Prediction of Intake Swirl Applying CFD Technique

Y.Oishi, M.Otake and Y.Watanabe

Nissan Diesel Motor Co., Ltd.
1-1 Ageo,362
Japan

ABSTRACT

To obtain a desired swirl intensity and a higher flow coefficient, the shapes of intake port have so far been determined experimentally on swirl test rigs. Because this is very time-consuming work, this paper studies an application of computational fluid dynamics (CFD) for predicting intake swirl intensity. A commercially available CFD code is used and the accuracy of prediction is evaluated by comparing calculation results with results measured on a swirl test rig when using a CAM system to produce 11 variations of intake ports. To increase modeling accuracy, software for generating calculation models based on the same CAM database was developed. The calculated values for higher flow rate and swirl intensity are underestimated but the relationships between measured and calculated values are well described by second-order polynomial equations. Accordingly, the accuracy with which both flow rate and swirl intensity are estimated can be improved by using these empirical equations to correct the estimated values.

Because intake swirl plays an important roll in the formation of mixtures, it is a key factor determining diesel combustion. To optimize the air intake swirl, several intake port shapes - such as directional ports, spiral ports and combinations of these - have been developed. To obtain desired swirl intensity and a higher flow coefficient, intake port shapes have so far been determined experimentally on a steady-state swirl test rig. Because experimental methods are very time-consuming, many researchers have attemted to predict intake port performance analytically by using a technique based on computational fluid dynamics (CFD)[(e.g. 1-3)]. This paper deals with our experience.

CFD CODE

This investigation used a commercially available CFD code, STAR-CD[(4)]. The basic equations are the equation of continuity and the Navier-Stokes equations. It employs special algebraic formulae, often called "wall functions", to represent the distributions of velocity, temperature, turbulence, energy, etc. within the boundary layers which form adjacent to walls. It employs also mathematical models of turbulence to determine the Reynolds stresses and turbulent scaler fluxes. The widely used "K- ε"model was selected in this analysis from among three alternative turbulence representations. The difference equations governing the conservation of mass, momentum, energy, etc. within the flow are discretised by the finite volume method.

CALCULATION MODEL GENERATION

Unification of solid-model database

Formerly, three-dimensional solid models were generated with FEM-model-generating software, CAEDS, based on two-dimensional CAD data. This method, however, had the following problems impairing prediction accuracy:

(1) The complicated shape of intake ports in diesel engines made it difficult to unconditionally define a three-dimensional solid model based on two-dimensional CAD data.
(2) The port shape of an experimental solid model produced by the CAM process could not be guaranteed to be identical to that of the corresponding calculation model.
(3) The port shape must be greatly simplified when using CAEDS to define the solid model.

We therefore developed a system that unifies the database for CAE and CAM, setting the CAD data for CAM as the primary design data. Figure 1 shows, for comparison, the model generating process of the former method and that of the database unification system. The modeling accuracy, can be evaluated by comparing the volumes of five port variations between the solid model for CAM and that for CAE (cf. Figure 2). Because the base data of the database unification system is the CAM data for manufacturing an intake port by using a numerically controlled machine, even parting lines and leaves can be faithfully modeled. Consequently, the volume difference can be maintained within 0.5%. Furthermore, the definition of a solid model with CAEDS becomes unnecessary, and the time needed for generating the simulation model can be reduced by about 60%.

Modeling a swirl test rig

To represent measured results on a wind-wheel-type swirl test rig, a simulation analysis was performed in modeling the airflow path from a part of the atmosphere adjacent to the port

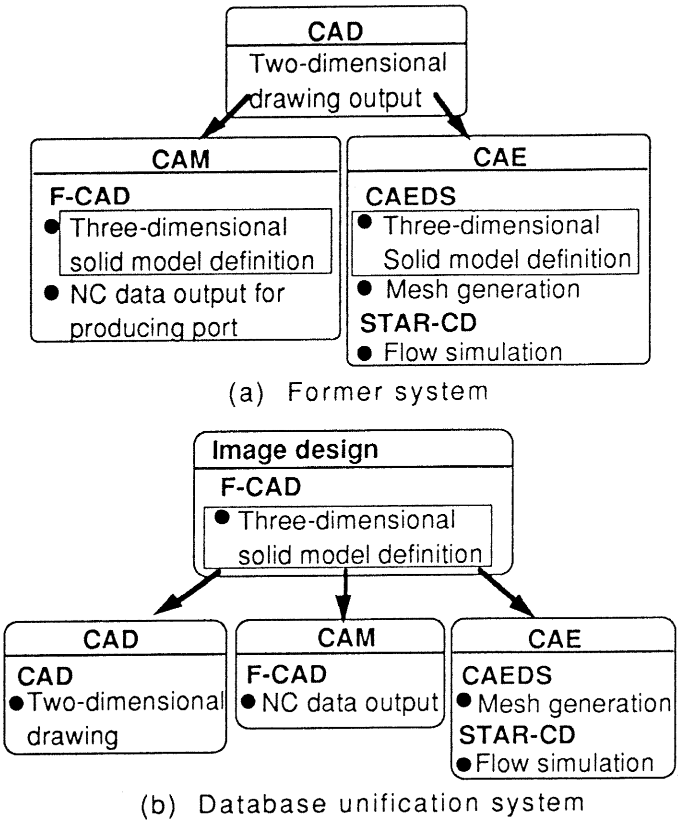


Fig. 1 Comparison of model generating processes

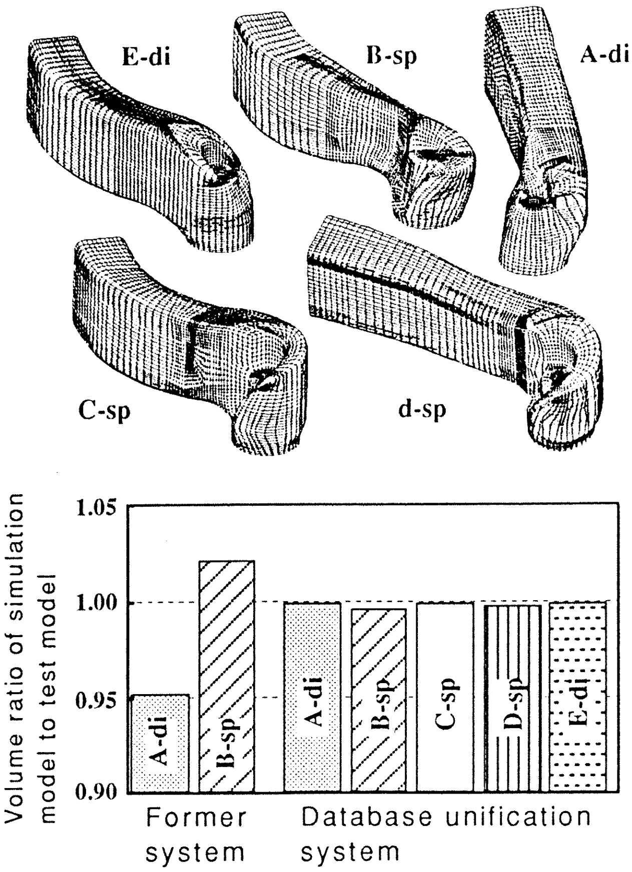


Fig. 2 Comparison of modeling accuracy
(Note: di: directional port; sp:spiral port)

inlet to the site of the flow rate measurement (cf. Figure 3). There is a rectangular bend downstream from the pressure measuring point in the swirl test rig, so we analyzed the effect of this bend. The pressure loss and averaged tangential velocity at several points between with and without the bend are compared in Figure 4. Velocity becomes faster at the port inlet (Part C) because of the reduced flow area, resulting in pressure loss of about10%. To improve the representability of the simulation analyses, we must ensure that the boundary conditions comply with the experimental ones. The atmospheric condition adjacent to the port inlet was modeled in the form of a balloon (Figure 3). When the bend is considered, the averaged tangential velocity calculated is 6% less than it is when the bend is not considered, and the flow rate is 30 % less. Modeling the bend and connected

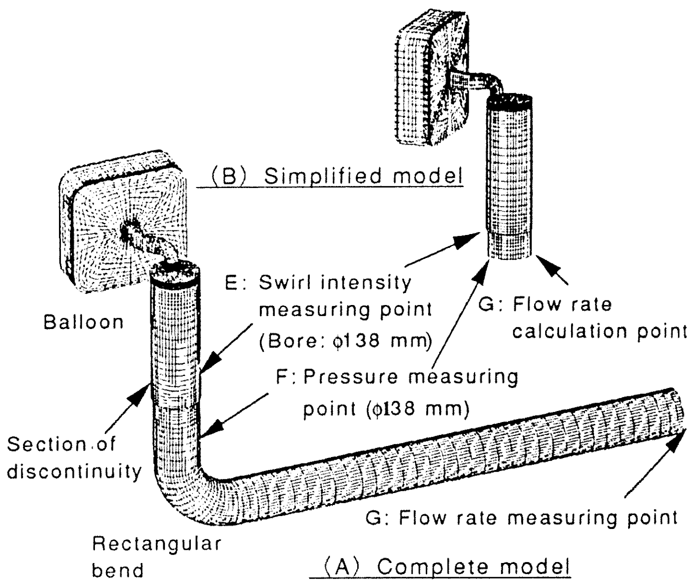


Fig. 3 Simulation models

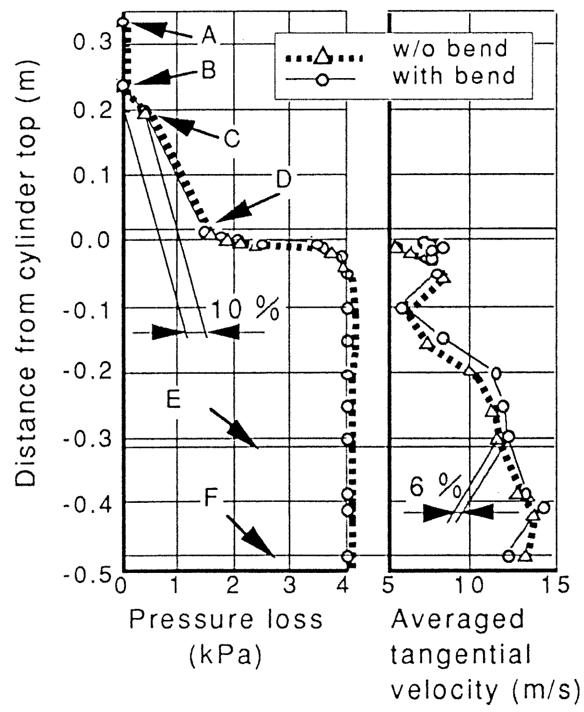


Fig. 4 Effect of bent

straight pipe is time-consuming for generating a model and for computation. Accordingly, models simplified by neglecting them were used in the further calculations, and the flow rate values calculated were corrected for the effect of neglecting them (which effect had been obtained beforehand through a preliminary simulation analysis). Calculated swirl intensity values were not corrected because their discrepancy was relatively small .

Study of mesh size effect

A complicated and very fast flow-field is formed in the valve seat region where air is streaming into the cylinder from the intake port. As a result, the size of the calculation mesh for this region has a large influence on the calculated downstream flow in the cylinder. We therefore studied the influence of mesh size on calculated results.

Mesh size of valve seat region. A larger mesh size averages flow damping, resulting in a larger calculation error. To study this effect, finer models were generated by either bisecting or trisecting the members of the original grids for the intake port (B-sp), as shown in Figure 5. Figure 6 shows that making the calculation mesh finer increases the calculated swirl intensity and flow rate and that this effect is greater for the swirl intensity. The difference between bisecting and trisecting is not evident; accordingly, a finer mesh size achieved by bisecting improves the prediction accuracy with a slight increase in computational time. We further studied other ports to see if similar effects were also obtained there. As shown in Figures 7 and 8, the effect of the finer mesh on flow rate values is relatively small. For spiral ports with high swirl intensity, the swirl intensity values calculated with the finer meshes increase by about 12% and come closer to the experimental results. For the directional ports with low swirl intensity, however, the swirl intensity values calculated with the finer meshes becomes lower by 4 to 8 percent and come even closer to the experimental values.

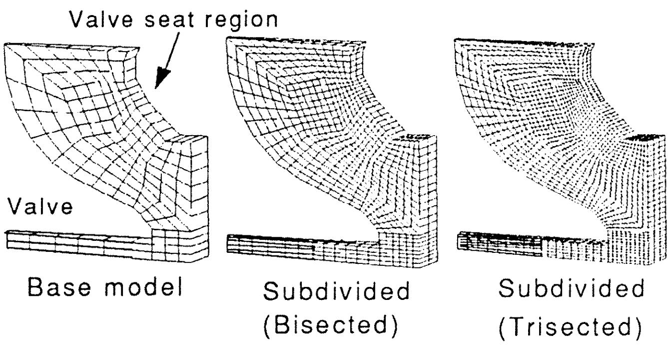


Fig. 5 Cmparison of subdivided models of valve seat region

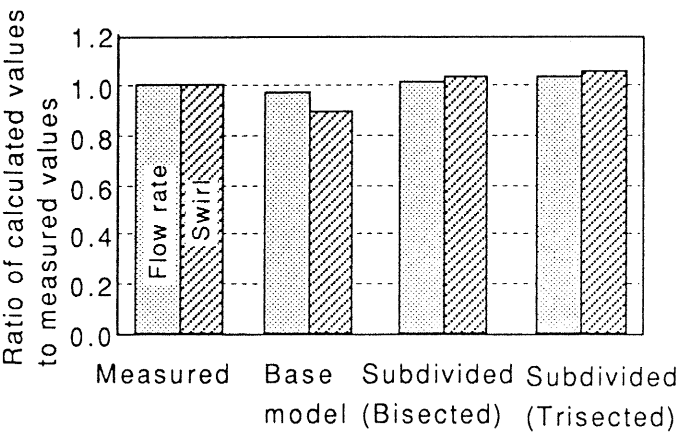


Fig. 6 Effect of subdividing (Port B-sp)

Mesh size of cylinder region. To study the effect of the mesh size of the cylinder downstream from the valve seat region, a finer cylinder model was generated by bisecting the grid members of the original cylinder model. Figure 9 compares the calculation models, and Figure 10 shows results calculated for two ports. Making the mesh finer evidently does not affect calculated swirl intensity or flow rate.

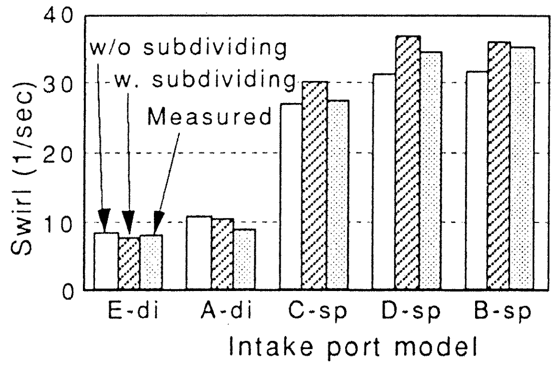


Fig. 7 Effect of subdividing on swirl

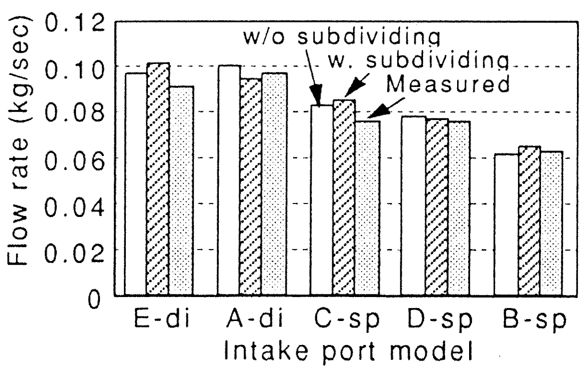


Fig. 8 Effect of subdividing on flow rate

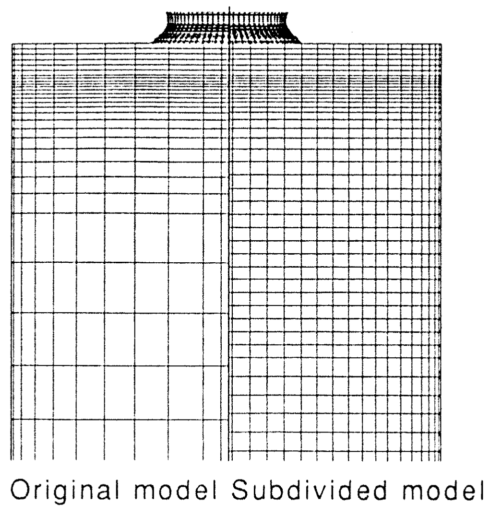


Fig. 9 Comparison of original and subdivided models of cylinder region

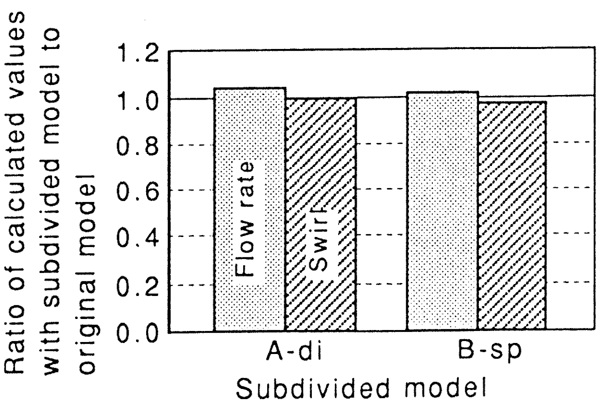


Fig. 10 Effect of subdividing

VALIDATION OF ANALYSIS RESULTS

To evaluate the prediction accuracy, we compared calculation results with results measured on a swirl test rig. We compared results obtained while producing eleven port variations: five single ports (cf. Figure 2), and six combinations of two different ports, (Figure 11). The higher the flow rate and swirl intensity, the lower the calculation results. However, the relationships between measured and calculated values of flow rate and swirl intensity can be described with second-order polynomial equations. Accordingly, the accuracy with which flow rate and swirl intensity are predicted can be improved by correcting the calculated values with these empirical equations.

STUDY OF IN-CYLINDER FLOW

Effect of intake port shape on in-port flow

To study the effect of port shape on flow in a port upstream from the valve seat area, we compared results calculated for two directional ports and three spiral ports. To compare only the effect of the port shape, velocity vectors around the valve stem on the plane over the valve seat 10 mm upward from the cylinder head bottom surface are indicated (cf. Figure 12). For the directional ports the flow vector distribution is symmetric with respect to the valve stem axis, that is, the direction of the main flow faces toward the valve centerline, and the air flow separates into two around the valve stem and the divided flows collide and meet one another downstream from the valve stem. For the spiral ports the direction of the main flow is tangential to the port wall; that is, the airflow swirls around the valve stem and along the inner wall of the port. Thus the port shape determines the direction of the main flow and swirling flow characteristics for both directional and spiral ports.

Effect of intake port shape on swirl in cylinder

Calculation results (Figure 13) show that the swirl formed by a directional port remains almost constant along the cylinder but that the swirl formed by a spiral port reaches a peak value around the swirl measuring point (200- 300 mm from the cylinder top) and is then damped rapidly. The swirl formed by a combination of two ports has no peak, except for the swirl formed by combinations including port B-sp. The tangential velocity of port B-sp hardly damps even in a combination with another port.

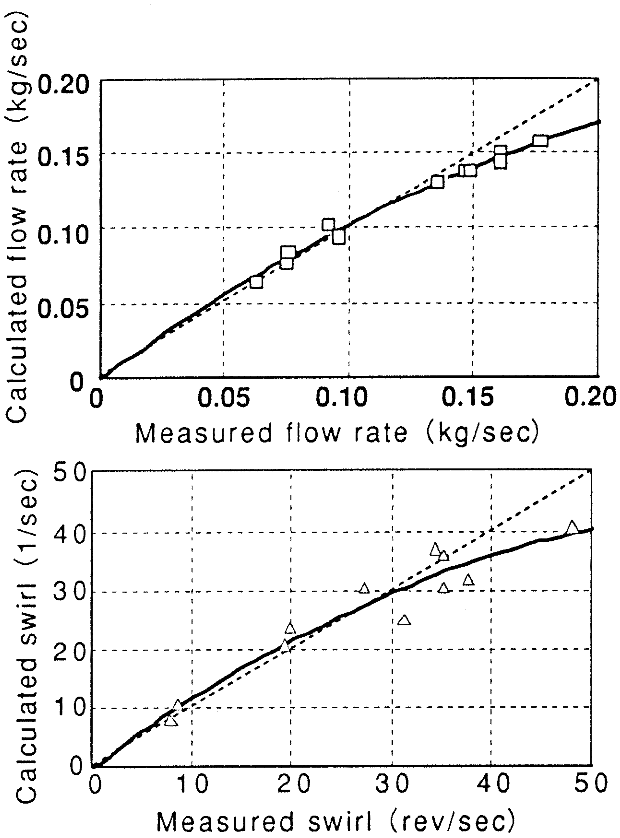
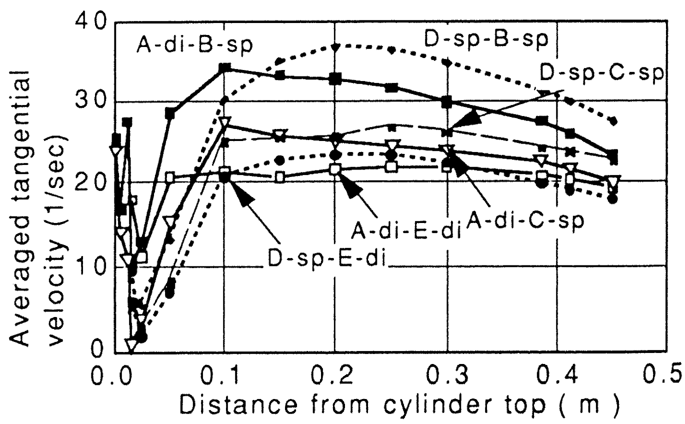
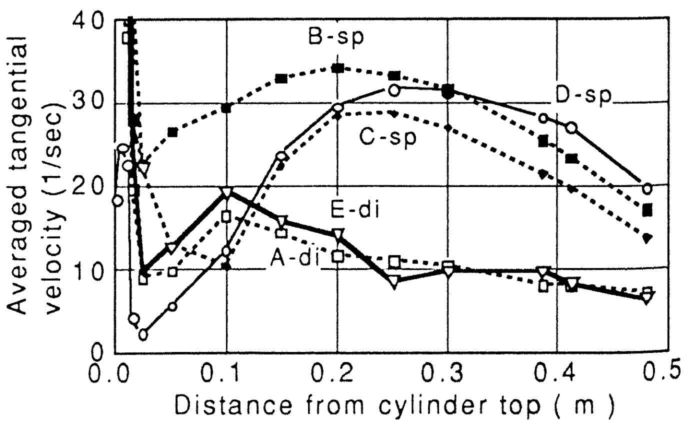


Fig. 11 Correlation between calculated and measured values



(A) 4 valve head



(B) 2 valve head

Fig. 13 Effect of port shape on swirl in cylinder

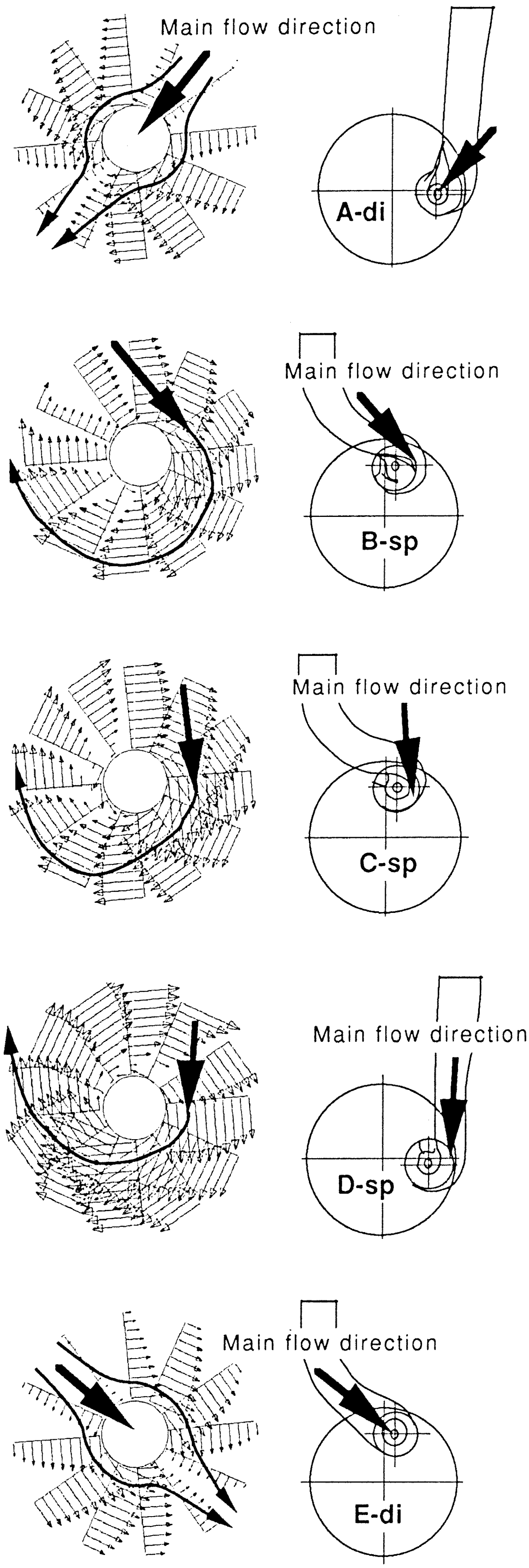


Fig. 12 Comparison of swirl in intake port

Effect of intake valve position

To investigate the effect of the eccentricity of the intake valve center toward the cylinder center on swirl, we analyzed cases, for both directional and spiral ports, in which there was no eccentricity. To reduce the influence of the flow along the cylinder wall, the enlarged cylinder diameter of 260 mm was selected. Figure 14 shows velocity vector distributions over the valve seat area, and Figure 15 shows particle traces. Swirling flows formed in a intake port are the almost the same as those for the normal valve position (Figure 12), but almost no swirl is generated in the cylinder even for the spiral port. Accordingly, not only the port shape but also the eccentricity of the intake valve position plays a large role in swirl generation.

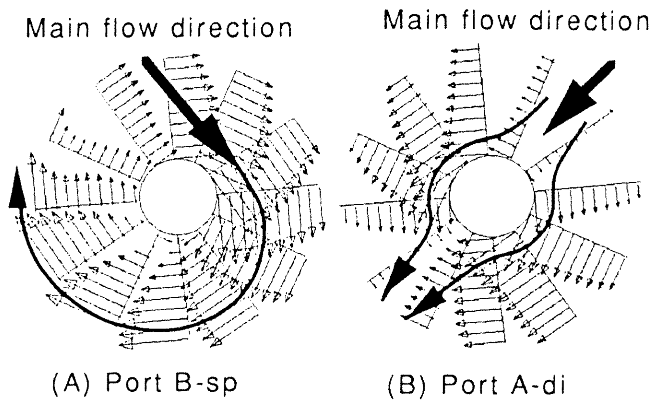


Fig. 14 Velocity vector distribution in intake port

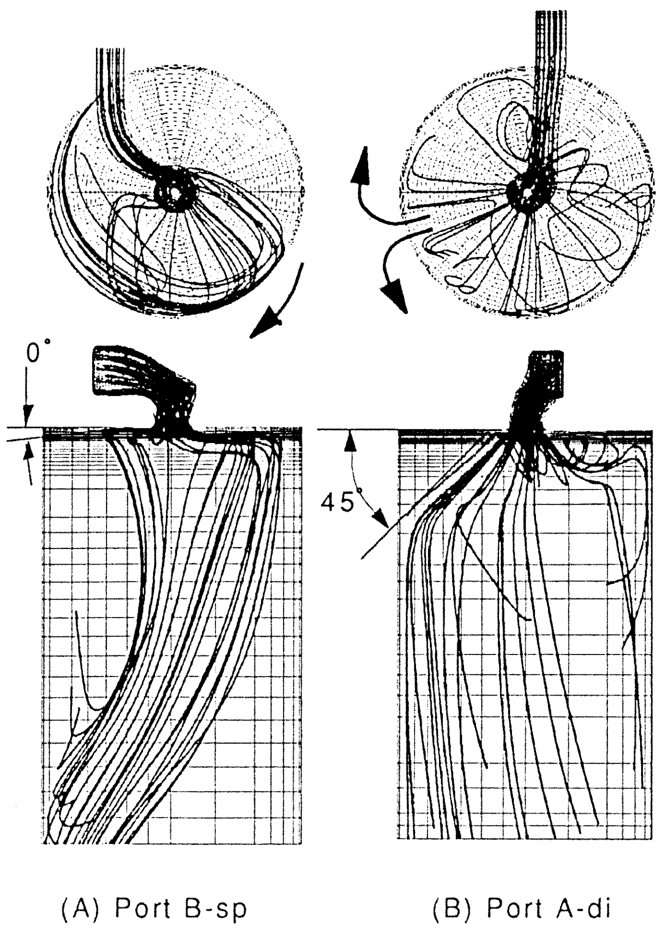


Fig. 15 Comparison of particle traces

CONCLUSIONS

To investigate the feasibility of predicting intake swirl intensity by using CFD-based techniques, we compared the results of simulation analyses with those of rig tests. The following results were obtained:

(1) Using the solid model database for CAM as the basic data for generating computational mesh reduces the volume difference between calculation models and experimental models to within 0.5% and also reduces the time needed for mesh generation by 60%.

(2) The calculated values for higher flow rate and swirl intensity are underestimated, but the relationships between measured and calculated values are well described by second-order polynomial equations. Accordingly, the accuracy of predicted values can be improved by using these empirical equations.

REFERENCES

(1) Gosman, A. D., and Ahmed, A. M. Y., "Measurement and Multi-dimensional Prediction of Flow in a Axisymmetric Port/Valve Assembly", SAE-paper No.870592, 1987

(2) Aita, S., Tabbal, A., Munck, G., Montmayeur, N., Takenaka, Y., Aoyagi, Y., and Obana S., "Numerical Simulation of Swirling Port-Valve-Cylinder Flow in Diesel Engines", SAE-paper No. 910263, 1991

(3) Ntone, F., and Zehr, R. L., "Multidimensional Fluid Flow Calculations in Diesel Engine Exhaust Valve-Port Geometries", SAE-paper No.930073, 1993

(4) STAR-CD MANUALS, Computational Dynamics Limited