 2010; Accepted 24 February 2011)

ABSTRACT

This paper describes a new method for estimating the induction swirl intensity by means of the numerical simulation of gas flow in the intake port and cylinder to optimize combustion chamber design.

The optimization procedure adopted in the present investigation is based on GENERIC ALGORITHIMS and allows different fitness functions to be simultaneously maximized. The parameters to be optimized are related to geometric feature shape of the combustion chamber. The evaluation phase of GENERIC ALGORITHIMS was performed by simultaneity behavior of each design with modified version of FLUENT code 6.2.

The present work represents Direct injection spark ignition engine with different combustion chamber design to get optimum swirl ratio with different engine speed. The study of the chamber angle variation is to show the change in air jet value around inlet valves and then design optimum angle which gets uniform annular jet around each intake valve to give as a result high swirl ratio. Good results are obtained at engine speed 2500 rpm with pentroof angle 15º for both sides. These results give good agreement with experimental result by Ref [1], where he used single inlet valve chamber.

Key words: internal combustion engines, fluid field, numerical analysis, swirl intensity

INTRODUCTION

Two general approaches are used to create swirl during the induction process. In the first approach, the flow is discharged into the cylinder tangentially toward the cylinder wall where it is deflected side way and downward in a swirl motion. In the second approach swirl is largely generated within the inlet port, where the flow is forced to rotate about the valve axis before it enters the cylinder[1]. Recently pentroof chamber is developed to avoid any restriction through the flow motion and reduced the volumetric efficiency[2].

In the indirect injection engines for automotive application the combustion chamber is usually characterized by a re-entrant bowel piston to improve the air fuel mixing as a result of a suitable air swirl motion. To avoid the complex design in piston or chamber pentroof chamber is developed for direct injection spark – spark ignition engines. Although this design uses fuel system with high pressure reached 8 Mpa compared with 1 Mpa for other design to operate with stratified charge[3]. In these chambers, the generation of induction swirl which affects the combustion process is controlled by contriving the configuration of the chamber design, i.e: inclined angle of pentroof chamber wall with a little increase in the compression ratio.

Moreover, the development of CFD codes for engine simulations and the use of high performance computer allowed researchers to overcome the limitations of experimental investigations[4].

In order to estimate exactly the swirl intensity it is necessary to analyze the whole process of flow through the intake valves and the cylinder. However the three dimensional analysis of the whole process with unsteady flow is considered. Fig .(1) illustrates the locations of inlet and exhaust valves as a top view of the pentroof chamber with valves timing along four strokes engine.

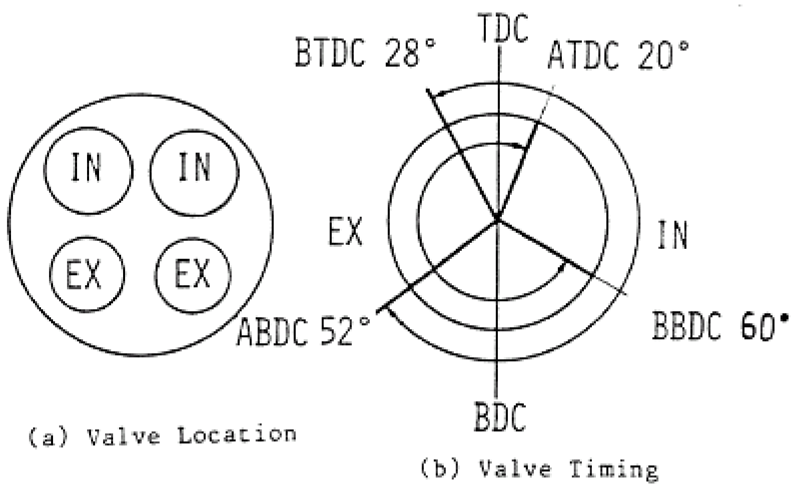

Fig.(1) Top view for the valves location in combustion chamber and timing[2]

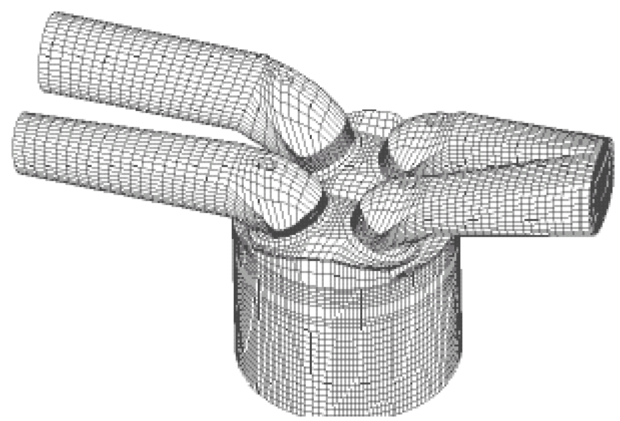

Fig.(2) View of the computational grid for the pentroof chamber and cylinder[6].

Method of numerical analysis

A new calculation method has been developed for computing three dimensional compressible unsteady and turbulent flow in the whole region of the intake valves and cylinder. This method named Generalized Tank and Tube(GTT) method has been created by introducing the technique of general coordinate transformation into finite – volume method by Cartesian coordinates.[3]

Assuming that the gas is compressible and viscous fluid, the conservation equations of its mass, momentum and energy can be written as[5]:

$$\frac{\partial}{\partial t}(\rho\phi)+\frac{\partial}{\partial x}\left(\rho v_x\phi-\Gamma_\phi\frac{\partial\phi}{\partial x}\right)+\frac{\partial}{\partial y}\left(\rho v_y\phi-\Gamma_\phi\frac{\partial\phi}{\partial y}\right)+\frac{\partial}{\partial z}\left(\rho v_z\phi-\Gamma_\phi\frac{\partial\phi}{\partial z}\right)=S_\phi \qquad (1)$$

where: ϕ =dependent variable (velocity components , temperature , both kinetic and dissipation energies). Γ_ϕ =effective exchange variable coefficient of ϕ . S_ϕ = source term with the total pressure gradient. Eddy viscosity must be determined on the basis of an adequate turbulent model.

The transport equations of turbulent energy and dissipation are expressed as the folwing form.

$$\frac{Dk}{Dt}=\frac{\partial}{\partial X_i}\left\{\left\{(\nu\delta_{jk}+c_s\frac{k}{\varepsilon}\bar{u}_k\bar{u}_j)\frac{\partial k}{\partial X_k}\right\}-\bar{u}_k\bar{u}_j\frac{\partial u_i}{\partial X_k}-\varepsilon\right. \quad (2)$$

$$\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial X_i}\left\{\left\{(\nu\delta_{jk}+c_1\frac{k}{\varepsilon}\bar{u}_k\bar{u}_j)\frac{\partial \varepsilon}{\partial X_k}\right\}-\frac{\varepsilon}{k}c_2\bar{u}_k\bar{u}_j\frac{\partial u_i}{\partial X_k}-c_3\varepsilon\right. \quad (3)$$

Model constants: c_s , c_1 , c_2 , c_3 are 0.22 , 0.18 ,1.44 and 1.92 respectively, and turbulent viscosity is[5]:

$$\nu=C_\mu\frac{k^2}{\varepsilon} \quad (4)$$

C_μ =0.09

Fig.(2) illustrates the computational grid in three dimensional coordinates for the pentroof combustion chamber. This domain is divided into a number of control volume of rectangular prism as shown in Fig(3-a). Density ρ , pressure **P** and enthalpy **h** which are scalar variables and the values placed within the central axis of the control volume shown in Fig.(3-b), while each velocity vector component is placed on each surface of the control volume. Regarding the energy conservation equation $(\phi = T)$ in Eq.(1) to calculate enthalpy **h**. As a turbulence $k-\varepsilon$ model is applied and the law of the wall is used as the wall boundary conditions for velocity and temperature .
The hybrid scheme is used for the convection term, pressure velocity coupling is accomplished by the SIMPLE algorithm. Using the fully implicit sheme, all dependent variations are calculated iteratively at each time step. Using the known quantities in a time step, the discritized conservation equations of momentum and velocity are solved to give the velocity components an enthalpy in the next time step. As the velocity field thus obtained does not satisfy the mass conservation equation, the velocity components and pressure must be corrected to satisfy the mass conservation equation using SIMPLE algorithm. The density is calculated with the equation of state. As the quantities obtained through above calculation do not still satisfy the conservation equations of momentum and energy , the calculation needs to iterated. When the space average of the absolute value of relative residual calculation from each discritization equation becomes less than a permissive error, the calculation results are recognized as a converged solution and the calculation is advanced to the following time step.

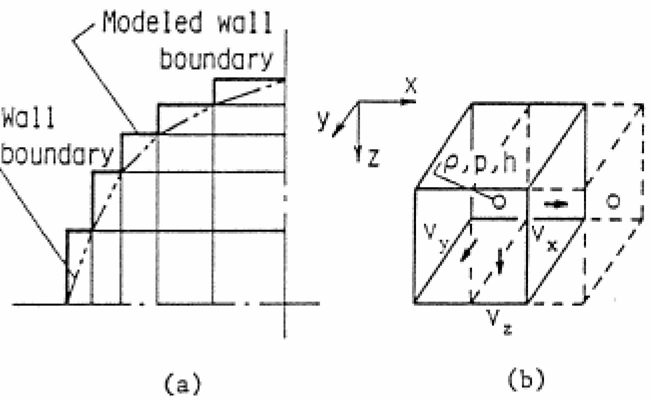

Fig.(3) Generalized Tank and Tube(GTT) method into control volumes[3]

Swirl ratio Estimation

The swirl ratio S_c was calculated as the ratio of the equivalent solid- body swirl speed about the cylinder axis to the engine speed. Pentroof combustion chamber means swirl chamber through both intake and compression processes due to the high effect of the chamber inclined wall[1]. Then, the angular momentum around the cylinder axis, which supplied into the cylinder in a unite time, is expressed as[4]:

$$\Omega_C = \int \rho\, v_y v_{xy}\, R_C\, ds \tag{5}$$

Therefore, the swirl ratio S_c at any crank angle θ defined as.

$$S_C = \int_{\theta=28} \frac{\pi}{180\omega} \Omega_C d\theta \Big/ \frac{\omega}{2} R_C^2 M(\theta) \tag{6}$$

where: θ_{28} =timing of intake valve opening, ω =angular speed of engine, R_c =cylinder radius and $M(\theta)$ = mass of the cylinder contents at crank angle θ

Initial & Boundary conditions

In this study, the initial condition at the timing of the inlet valve opening is set to be stationary state at every calculation step of the crank angle degree.
At the inicial condition both temperature and pressure kept constant along manifold.
The following assumption are made:

• Inlet boundary: It is assumed that the pressure through the inlet section is uniform and the velocity direction is perpendicular to the section and calculated from mass flow equation with fully developed condition . The inflow condition of turbulent energy and dissipation are assumed[5], [6]:

$$k_{in} = \frac{3}{2} u_{in}^2 x\ I \quad , \quad \varepsilon_{in} = C_\mu^{3/2} \frac{k^{3/2}}{I} \quad ,, \ I = 0.07R$$

• Wall boundary: The velocity of gas at the stationary wall equal to zero (no slip condition). The velocity of gas at the moving valve is the same as that of the valve. The shear stress on the wall is proportional to the velocity gradient perpendicular to the wall. The temperature of the wall is constant and k & $\varepsilon = 0$
Since the present model can be classified into the high Reynolds number, near wall function is adopted as the boundary condition for the turbulence model as shown in Fig(3) .

$$k_p = \frac{\tau_p}{\rho C_\mu^{1/2}} \quad , \quad \varepsilon = \frac{C_\mu^{3/4} k_p^{3/2}}{0.4 (y_p)}$$

Piston boundary condition (outlet boundary): The axial distribution velocity near the piston face was normalized set to the piston speed[8] :

$$V_p = \frac{2\pi N}{60} a. \left(\sin\theta + \frac{\sin 2\theta}{2\pi} \right)$$

The influence of chamber geometry on the air jet

Preliminary calculations were carried out for the simple intake model which has an intake area of $1/16$ sector of the inclined wall of the pentroof chamber without intake ports. As shown in Fig.(4).The intake jet enters into the cylinder directed at 45° downward from the intake port. The tangent velocity component is the same at any radius location of the intake sector and the radial component is zero[5]. Thus, the intake jet velocities are given by the use of piston speed with given sector angle and annular orifice on the cylinder head surface as shown in Fig.(5) [4]. The average velocity V_m around the valve periphery is calculated by dividing the piston induced volume flow rate by the valve effective area which assumed to 70% of geometrical valve flow area. As illustrated in Fig.(5) the velocity at each segment around the valve periphery is designated by V_i and it is represented by the following relation[4].

$$V_i = C_d V_m \tag{7}$$

where: C_d is discharge coefficient can be estimated by straightforward equation as:
C_d = prescribed mass flow rate / ideal mass flow rate for` the prescribed pressure drop

$$\dot{m}_i = \rho C_d \cos \Delta\gamma_i . \Delta S_{vi} V_m \quad (8)$$

$$\Delta S_{vi} = F_v \Delta\psi_i / 2\pi \quad (9)$$

where: F_v is the valve effective flow area. $\Delta\psi_i$ is the sector angle of each of the segments, i.e: 10,12.5,15,17.5 degree are selected in the present work. γ_i is the direction of V_i as shown in Fig.(5) as a result of pentroof angle. The mass flow rate $\dot{m}_i$ is used for the inlet boundary condition at each boundary grids.

By applying Eq.(4) the mass flow rate $\dot{m}_i$ through each of the segments is given by [4] :

The radial, tangential and axial components of V_i are designated by u_{vi} , v_{vi} w_{vi} respectively.

The effect of inclined wall of petroof chamber was considered .Values of swirl ratio versus crank angle for deferent cases considered were calculated and plotted in Fig.(7).

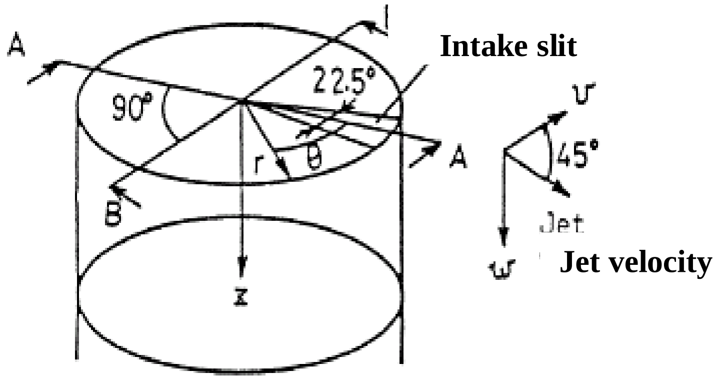

Fig.(4) Intake arrangement for preliminary calculation

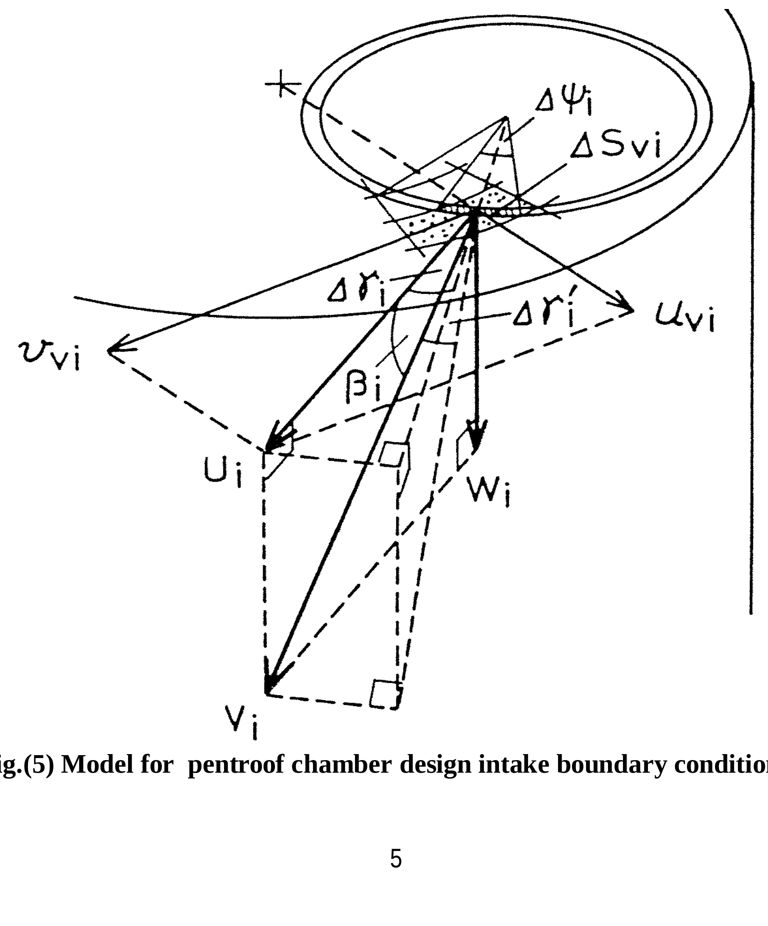

Fig.(5) Model for pentroof chamber design intake boundary condition[2]

Results & Discussions

Swirl ratio fields had been computed numerically using the GENERIC ALGORITHIMS method. Air jet through the inlet valves and inside cylinder during the suction stroke are based on air flow through the intake manifold as initial and boundary conditions. The results were optained at engine speed 2000 & 2500 rpm along one cycle (360°)

Fig.(6) shows maximum jet velocity at engine speed 2500 rpm with different angles $\Delta\psi_i$ to the petroof chamber. High velocity appears at suction process near crank angle at 90 °ATDC with 15 degree pentroof chamber due to maximum valve lift, maximum piston speed and the resultant between two jets from the intake valves. The components of jet at the cylinder center line are strongly dependet on the pentroof chamber angle $\Delta\psi_i$ which controlled the component velocity U_i by the angle γ_i as shown the analysis in Fig.(5).These results show that the resultant velocity is affected by the components direction to give main jet through the cylinder center line. So this is a good parameter to optimize pentroof combustion chamber. At both angles 10 & 17.5 the component velocity U_i is decreased due to the increase of the impact between two jet particles that decrease the discharge coefficient and weakness of center jet takes place. At the similar conditions Fig.(7) shows the results at engine speed 2000 rpm. Air velocity is normalized to mean piston speed, so the maximum velocity decreases at slow engine speed. These results indicate high jet at combustion chamber with $\Delta\psi_i$ equal to 15 degree due to the same above reason.

Figs.(8&9) represent the air swirl ratio through one cycle of engine with different combustion chamber angle at 2500 & 2000 rpm respectively. The results indicate high swirl during the first half of the intake process due to high velocity generated during this portion. At the second half after 180°ATDC swirl decays due to the rest into piston speed. Typically one third of the initial moment of momentum about the cylinder axis will be lost by compression process due to the effects of friction. Also Figs.(8&9) illustrate the substantial increase at the end of compression process by the effect of the optimum design of petroof chamber. For the chamber with angle 15º there is an enhancement of the swirl motion at the end of compression process which helps engine to operate with stratified mixture. These results are compared with the experimental results obtained by Ref [1], who designed mixer inside the inlet manifold to improve swirl flow. There is good agreement with the experimental results by Fahad[1], the scattering of the experimental results at the beginning of the cycle are related to the difficulties in measurement near the seat valves[1].

Conclusions

The following major conclusions are derived from the present study:

1-Secondory motions through the swirl flow region are largely with high degree pentroof chamber angle responsible for enhanced air fuel mixing,

2- Swirl intensity occurs at the beginning of intake stroke near 10 degree ATDC , that the important effects of the new chamber without any modification of the inlet geometry which decreased the volumetric efficiency .

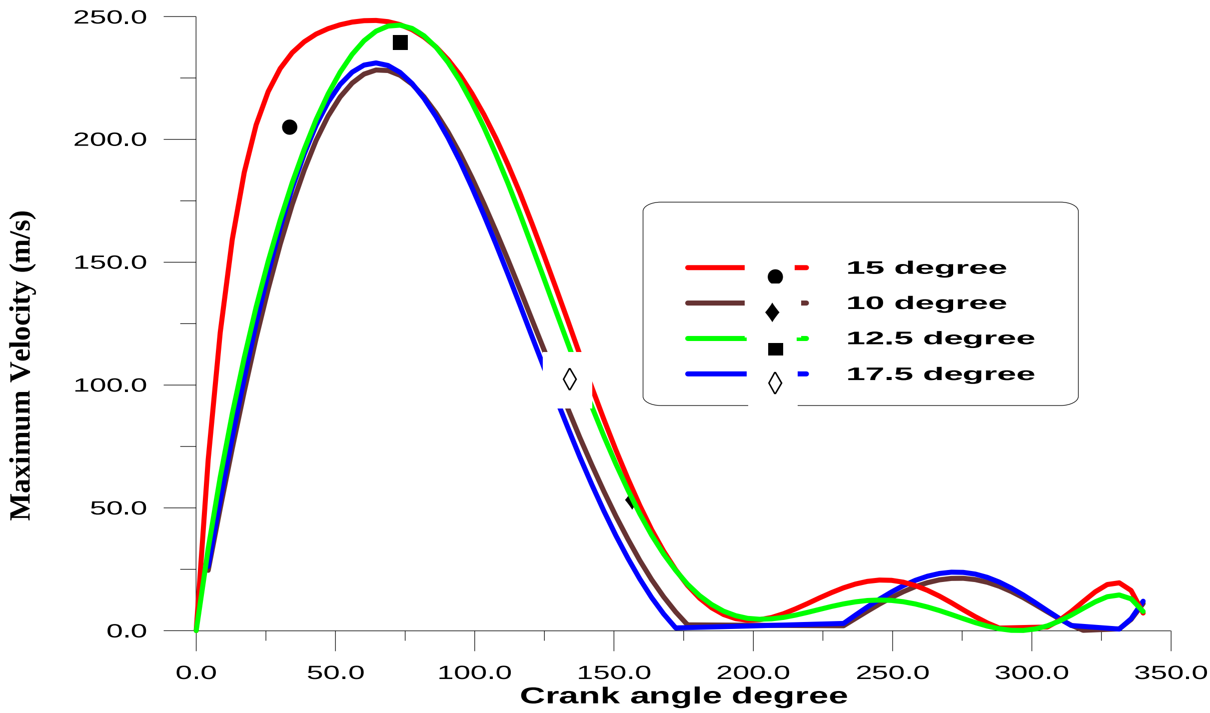

Fig.(6) Maximum intake air velocity with different pentroof chamber angle. Engine speed at 2500 rpm.

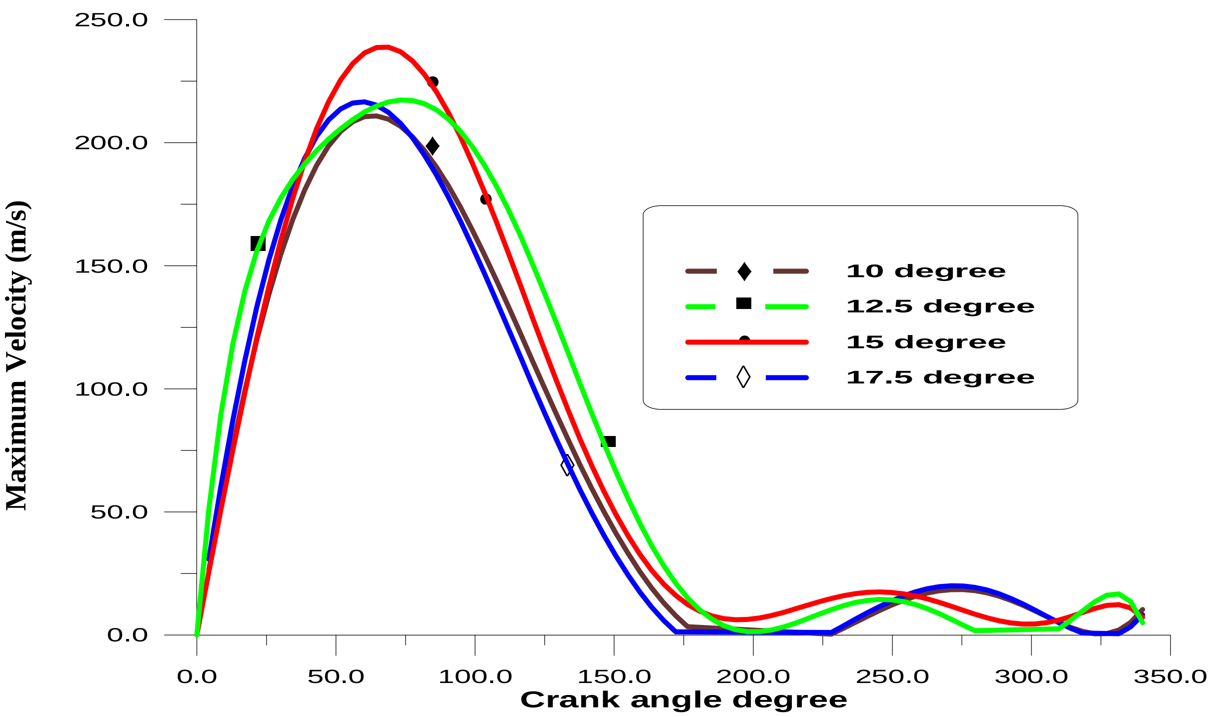

Fig.(7) Maximum intake air velocity with different pentroof chamber angle Engine speed at 2000 rpm.

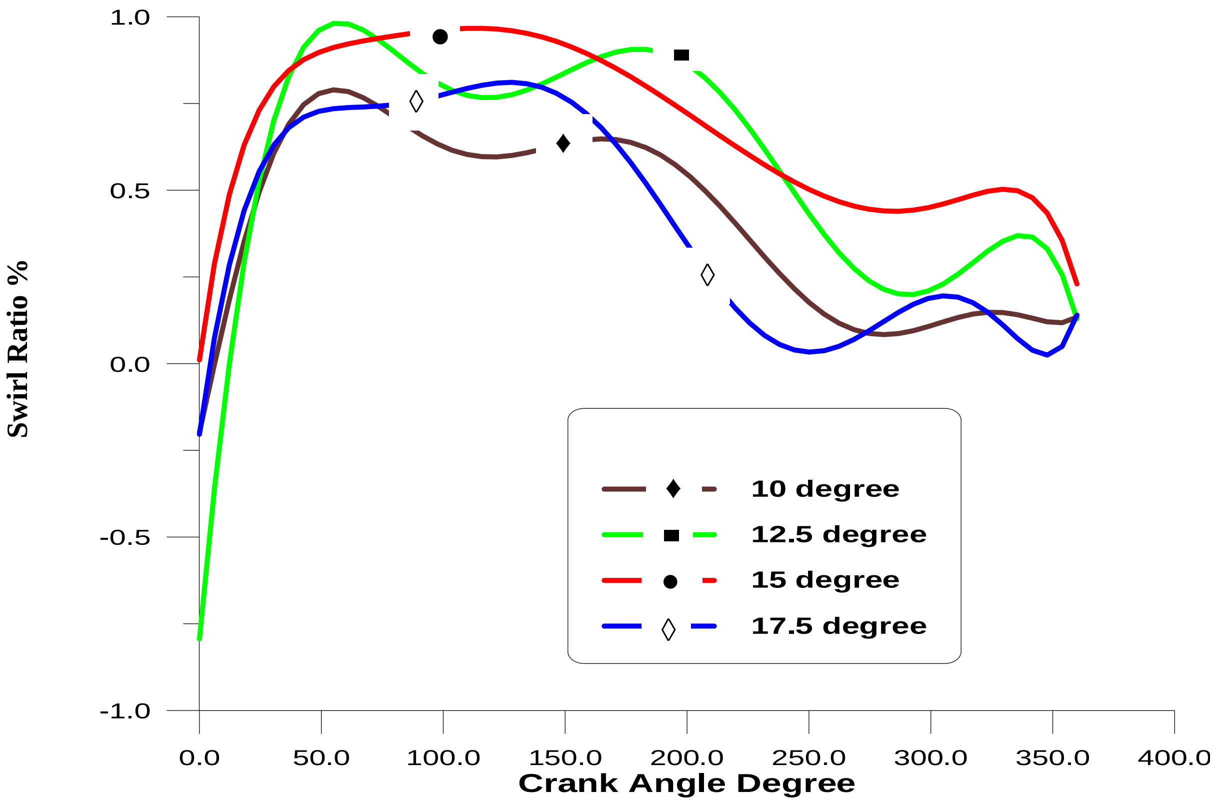

Fig.(8) Air Swirl ratio along engine processes at 2500 rpm with different pentroof chamber angle .

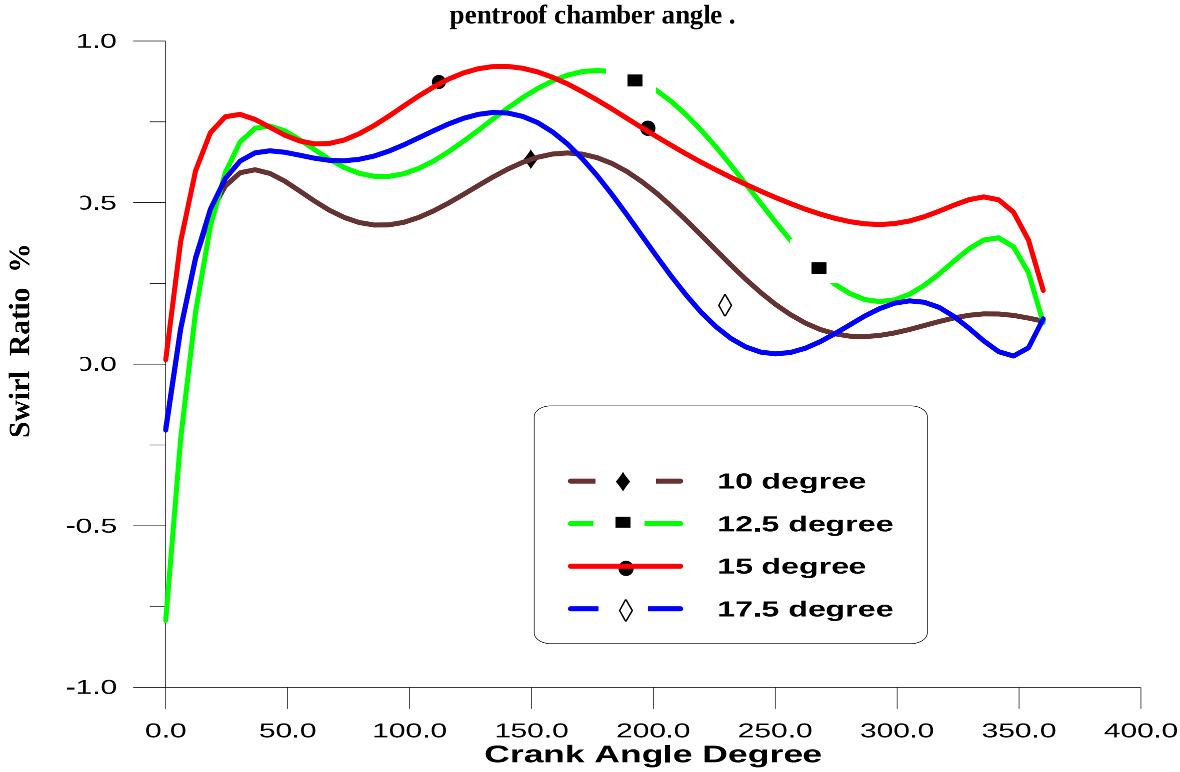

Fig.(9) Air Swirl ratio along engine processes at 2000 rpm with different pentroof chamber angle.

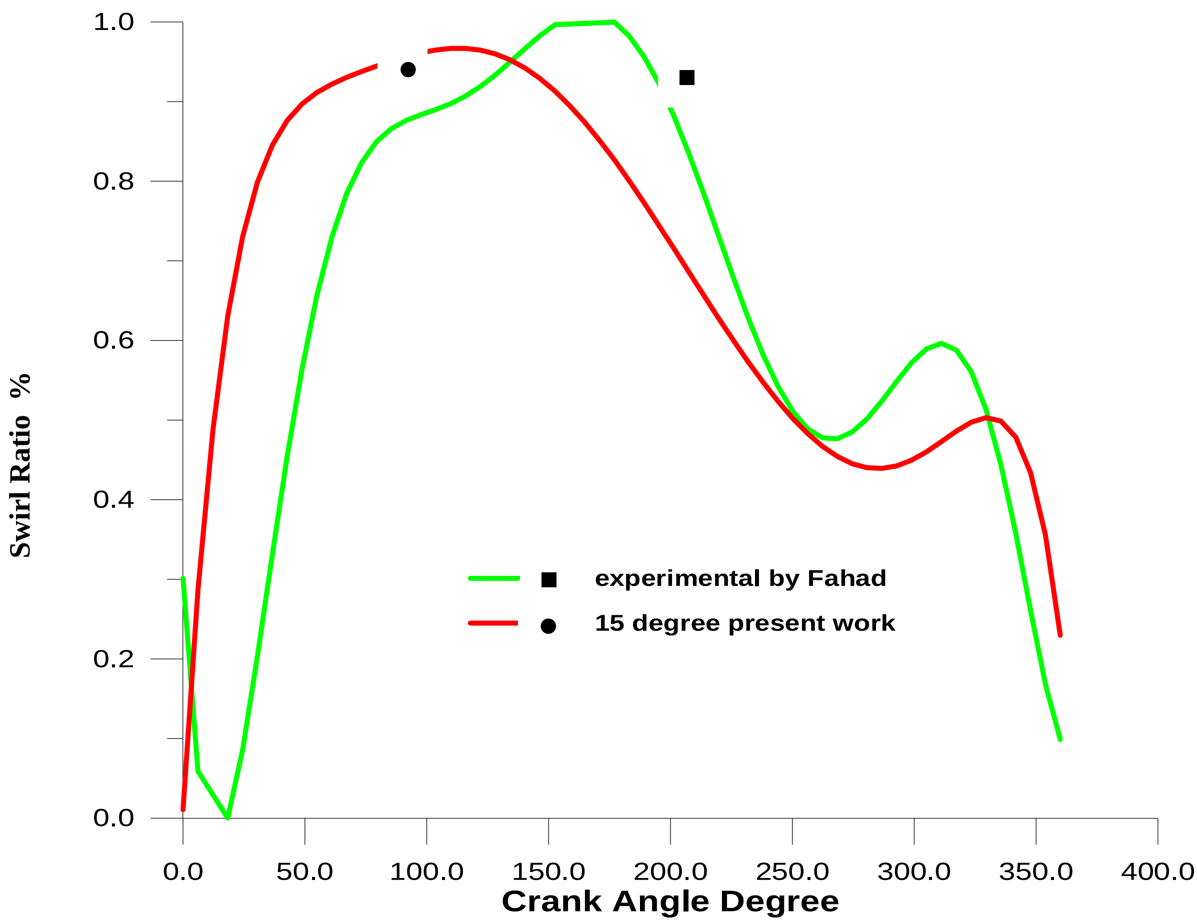

Fig.(10). Comparison of the swirl ratio along engine cycle at 2500 rpm

List of symbols

Symbol	Definition	unit	subscript	Definition	unit
k	turbulent kinetic energy	m^2/s^2	v	valve	--
ε	turbulent energy dissipation	m^2/s^3	m	mean	-
ψ_i	pentroof angle	degree	in	inlet	-
	exhaust	-	rpm	revelation per min	1/min
ex	jet at x axis	-	p	piston,point	
γ_i			iv	inlet valve	
	jet at y axis	-			
γ'_i	jet x velocity angle	degree			
β	resultant velocity	m/s			
	after top dead center				
ν_i	effective viscosity	kg/m.s			
	cylinder radius	m			
ATDC	engine speed	rpm			
Γ	turbulent kinetic energy	m^2/s^2			
R_c	at point near wall				
N					
k_p					

References

[1] **S. Fahad** ″ Swirl Effect on the Brake Specific Fuel Consumption in SI Engines ″ Al-Tiqani Journal, Foundation of Tech. Education,Vol.1, 1986 Iraq .

[2] **T. Wakisaka , Y.Shimamoto & Y. Isshiki** ″Analysis of the Effects of in Cylinder Flows During Intake Stroke in Four –Stroke Cycle Engines″ International Symposium, COMODIA, PP. 487-492,1990, Japan.

[3] **O.Bailly, C.Buchou , A. Floch & L. Sainsaulieu** ″ Simulation of the Intake and Compression Strokes of a Motored 4-valve SI Engine with a Finite Element Code ″ Oil & Gas Science and Technology Ð Rev. IFP, Vol. 54, No. 2, pp. 161-168, ,(1999) France.

[4] **Y.Isshiki, T.Wakisaka & Y.Shimamoto** ″ Numerical Prediction of Effect of Intake Port Configurations on the Induction Swirl Intensity by 3D Gas Flow Analysis″ 4th Int Symposium on I C engines , PP.295-304,1984,1999.USA

[5] **H.Min , W. Kim , I .Chung & I.Chyun** ″ Three Dimensional Flow Characteristics and Engine Performance for the Geometry Modification of Intake Manifold in Multi Cylinder Engine ″ Seoul FISITA World Automotive Congress , Paper Series F2000A007, PP. 12-15, June 2000,Korea.

[6] **S.Fahad ,** ″ An aerodynamic analysis of gasoline direct injection engine processes operating under stratified charge″ PhD thesis , BASRAH UNIVERSITY ,2007.Iraq

[7]**H.Sugiyama**," Numerical analysis of developed turbulent flow in a bend tube "Int,J.Numer.Meth,Fluids, Vol 47, pp1431-1449, 2005.

[8]**Hywood ,J.B**" Fundamentals of internal combustion engine "McGraw,Hill,1988,ISBN 0-07,100499-8.USA.

تحليل نظري لتأثير تصميم غرف الاحتراق على شدة دوامات السحب

الخلاصة

استخدم في الدراسه الحاليه طريقه حديثة لحساب شدة الاضطراب للهواء الداخل للمحرك لغرض الحصول على امثل تصميم لغرف احتراق المحرك. حيث تعتمد خطوات التحليل على نظرية

والتي تعطي وصفا دقيقا لسلوك الجريان عند مجاري السحب بالتالي نمذجه الشكل (GENERIC ALGORITHMS)
بالبرنامج الجاهز

أجريت الدراسة على محرك يعمل بالشرارة وحقن مباشر واستخدام غرف احتراق بزوايا وسرع FLUENT code 6.3 .
للمحرك مختلفة للحصول على امثل غرفه احتراق تظمن شدة اضطراب عليه دون إعاقة الجريان. دوران

توضح النتائج الحصول على تغير كبير في قيمه منفث الهواء حول صمامات السحب عند تغير زاوية ميل غرفة الاحتراق، والتي تعطي شدة اضطراب عاليه. تم الحصول على أفضل نتائج عند سرعة دوران 2500 د/دقيقه وغرفة بزاوية 15 درجه لكـلا الجانبين .كما تم المقارنة مع نتائج عمليه للباحث (1) هنالك اختلاف قليل في النتائج لكونه استخدم غرفه ذات صمام واحد.