Numerical Prediction of the Gas Flows in a Reciprocating Engine Cylinder During Intake Stroke

(1st report: inviscid flow)

محاكاة عددية لسريان الغاز داخل اسطوانة آلة ترددية خلال شوط السحب
(سريان غير لزج)

M.S. EL kady, E.B. Zedan and M.M. Awad

Mechanical Engineering Departement-Faculty of Engineering
Mansoura University

الخلاصة - يقدم هذا البحث نموذجا عدديا لاستنباط حركة الهواء داخل اسطوانة محرك ترددي وذلك خلال شوط السحب . ولقد استخدم نموذج مبسط مكون من اسطوانة ذات غرفة احتراق مركزية في المكبس وذات صمام سحب مركزي مع محورها ، وتم دراسة السريان الغير لزج الغير مستقر ثنائي الأبعاد المتماثل حول المحور . وقد صيغت معادلات الحركة من معادلات الاستمرار وكمية الحركة والتي تحولت داخل النموذج العددي باستخدام طريقة الفروق المحددة واستخدمت فيها شبكة متحركة تتمدد في الاتجاه المحوري ثم طريقة الاتجاه المتغير الغير صريح وذلك لعملية تصحيح الضغط بالتكرار . وقد تم التركيز على حركة المائع الخارج من صمام السحب أثناء شوط السحب ، والتي كانت عالية جدا مما كان له أثرا في تصغير الخطوة الزمنية أثناء الحل العددي الى نصف درجة بدلا من خمس درجات في شوط الانضغاط وذلك لتتميم التقارب والاستقرار للحل . وقد أظهرت النتائج بعضا من خصائص السريان داخل الاسطوانة خلال شوط السحب وتم التمهيد بهذا لدراسة حركة المائع اللزج في الخطوات التالية .

ABSTRACT

This paper presents a method for predicting the in-cylinder gas flows during the intake stroke of four-stroke engines. A simplified engine model with a deep-bowl piston confguration, and a central intake valve was used. An inviscid two-dimensional axisymmetric flow was investigated. The model used a finite difference procedure with an expanding/contracting grid in axisymmetric representation. The major source of troubles for the convergence of the solution was the unsteady turbulent jet flow issuing from the intake valve which required very small time intervals for the solution compared with that during the compression stroke where there is no jet flow exists. Results show a complicated flow field during the intake stroke in engine like-geometries.

1. INTRODUCTION

The combustion rate, fuel-air mixing, engine performance and formation of pollutants of an engine are all highly dependent on the flow field that exists in the engine cylinder [1]. Most attention has been given to simulate the compression stroke leading combustion process [2,3] It is clear that the flow during the compression stroke is strongly influenced by the initial conditions at the beginning of compression stroke, i.e., the end of intake stroke. Since early experiments, it has been recognized that the major source of turblence and swirl in the engine cylinder in the intake stroke is the shear flow past the intake valves [4-8]. So, it is useful to have valuable insight into the structure of the flow field during the intake stroke.

A three-dimensional model is required to calculate such a complex flow. However, two-dimensional studies are still useful in allowing the testing of new ideas easily and economically [9]. As a first approximation, solution is obtained in a two-dimensional model of a cylinder with an axisymmetric inlet valve. The engine speed of rotation is so high during the intake process that the flow is modeled via a large inviscid core plus a small viscous boundary layer at the walls as conventionally done in aerodynamics. An inviscid solution might also be the practical vehicle for the initial studies of the flow field during the intake stroke [11]. It allows the concentration upon the problem of unsteady turbulent jet flow issuing from the intake valve and the convergence of its numerical solution.

In the present study, the turbulent flow generated by the movement of the piston from the TDC to the BDC during the intake stroke in a deep-bowl axisymmetric combustion chamber with the flow from a centrally located valve is predicted. The prediction was carried out using a computer program based on a finite-difference algorithm.

2. MATHEMATICAL FORMULATION

2.1 GOVERNING EQUATIONS

Use is made of a cylindrical coordinate system (r,θ,z) whose-axis is set on the cylinder axis and its origin is at the centre of the cylinder head as shown in Fig.1. The components of the velocity $\vec{U}$ are u, v, w in the radial, tangential, and axial directions respectively.

Since the flow pattern can be regarded as axisymmetrical, all derivatives with respect to the tangential direction θ are neglected in order to simplify the mathematical description and to reduce computations required. The governing equations are the continuity and the momentum equations. For a two dimensional inviscid unsteady axisymmetric flow, they take the following form:

$$\frac{1}{r}\frac{\partial}{\partial r}(r\rho u) + \frac{\partial}{\partial z}(\rho w) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho u^2) + \frac{\partial}{\partial z}(\rho u w) - \frac{\rho v^2}{r} + \rho\frac{\partial P}{\partial r} = 0 \tag{2}$$

$$\frac{\partial}{\partial t}(\rho v) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho u v) + \frac{\partial}{\partial z}(\rho v w) + \frac{\rho u v}{r} = 0 \tag{3}$$

$$\frac{\partial}{\partial t}(\rho w) + \frac{1}{r}\frac{\partial}{\partial r}(r\rho u w) + \frac{\partial}{\partial z}(\rho w^2) + \rho\frac{\partial P}{\partial z} = 0 \tag{4}$$

where t denotes the time, ρ the fluid density, and P the pressure devided by the density.

Because the piston velocity is small compared with sonic speed, the density is assumed to be constant during the intake period. It is also assumed that the working fluid is an ideal air, chemical reaction and heat transfer are not considered.

During the motion of the piston, the boundaries of the integration area will vary with time. To make these boundaries time independent, the movable coordinate system discussed in [3] is used, in which the time derivative of the general dependent variable ϕ will be calculated from the relation:

$$\left(\frac{\partial \phi}{\partial t}\right)_{fix} = \left(\frac{\partial \phi}{\partial t}\right)_{mov} - w_g \frac{\partial \phi}{\partial z} \tag{5}$$

$$\text{where} \quad w_g = \begin{cases} \dfrac{z}{h} w_p & \text{in the cylinder space.} \\ w_p & \text{in the bowl.} \end{cases} \tag{6}$$

ϕ, w_p and h stand for general dependent variable, the piston speed and the distance between the piston face and the cylinder head, respectively.

2.2 BOUNDARY AND INITIAL CONDITIONS

The normal velocities at all solid boundaries are set equal to the velocity of boundary, i.e., zero for all solid boundaries except at the piston, where the axial velocity is equal to the piston velocity. Slip conditions are assumed for the other velocity components. At the axis of symmetry, the radial and tangential velocities are equal to zero, and $\partial w/\partial r=0$ and $\partial P/\partial r=0$.

The inlet flow was assumed to enter the cylinder parallel to the direction of the valve-seat annulus inclination, and, the mass flow rate is assumed equal to the rate of change of volume displaced by the piston multiplied by the ambient density. Computations were initiated as the exhaust valve closed, radial and axial velocities were set to be zero, and a solid body of rotation is assumed with a swirl number of the order of the piston rotational speed.

3. CALCULATION ALGORITHM

The finite-difference method is used for this problem. The governing equations, shown above, are transferred to the finite form using the same algorithm presented in [3]. Two different staggered grids are used for the vector and scalar variables [12,13]. The staggered grid system was discussed in detail by Stephens et al. [14], and successfully used by El Kady et al. [3]. The grid layout used, is illustrated in Fig.2.

A variable axial spacing was used to allow for the change in the distance between the cylinder head and the piston top. The grid system consists of 16 radial nodes, 7 radial nodes for the piston deep-bowl, and variable axial nodes ranging from 5 to 17 in the cylinder space and from 14 to 7 in the piston deep-bowl.

The first step for obtaining the flow field after a time increment Δt is to guess the velocity components u, v and w by using the pressure field in the present time t. As these velocity components do not satisfy the continuity equation, which is an elliptic Poission's equation, velocity components and pressure must be corrected as to satisfy the continuity equation. An iterative procedure is applied until the continuity equation is satisfied for each grid node. The absolute values of residuals, which are calculated from each discretization equation, are used in order to confirm the covergency of solution. Computation has been performed relying on Alternating Direction Implicit method ADI using the TriDiagonal Matrix Algorithm TDMA [15].

As a fully converged solution is reached, the calculation is advanced to the following time step.

4.RESULTS AND DISCUSSION

Calculations were made using an axisymmetric engine model. A small size direct injection diesel engine of 120mm bore/102.8mm stroke with a centrally located bowl of 37.5 mm depth, and a diameter of 48 mm and an axisymmetric intake valve of 45 degree -seat angle were used. The engine speed considered is 2400 rpm. A related problem developed in the convergence of the numerical solution to the air jet flowing from the intake valve. It required a very small time step for the solution compared to its value during the compression stroke where there is no jet flow exists [2,3].

The total time interval which corresponds to the complete stroke travel is divided into a number of time steps. Several time step calculations were performed. A typical time steps of 0.5 and 1.0 degree were used, ten solution sweeps of variables per time step were sufficient enough to ensure full convergence. The resultant velocity in the r-z plane, the pressure difference and the tangential velocity are shown in Figs.(3-6) by the way of contours for different crank angles during the intake stroke.

The overall structure of the flow is characterized by a jet flow with a large vortex residing in the vicinity of the centerline This is due to the high speed flow issuing from the narrow valve clearance.

A complicated flow is generated in the cylinder and the vortices varies with time and space. The flow complexity is much higher near mid-stroke, peak velocities also exist. The velocities near corner are low showing a dead region with width varying with time. The tangential velocity changes in both radial and axial directions, hence the assumption of a solid body of rotation at the beginning of compression stroke seams to be a crude approximation.

The calculated results of pressure and velocity distributions at BDC can be now used as initial values for compression flow simulation.

5. CONCLUSIONS

The in-cylinder flows in the intake stroke, which could influence the flow during compression, combustion and then the performance of reciprocating engines were estimated numerically The flow was calculated from an inviscid axisymmetric two-dimensional model. A centrally located intake valve is assumed with a flow parallel to the direction of the valve-seat annulus.

Flow in the cylinder and the vortices varies with time and space. At the end of this stroke the axial and tangential velocities were varying in both radial and axial directions, which make it crude assumption to take a solid body of rotation at the beginning of compression stroke. It can be safely said that the numerical simulation method with this treatment of the jet flow from the valve can be extended and used to simulate the viscous unsteady flow as a better approximation for the flow by choosing a suitable turbulent model.

6. NOMENCLATURE

b_t	Bowl depth.
d_i	Grid spacing in r-direction.
d_k	Grid spacing in z-direction.
f	Scalar field.
h	The instantaneous piston displacement.
P	Pressure/density.
r, θ, z	Cylindrical coordinates.
r_b	Bowl radius.
r_c	Cylinder radius.
t	Time
$\vec{U}$	The velocity vector.
u	Radial velocity.
v	Tangential velocity.
w	Axial velocity.
w_p	The instantaneous piston speed.
β	Valve-seat inclination angle.
φ	Crank angle.
ϕ	General dependent variable.
ρ	Density.

7. REFERENCES

1. Heywood, J.B., "Fluid Motion Within the Cylinder of I.C.E.," ASME Jonrnal of Fluids Engineering, Vol. 109, No. 1, 1987, pp. 3-35.

2. Bernard, P.S. "Computation of the Turbulent Flow in an Internal Combustion Engine During Compression, "ASME Journal of Fluids Engineering, Vol. 103, No. 1, 1981, pp. 75-81.

3. El Kady, M.S., Mascheck, H.J. and Hoche, A., "Numerical Simulation of the Flow in a Symmetrical Deep-Bowl Combustion Chamber of Diesel Engine Cylinder During the Compression Stroke," MEJ, Vol. 12, No. 2, 1987, pp. M20-M30.

4. Tabaczynski, R.J., Hoult, D.B., and Keck, J.C., "High Reynolds Number Flow in Moving Corner," J.Fluid Mech., Vol. 42, 1970, pp. 249-255

5. Komatsu, G., Shirao, S. and Hsokawa, T., "Flow Visualization of Process of Producing Induction Swirl using a Water Model," Bulletin of JSME, Vol. 29, No. 245.1986.pp. 2592-2597.

6. Morse, A., Whitelaw, J.H. and Yianneskis, M., "The Influence of Swirl on the Flow Characteristics of a Reciprocating Piston-Cylinder Assembly," ASME Journal of Fluids Engineering, Vol 102, No.2 Dec. 1980, pp. 478-480.

7. Arcoumanis, C., Bicen, A.F. and Whitelaw, J.H. "Squish and Swirl-Squish Interaction in Motored Model Engines," ASME Journal of Fluids Engineering, Vol. 105, No.1, 1983, pp. 105-112.

8. Bicen, A.F., Vafidis, C. and Whitelaw, J.H. "Steeady and Unsteady Airflow Through the Intake Valve of a Reciprocating Engine," ASME Journal of Fluids Engineering, Vol. 107, No. 3, 1985, pp.413.

9. Gosman, A.D., Johns, R.J.R., and Watkins, P.A. "Development of Prediction Methods for In-cylinder Processes in Reciprocating engines," Combustion Modeling in Reciprocating Engines, Mattavi, J. N., and Amann, C. A. (Eds.), Plenum Press, New York, 1980, pp. 69-129.

10. Lyford-Pike, E.J. andHeywood, J.B. "Thermal boundary layer thickness in the cylinder of a Spark Ignition Engine," Int. J. Heat Mass Transfer, Vol. 27, No. 10,1984, pp.1873-1878.

11. Diwakar, R., Anderson, J.D. and Grifin, M.D. "Inviscid solutions of the flow field in an I.C.E.," AIAA Journal Vol.14 ,1976, pp.1667-1668.

12. Roache, R.J., Computational fluid dynamics, Hermosa Publishers, Albuquerque, 1972.

13. Patankar, S.V., Numerical heat transfer and fluid flow, Hemisphere, Washington, D.C., 1980.

14. Stephens, A.B., Bell, J.B., Solomon, J.M. and Hackerman, L.B. "A Finite Difference Galerkin Formulation for the Incompressible Navier-Stokes Equations," J. Comput. Phys., Vol. 53, 1984, pp. 152-172.

15. Peaceman, D.W., Rachford, H.H. "The Numerical Solution of Parabolic and Elliptic Differential Equations," J. Soc. Indust. Appl. Math. 3, 1955, pp. 28-41.

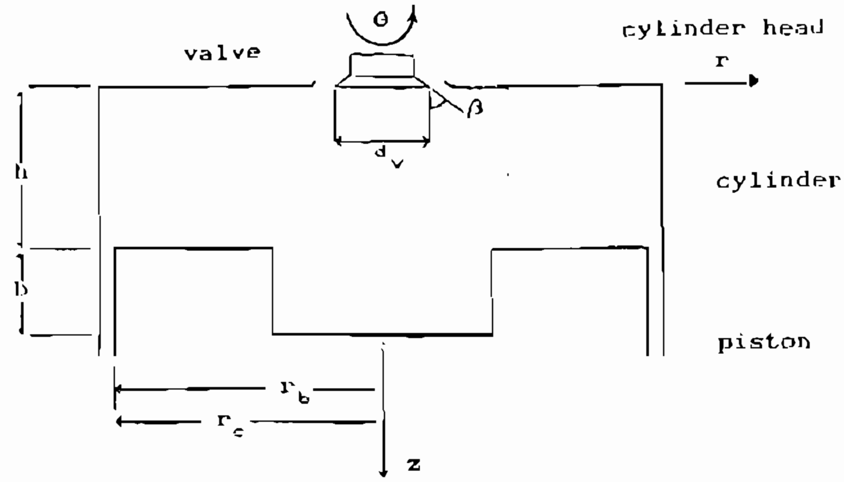
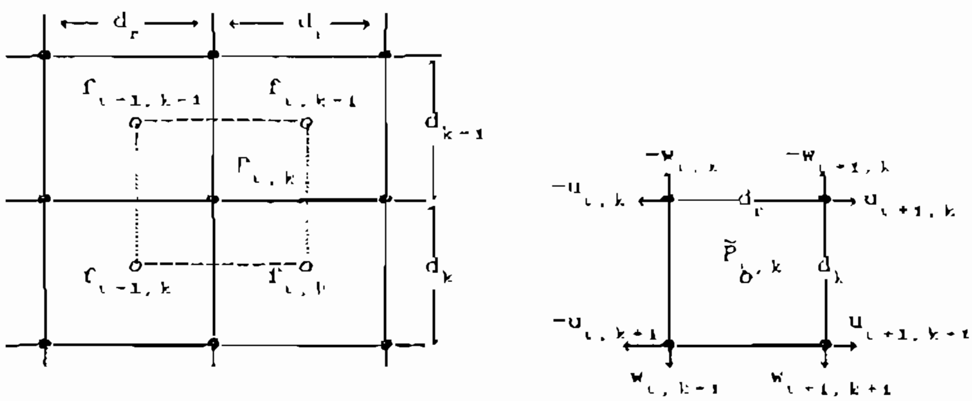


Fig.1. Central intake valve



Fg.2 Staggered grid system

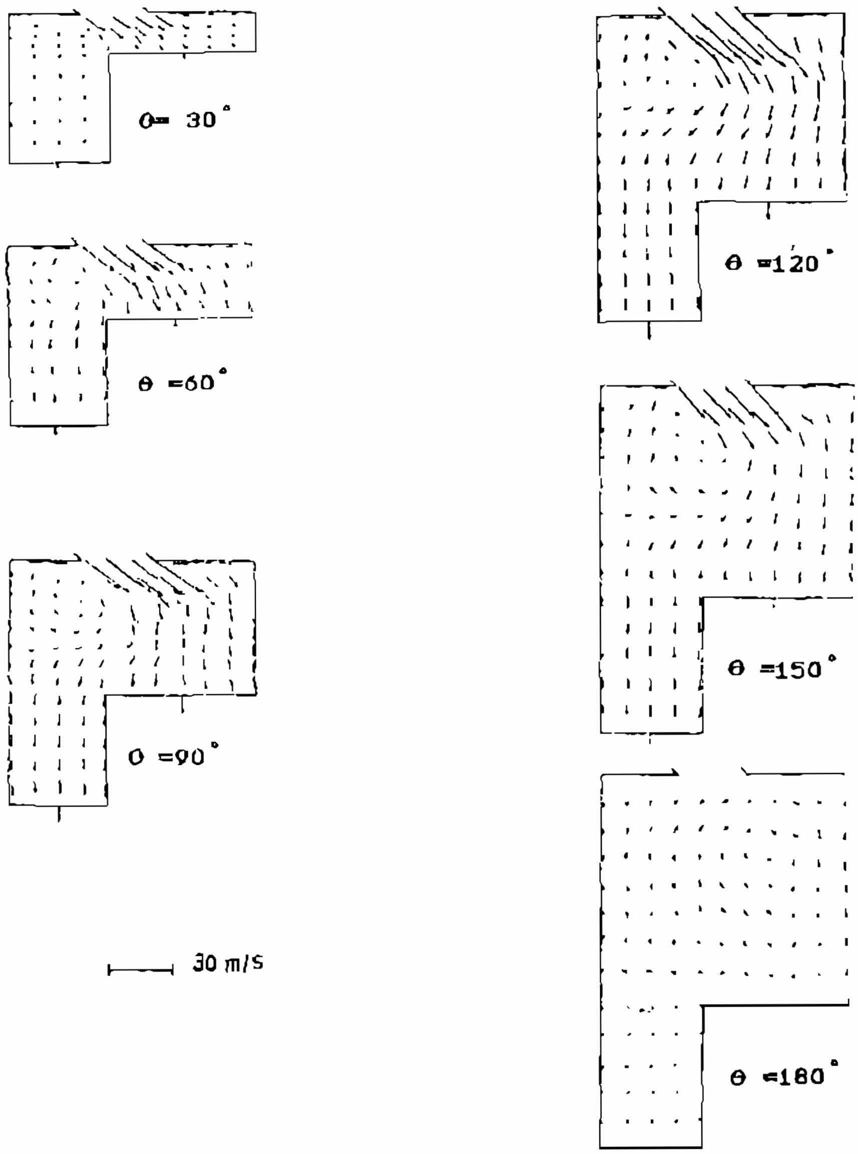


Fig. 3 The vector lines of the velocity U_{rz} during intake stroke.

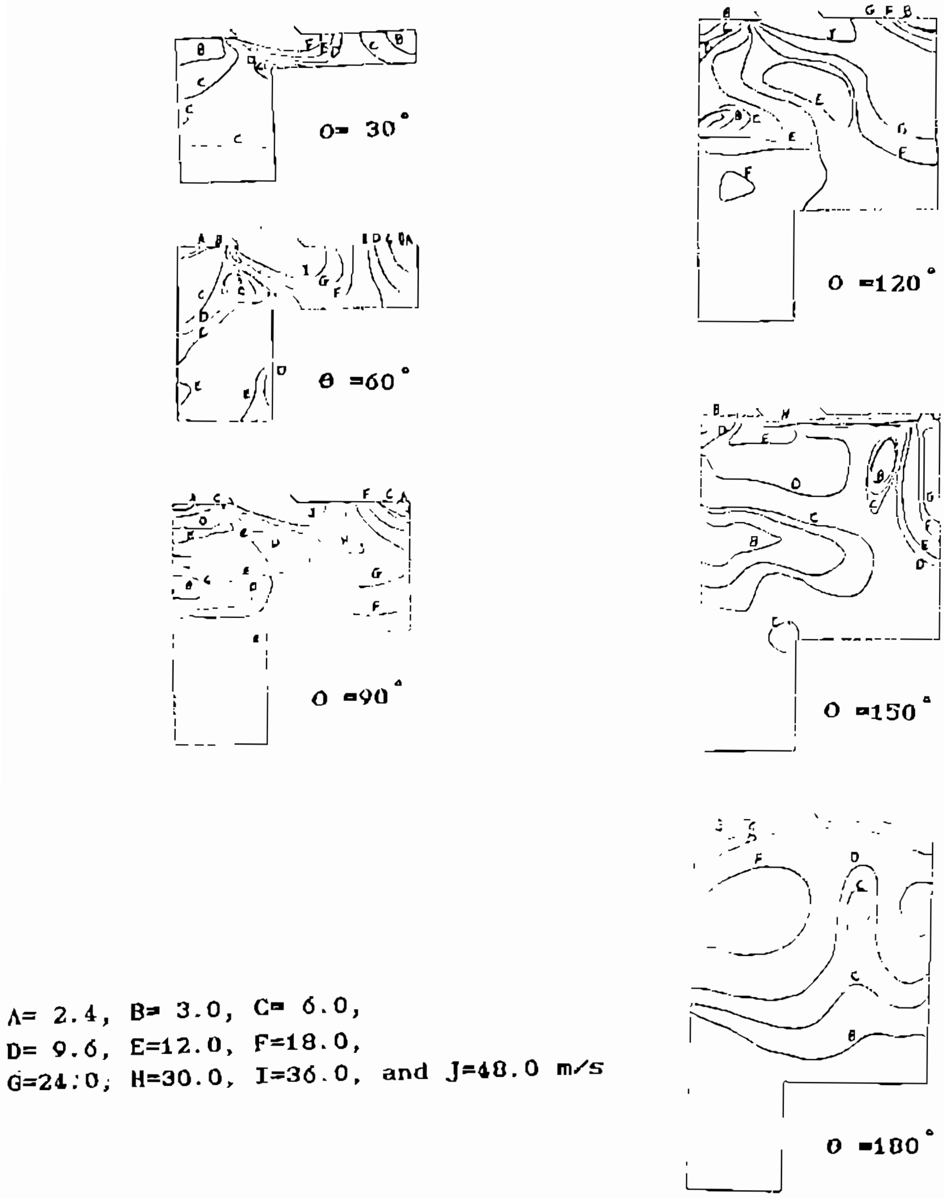
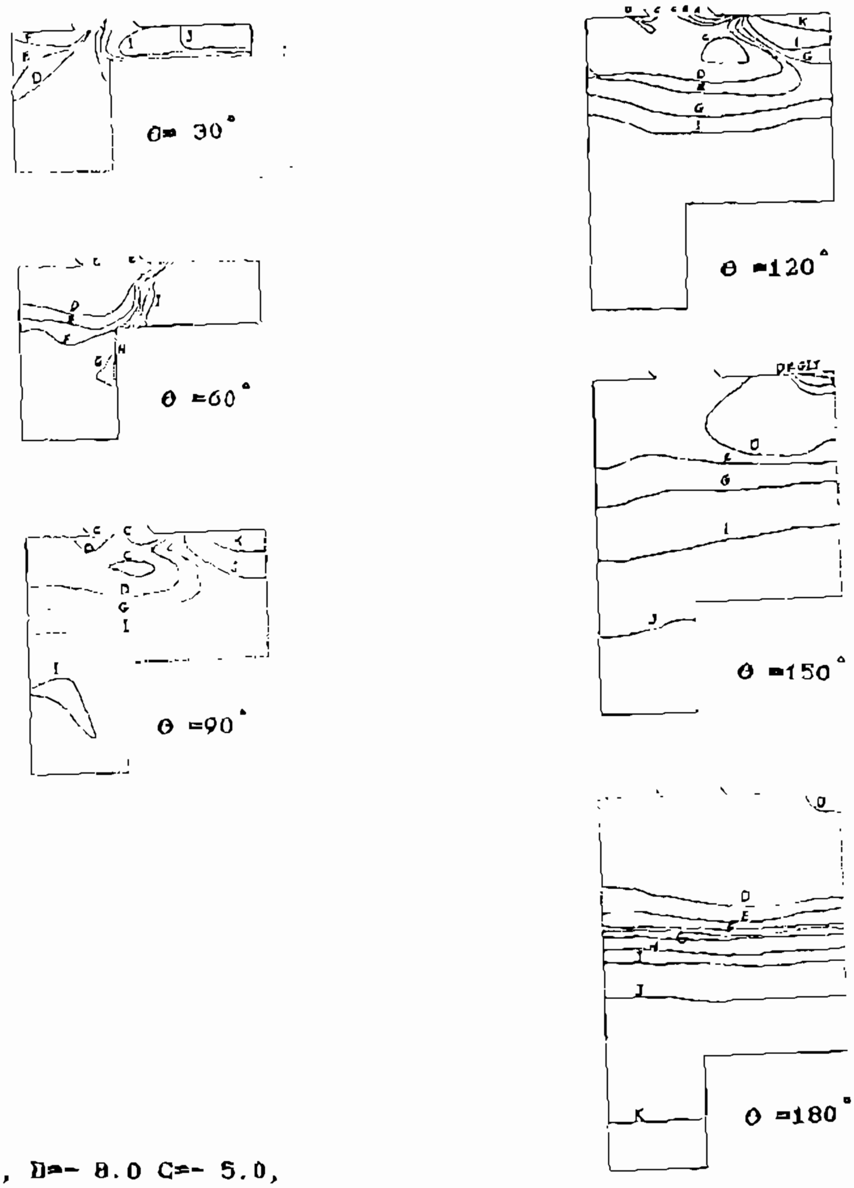
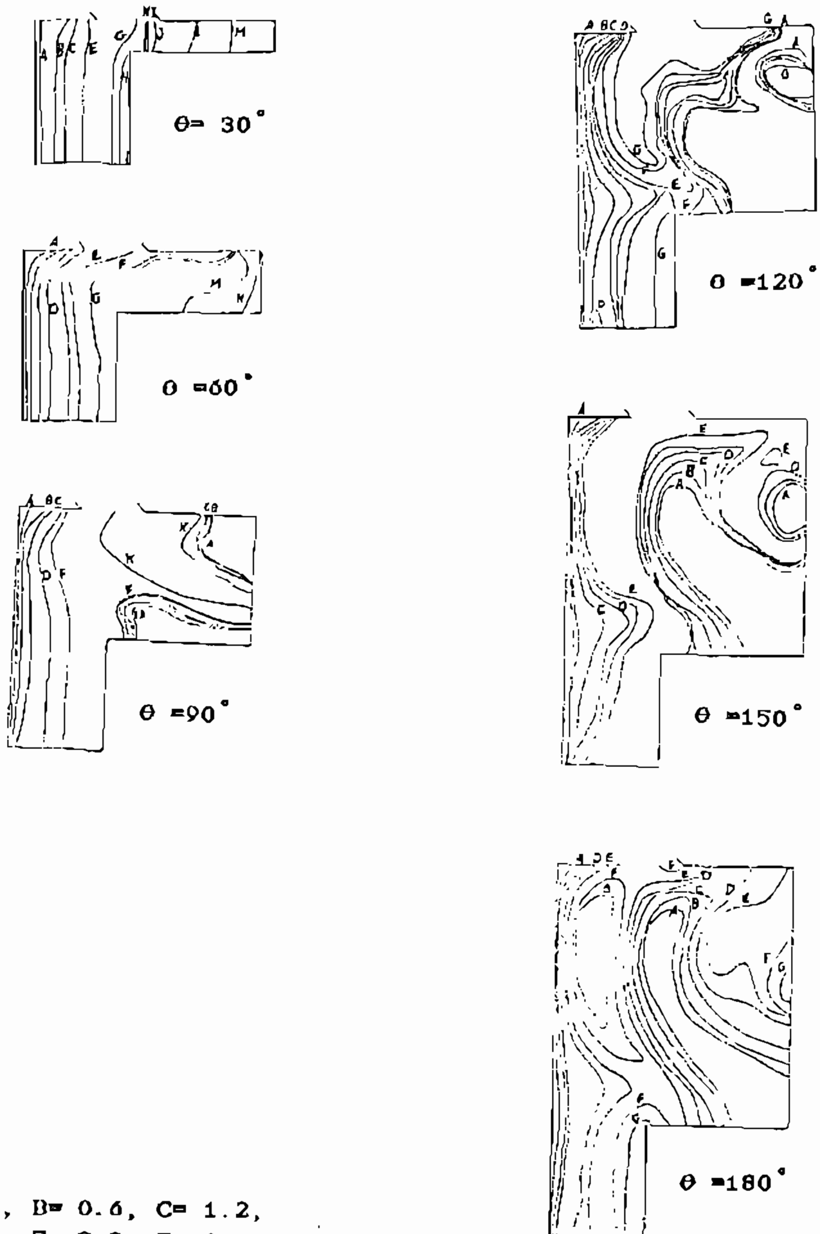


Fig. 4 The contour lines of the resultant velocity in the r-z plane during intake stroke.



A=-10.0, B=- 8.0 C=- 5.0,
D=- 2.0, E=- 1.0, F=- 0.5,
G= 0.0, H= 0.5, I= 1.0, J= 2.0, and K= 3.0KN/m²

Fig. 5 The contour lines of the pressure difference of the flow during intake stroke.



A= 0.3, B= 0.6, C= 1.2,
D= 1.8, E= 3.0, F= 3.6,
G= 4.8, H= 5.4, I= 6.0,
J= 7.2, K=8.4, L=9.0, M=12.0, and N=14.4 m/s

Fig. 6 The contour lines of the tangential velocity during intake stroke.