Three Dimensional Computation of In-Cylinder Flow with Intake Port in DI Diesel Engine

Y.Takenaka, M.Yabe, Y.Aoyagi and T.Shiozaki

Engine R&D Division
Hino Motors Ltd.
1-1 Hino-dai 3-chome
Hino, Tokyo 191
Japan

ABSTRACT

It is important for the development of an engine to investigate the internal flow and many studies have been made on this. Recently, much progress has been made in calculating the flow of an intake port and in-cylinder. The Finite Element Method (FEM) has a good flexibility to fit a complex shape, but its application to internal flow analysis of the engine is very few.

In this paper, a multidimensional modeling by FEM is studied for the flow in an intake port and cylinder with an intake valve. First, the flow in a cylinder liner with an intake port and valve is calculated in the steady state condition. The calculations are performed for three different valve lifts and compared with the measurement.

The flow is investigated for an actual diesel engine configuration, for which the calculation is performed for the in-cylinder flow with the intake port and the moving valve and piston. The unsteady complex flow during the intake stroke is investigated and compared with the steady state case. The flow patterns in an intake port are similar but those in the cylinder are quite different between the steady state case and the transient case.

The comparison between these calculations and measurements shows that the predictions of this modeling are good for the engineering applications. Especially, the finite element method with explicit time discretizing method is effective for the analysis of transient flow in an intake port and piston cavity.

INTRODUCTION

In a DI diesel engine, the flow in the cylinder is one of the most important factors which have a strong influence for improvement of the exhaust emission. Many experimental and numerical analysis have been performed to investigate the flow in a cylinder of reciprocating engine(1)(2). The intake port shape determines the characteristics of the air motion, and the optimization of the intake port shape is a significant problem in the engine design.

In the last several years, the numerical analysis has been developed rapidly by many researchers and enables us to calculate the complex flow in the cylinder and the piston cavity(3)(4)(5)(6). However, as the air motion in the cylinder has a close relation with the intake port shape, it is desired that the calculation be applied to both the intake port and cylinder.

In this study, the numerical simulation by Finite Element Method(7) is applied to the analysis of the flow in the intake port and cylinder both in a steady state and transient case. The steady state analysis is made in the condition of a swirl meter. The transient analysis is made in the engine running condition with the moving intake valve and piston cavity.

METHOD OF CALCULATION

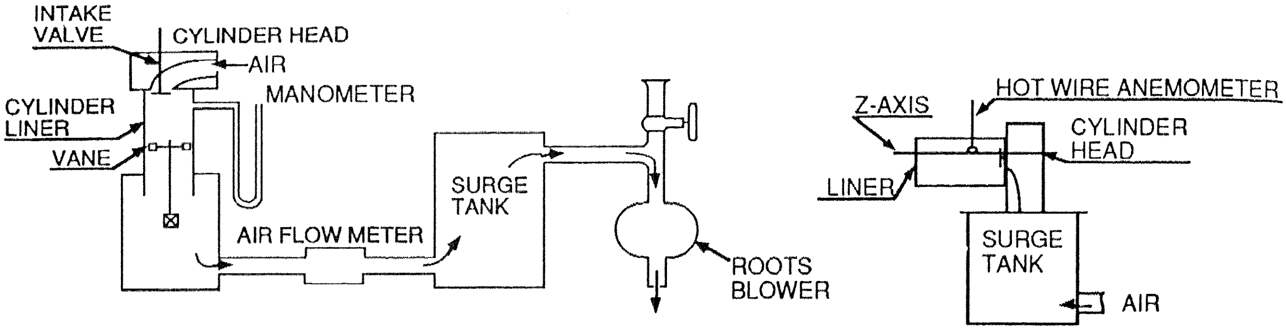
Basic equation

Calculations are performed by solving the Favre(mass)-averaged compressible Navier-Stokes equations for the mass, momentum and energy conservations. The closure of the system is achieved by using the eddy viscosity concept to model the Reynolds stress tensor according to the boussinesq hypothesis. In this study, the subgrid scale turbulence model is used to solve the turbulent flow.

Spatial discretization

The numerical scheme is based on the discretization of the momentum equation by the finite element technique whereas mass and energy are conserved in a control volume. This method enables us to use unstructured meshes and thus to handle the complex geometry of the intake port-valve-cylinder assembly and to refine the mesh only in the region needed(in particular in the valve region and near the wall).

The finite elements used in this study are



Flow rate measurement Velocity measurement

Fig.1 Experimental apparatus

mainly 8 node elements. For the transient case the Arbitrary Lagrangian Eulerian (ALE) technique is used to allow for the moving boundaries. The displacement of each mesh point is automatically controlled by a grid velocity algorithm to allow for a good mesh deformation.

Time discretization

The calculations of this study use an explicit forward marching in time scheme in order to avoid the usual drawback of Finite Element techniques of an inverting a matrix as with the implicit scheme.

Boundary condition

Nonslip boundary condition is adopted at the wall with the use of the law of the wall for the turbulent boundary layer treatment. Stagnation pressure is given at the inlet both in the steady state and transient case, and static pressure at the outlet in the steady state case.

EXPERIMENTAL MEASUREMENT

Figure 1 shows the experimental apparatus in the steady state case. The specifications are described at Table 1. The pressure difference between the inlet and outlet is kept constant, 250mmaq. The air flow rate is measured for three different valve lifts. The apparatus arrangement is different in the case of a velocity measurement as shown in Figure 1.

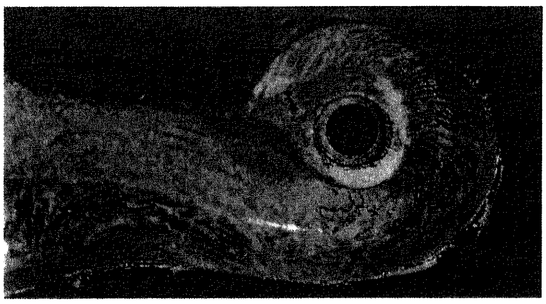
A flow visualization is performed for the intake port and valve. This visualization is obtained by the oil film method. White oil is painted on the intake port and valve. The intake port is cut into upper part and lower part. Figure 2 shows a picture of these visualization results. It is found that the flow is complex at the region around the valve stem and a separation can be seen at the edge of the valve. The streak lines coming from the inlet exit between the intake valve and seat mainly, but the streak lines are in the direction of the valve stem from the wall at the opposite side of the inlet in upper part .

In addition to this, the visualization by the burning carbon particles is performed for the flow in the cylinder. Figure 3 shows the pictures in which a carbon particle starts from the three different position of the inlet. The burning carbon particle coming from the upper position at the inlet turns about 180 degree around the valve stem and exits toward inlet side. The particle coming from the middle position of the inlet turns about 90 degree and exits toward view point. The particle coming from the lower position goes

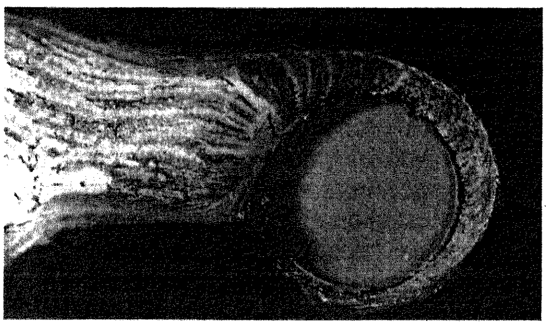
Table 1 Specifications

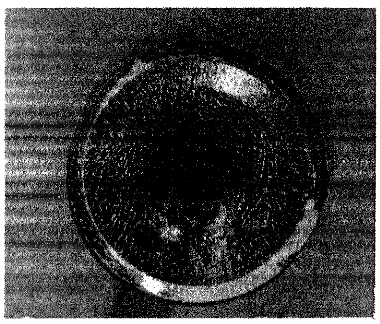
Bore	137 mm
Stroke	150 mm
Type of intake port	Herical
Maximum valve lift	13.93 mm

UPPER PART



LOWER PART





INTAKE VALVE

Fig.2 Flow in an intake port

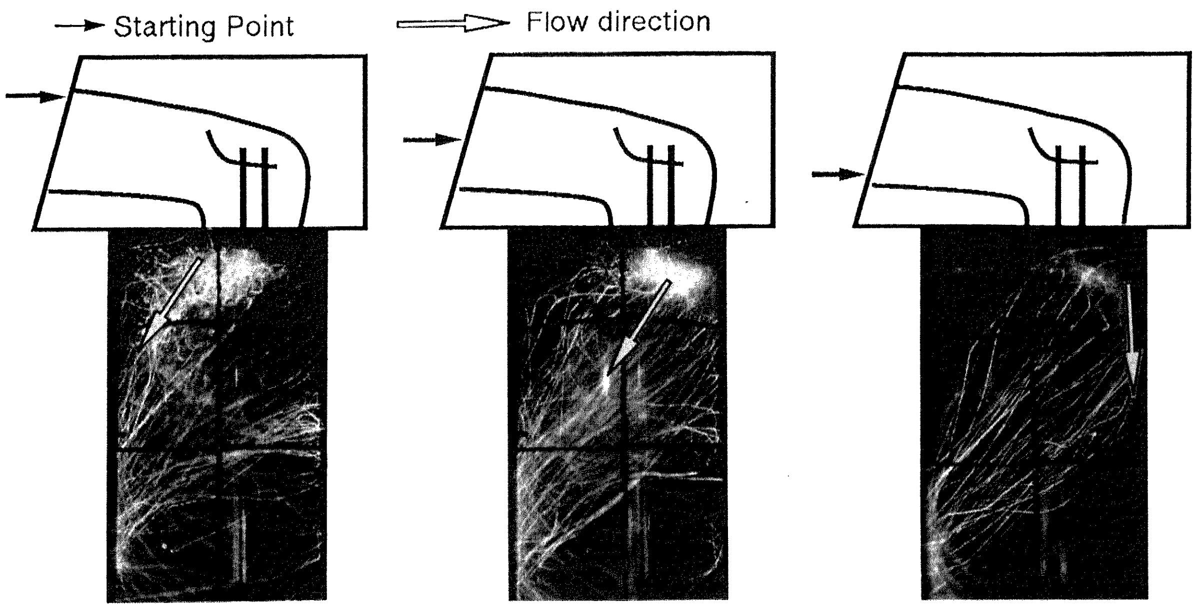


Fig.3 Flow in a cylinder

straight and exits from the opposite side of the inlet. These are the characteristic features of the flow in this intake port.

The velocity distributions are also measured on the horizontal cross section in the cylinder by a hot wire anemometer. The position of the horizontal cross section is located at 1.0D lower from the cylinder head (where D is the diameter of the cylinder). The measurement is limited to only z-direction indicated in Figure 1. Figure 4 shows the velocity distribution. The velocity contour lines are interpolated from 126 measuring points per cross section. The cross section has two high velocity regions and the maximum velocity is almost 23 m/s.

STEADY STATE CALCULATION

Here the detailed features of the air flow are investigated for the maximum valve lift. The computer mesh for the steady state case is shown in Figure 5. The total number of node is about 40000. Most of the mesh is used for the region around the intake valve and top part of the cylinder. Because the

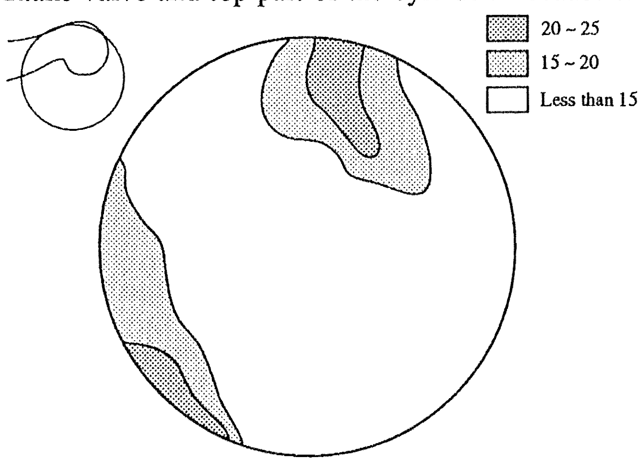


Fig.4 Z-component distribution of velocity by experiment

flow velocity changes rapidly in this region and the influence of this part on the flow pattern is stronger than other parts.

General feature of the flow

Figure 6 shows the flow pattern in the intake port and the cylinder and near the wall using streamlines. The general feature of the flow is that the major flow coming from the inlet goes around the valve stem. Especially, the flow coming from higher part of the inlet exits in an opposite direction. While the air coming from the lower part of the inlet exits in a same direction. The flow has the equivalent radial

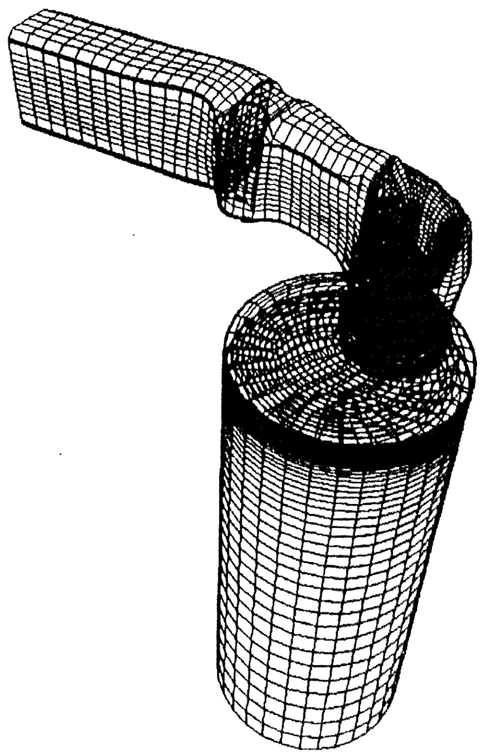


Fig.5 Computer mesh for the steady state case

and axial flow velocity in the region near the wall. This feature of the flow corresponds well to the visualized results. In the cylinder, the air motion consists of high speed downward and tumbling flow, and is complex.

Flow around the valve

Figure 7 shows the velocity distribution on the cross sections cutting the intake valve. A small separation can be observed near the cylinder head wall on both cut plane. On the cross section AA', the velocity vectors which flow from the wall to the valve stem can be seen at the upper part on the opposite side of the inlet, and correspond to the streak line of the visualized result. The velocity magnitude between the intake valve and seat varies with the direction. The opposite side of the inlet on the cross section BB' is the highest. The cross section AA' has the lowest velocity and shows a small separation at the valve seat in the intake valve side.

Velocity distribution

Figure 8 shows the axial velocity component distribution on the same horizontal cross section as Fig.4. The high velocity part more than 25 m/s exist opposite side of the intake valve and near the wall. The very low velocity part exists at the center and have a upward velocity component.

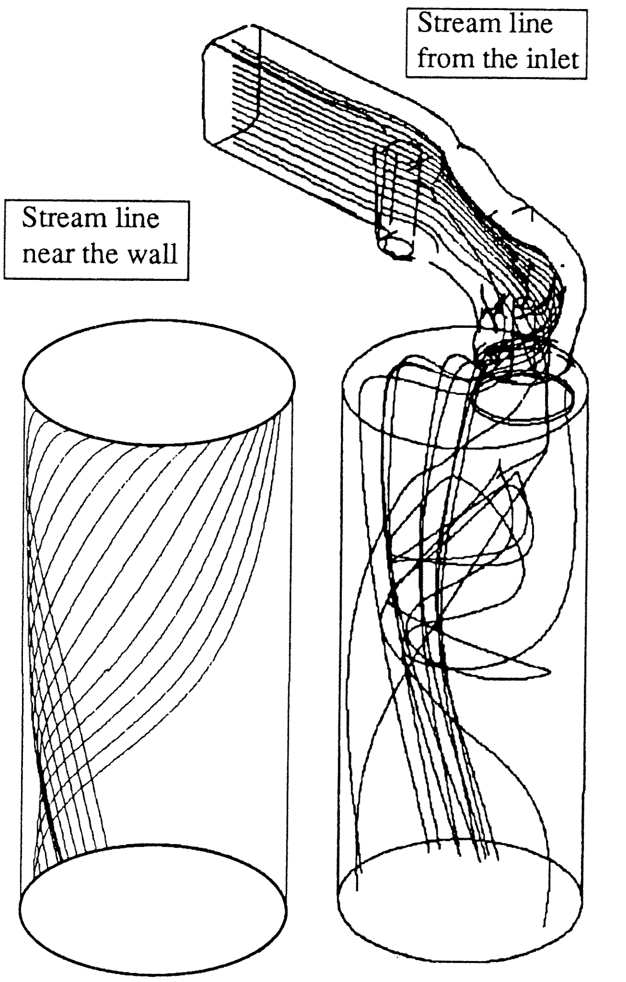


Fig.6 General flow feature in the steady state case

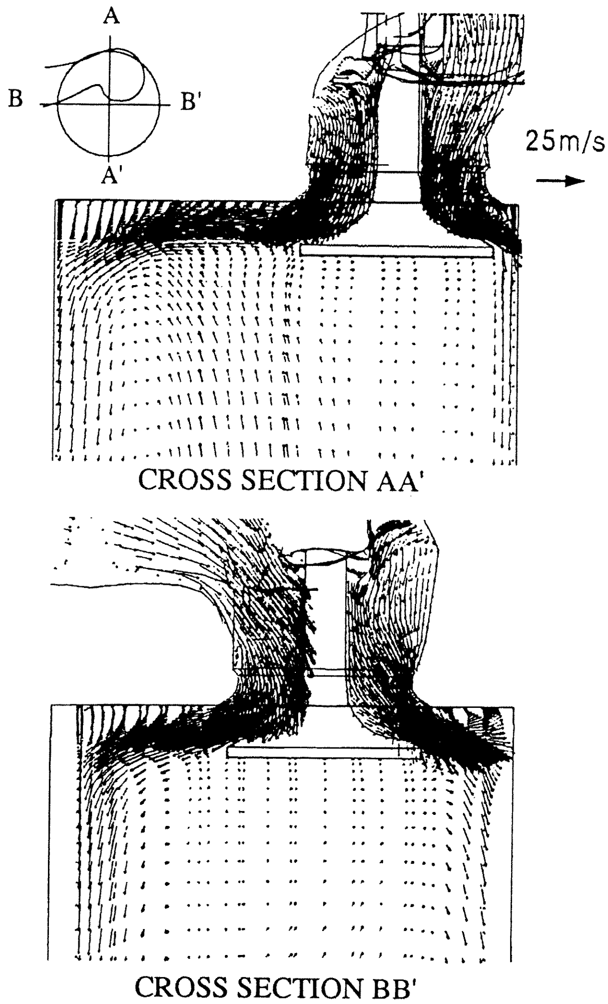


Fig.7 Flow around the intake valve

TRANSIENT CALCULATION

The calculation is performed for the engine running condition as well. Figure 9 shows the computer mesh for this calculation. The intake valve

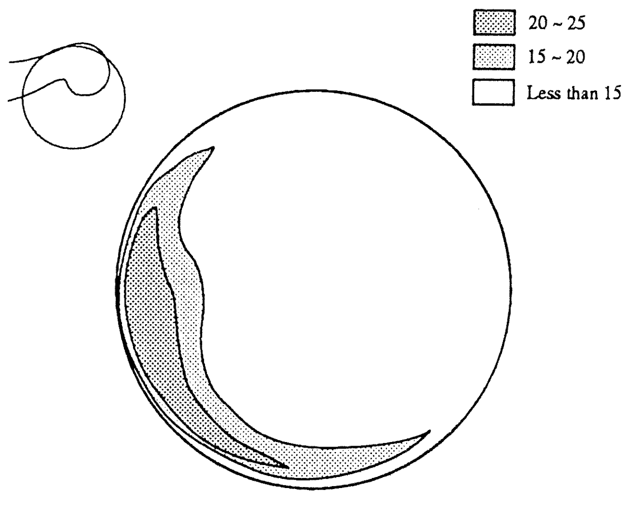


Fig.8 Z-component distribution of calculted velocity

and piston cavity move with the change of crank angle. The boundary condition for inlet pressure is assumed to be a constant value of the atmosphere.

General feature of the flow

Figure 10 shows the general features of the flow by the instantenious stream lines. The flow pattern in the intake port is similar to the one in a steady state case. The flow pattern in the cylinder is however quite different due to the influence of the piston cavity. As crank angle changes, the flow rotates around the valve stem, and the flow pattern in the cylinder is quite complex because of the combination of swirling and tumbling motions.

Flow near the valve and in-cylinder

Figure 11 shows the velocity vector distribution on the cross section cutting the cylinder. The flow near the cylinder wall has a big downward component,and the tumbling motion can be seen.

DISCUSSION

Comparison between calculation and experiment

At first, the comparison is performed for the flow rate. Figure 12 shows the result of a comparison and indicates that the calculated flow rates are in good agreement with experimental one, showing slightly lower(less than 10%).

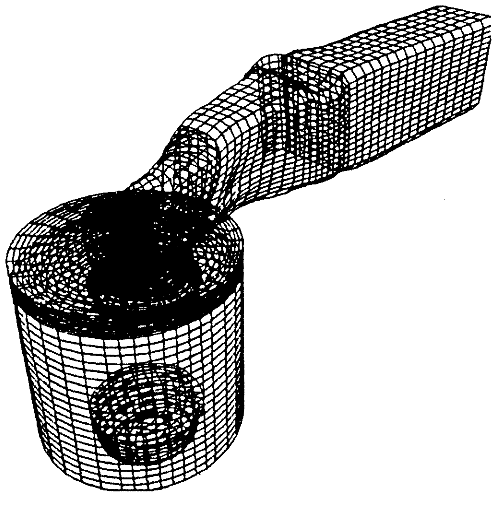


Fig.9 Computer mesh for the transient case

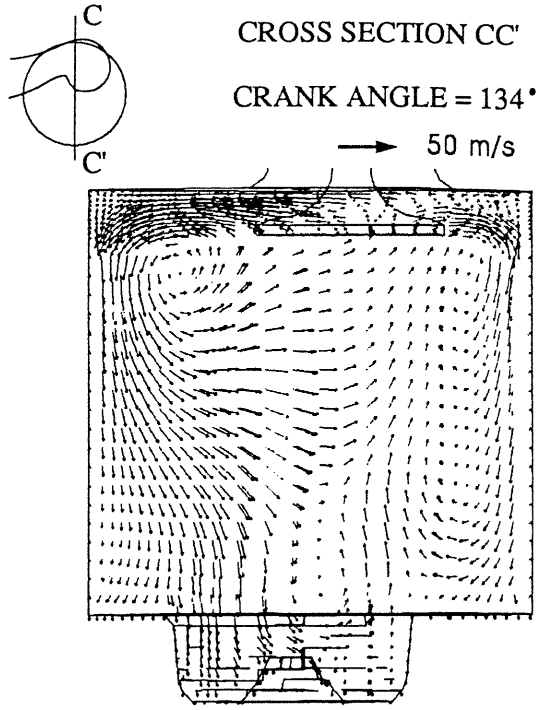


Fig.11 Vertical velocity component in a transient case

From Fig.4 and Fig.9, the velocity distribution is compared at each cross section. The pattern of high velocity region of the calculation is very similar to the experimental result, although these cross section is far from the inlet and located after the complex region like an intake valve part. Not only the flow patterns, but also the magnitude of the velocity component is agree well.

Remaining small disagreements between the calculation and experiment are seemed to be improved by a mesh refinement in the cylinder.

Difference between the steady state and transient case

The general flow pattern can be compared by the stream lines for the steady state in Fig.3 and the instantenious stream lines for the transient case in Fig.10. In the intake port, the flows in both cases have a quite similar pattern and have a different path line according to the starting position at the inlet. After passing the intake valve, the flow pattern is different

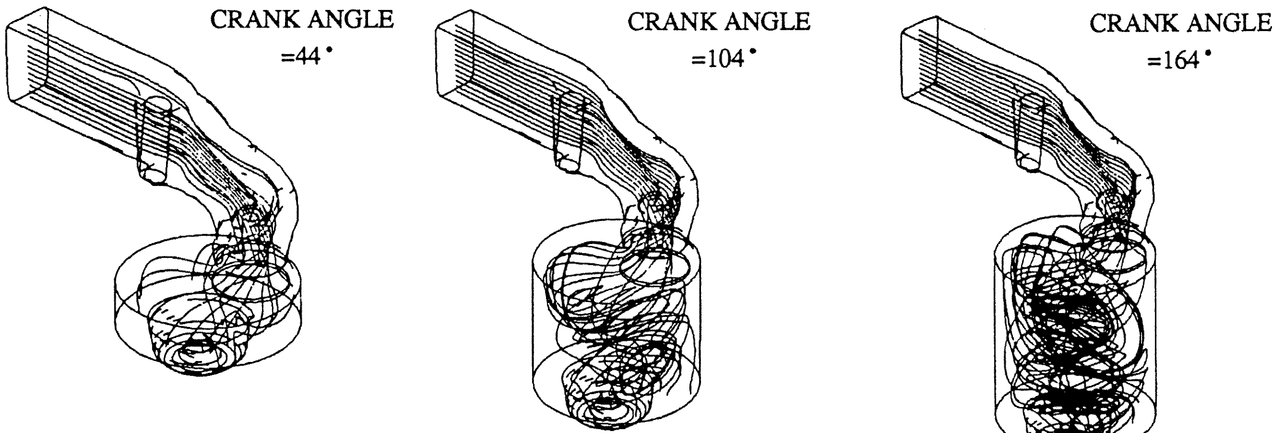


Fig.10 General flow feature in a transient case

between the steady state and transient case in the cylinder. Especially, the flow pattern in the transient case changes with crank angle and is very complex for the existence of the piston cavity.

From the velocity distributions in the horizontal cross section, it is found that the flow patterns in a transient case is different with a steady state case and change with crank angle. The tumbling motion in a transient case is stronger than in a steady state case.

These result of comparison means that not only the steady state flow analysis but also the transient flow analysis is important for the in-cylinder flow.

CONCLUSION

In this study, the intake flow is calculated for the steady state and transient conditions. A comparison between the experiment and calculation is performed. Furthermore the details of the flow in both cases is clarified from the calculated results. Especially, the difference between the steady state and transient case is investigated.

The conclusions of this study are:

1. The numerical simulation for the intake port valve-cylinder flow has been achieved both in the steady state and transient condition with moving valve and piston.
2. The comparison between the experiment and calculation in a steady state case shows that the agreement are quite good for the flow pattern and absolute value of the velocities. The good agreement is also found for the flow rate. It is thought that these agreement can be further enhanced by a mesh refinement.
3. The comparison of the calculated results between the steady state case and transient one leads to the following conclusion.
   -The flow pattern in the intake port is similar in both cases.
   -The flow pattern in the cylinder is different in both cases because of the influence of the piston cavity.
4. Finite Element Method provides a good capability for analysis of a complex shape region like an intake port and valve.

ACKNOWLEDGMENT

The author would like to thank Mr.E.Tamura and Mr.S.Obana of IBM Japan Tokyo NIC Center for their computational support. The author also would like to thank Mr.A.Tabbal, Mr.g.Munck, Mr.S.Aita, Ms.N.Montmayeur, and Mr.J.F.Levy of ESI for their support on the numerical simulation.

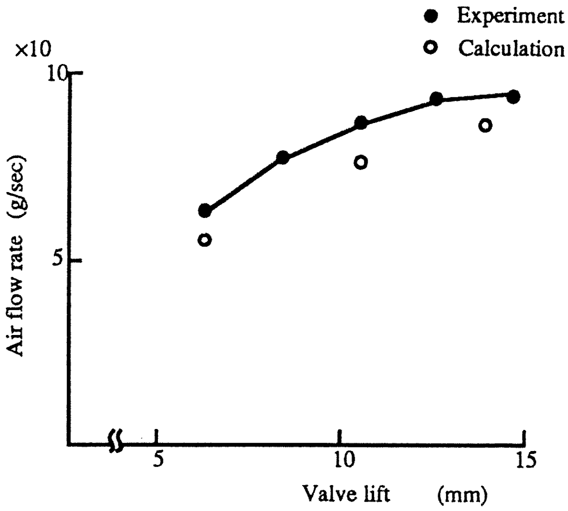


Fig.12 Flow rate of experiment and calculation

REFERENCE

(1)Transactions, Proceedings or Journal Articles:

1. C. Bertoli, F. E. Corcione, G. Police and G. Valentino, "Effect of Inlet Port Geometry on Turbulent Air Flow in the Combustion Chamber of a D.I. Diesel Engine," Proc. COMODIA, 1985, p.273

2. R.S.W.Cheung, S.Nadarajah, M.J.Tindal and M. Yianneskis, "An Experimental Study of Velocity and Reynolds Stress Distributions in a Production Engine Inlet Port Under Steady Flow Conditions," SAE paper No. 900058, 1990

3. Y.Isshiki, Y.Shimamoto and T.Wakisaka, "Numerical Prediction of Effect of Intake Port configurations on the Induction Swirl Intensity by Three Dimensional Gas Flow Analysis," Proc. COMODIA, 1985, p295

4. S. Aita, A. Tabbal, G. Munck, K. Fujiwara, H. Hongoh, E. Tamura and S. Obana, "Numerical Simulation of Port-Valve-Cylinder Flow in Reciprocating Engines," SAE paper No. 900820

5. Y. Takenaka, Y. Aoyagi, T. Tsuji,and I. Joko "3D Numerical Simulation of Fuel Injection and Combustion Phenomena in DI Diesel Engines," SAE paper No.890668

(2)Reports:

6. A. A. Amsden, J. D. Ramshow, P. J. O'Rourke, and J. K. Dukowicz (1985) "KIVA:A Computer Program for Two- and Three-Dimensional Fluid Flows with Chemical Reactions and Fuel Sprays," Los Alamos National Laboratory, LA-11560-MS.

7. ESI (1989) "PAM-FLUID Technical Presentation" Engineering Systems International, 20 rue Saarinen, 94578 Rungis-Cedex (France).