論 文 SAE No. 933716 自動車工學會誌 / Vol. 15, No. 2, 1993/105

Numerical Experiment for Compressible Effects around a Butterfly Valve

Butterfly 밸브 주위에서의 난류유동의 압축성 현상에 관한 연구

김 철 호*, M. Behnia**, B.E. Milton**
C. H. Kim

초 록

내연기관의 혼합 가스 생성 장치로 흡입되는 공기의 유량을 조절하기 위해 Butterfly 밸브가 사용된다.이 밸브는 유량의 제어에는 매우 유용한 반면, 밸브 후면에서의 복잡한 공기의 유동 현상[6]으로 인하여 혼합가스의 생성에 장해적인 요소를 제공하기도 한다. 특히 밸브가 많이 닫힌 상태에서는 밸브와 관로벽 사이의 간격이 좁아지는 Throttling 현상으로 인하여, 엔진에 고 부하시 흡입되는 공기의 양이 증가하게 되어, 밸브 주위에서 공기 속도의 급증으로 인하여 유동장의 압축성 현상을 기대할 수 있다. Throttle의 Choking 현상으로 인하여 공기의 입자는 운동에너지를 잃게 되고, 궁극적 혼합가스의 생성에 영향을 미쳐 엔진성능의 저하를 초래하게 된다.

본 연구에서는 실제 엔진(Central Fuel Injection Engine, 5 liters) [1]의 흡인 manifold를 modelling하여, 특히 밸브가 많이 닫힌 상태에서, 밸브 주위에서의 난류 유동의 압축성 현상을 정량적으로 분석해 보았다.

1. INTRODUCTION

The main component of the mixture gas generation system of a single point injection S.I. engine consists of a fuel injection nozzle to spray fuel into the air stream, and a butterfly valve to control the air/fuel mixture gas flowrate into the combustion chamber as shown in fig.1. A base airflow field analysis in the mixture gas generation system has an important meaning to improve the performance of the engine. Because of sudden changes of the operating conditions of an engine

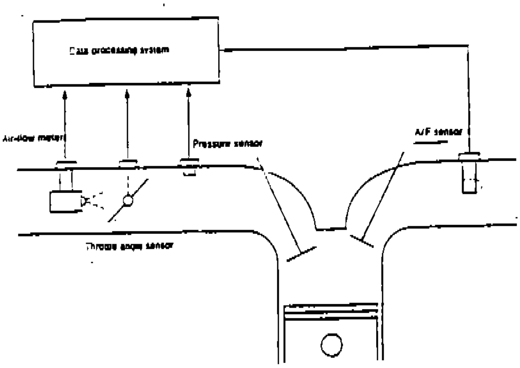

Fig.1 Schematic diagram of a single point injection S.I. engine

* 정회원, 호주 New South Wales대학교 기계공학과 박사과정
** The University of New South Wales, Sydney, Australia

(transient conditions), the airflow field in the manifold may experience quite different flow patterns and flow characteristics[2] which may considerably interfere with the generation of well-mixed gas which is required for perfect combustion in the combustion chamber. Especially when the valve has very small angle from the vertical, the flow field at the back of the valve is very complicated and it is hard to avoid compressible effects and sometimes the sonic flow region[3] at the tip zone of the valve. In this case, the flow simulation method has to use the compressible flow analysis method because the problem of heat transfer through the media has a great effect on the flow field.

In this study, a numerical experiment has been conducted to find out the compressible flow phenomenon in the airflow field around the valve as the valve has a small opening angle because this phenomenon has critical effects on the generation of the well-mixed gas for combustion. Air mass flowrate(m) and the valve opening angle(θ) have been chosen as primarily independent variables for the experiment. For the practical approach, the performance curve of the intake manifold of a 5-liter Central Fuel Injection engine[1] was modelled.

2. NUMERICAL MODEL

The manifold geometry of the 5-liter Central Fuel Injection Engine[1] was employed for the numerical modelling of flow around a butterfly valve for this study. The details of the characteristic airflow data of a manifold system are given in table 1 and Fig.2, Fig.3 shows a typical computational grid with 35°of valve angle(θ). To keep the real valve size on the grid generation for each valve angle, the blockage ratio(R) was defined as the area of the throttle pate divided by the duct cross-sectional area. Even though it was not convenient to keep the same valve size as the real one on the grid generation for each valve angle, the error ratio is not over 0.3%. As mentio-

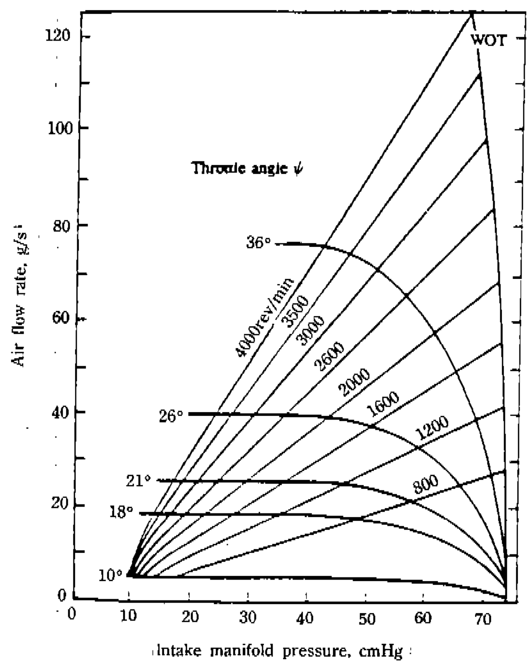

Fig.2 Airflow characteristic curve with respect to the change of valve angle(θ) and engine speed(rpm)

Table 1 Typical Valve Angles and Corresponding Inlet Air Velocity(Ui)

valve angle(θ)	blockage ratio(R)	Reynolds No.(Re)	Ui(m/s)
45.0°	0.9706	45,000~175,000	11.05~42.95
35.3°	0.9670	38,000~112,000	9.33~27.69
26.6°	0.9690	38,070~ 63,260	9.34~15.50
22.2°	0.9720	20,370~ 41,970	5.00~10.30

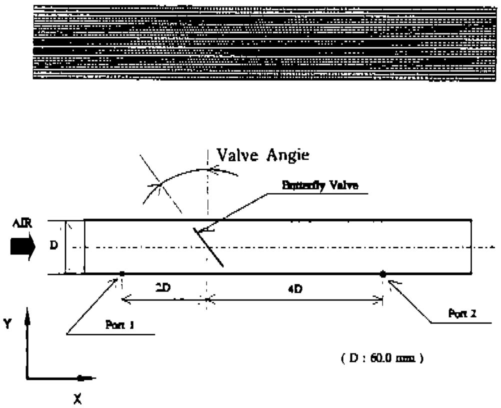

Fig.3 Duct geometry and a typical numerical grid(23×134)

ned above, the airflow field is very sensitive to the change of inlet boundary conditions and to the valve angle(θ) which is measured from the vertical. The angle was changed from 22.2°t o 45.0°and the different inlet Reynolds number range for each valve angle(θ) is given on the table 1.

The inlet air was assumed to be at the STP condition. The simulation code, FLOW3D[4, 5], was incorporated for this study. It solves the full Navier-Stokes equation, the continuity equation and the energy equation in either Cartesian or cylindrical co-ordinates. Here, the body-fitted co-ordinate system was used for grid generation. The standard(k-ε) turbulent model was used and the no-slip condition near the solid boundary is modelled by the logarithmic law. Conjugate gradient techniques for pressure corrections in the transport equations were incorporated and the velocity/pressure coupling was solved via 'SIMPLE' algorithm[7]. From the previous study[6], it was revealed that the velocity boundary condition is much more economical than the pressure boundary condition. Therefore the former was adopted for this study.

3. RESULTS AND DISCUSSION

Numerical experiment was performed to find the compressible effects of the airflow field around a butterfly valve with different inlet Reynolds numbers at the valve angle(θ). The convergence test criteria(i.e. residual mass) for the calculation was set at 10^{-4}. From a previous study [6], it was indicated that the simulation outputs with the smaller convergence criterial like 10^{-6} or 10^{-7} showed very little differences in the calculation results compared with the results of using 10^{-4}; moreover, it should be noted that an increase of the convergence test criteria will result in the reduction of the computer CPU time.

A comparison between compressible and incompressible computed average horizontal velocity (U) at the 4D(D : duct diameter) back from the valve centre is given in fig.4. As the valve angle (θ) increases from 22.2°to 45.0°, the differences between compressible and incompressible simulation results decrease at a given inlet Reynolds number(Re). That is, the flow field is less interfered with as the valve has a bigger opening angle. The difference is already over 10% as the Reynolds reaches to 35,000 at 22.2° of valve angle; however, it is still less than 10% at 130,000 Reynolds number as the valve angle is 45° as shown in fig.4. This means that the compressible effect must be considered for the simulation at a higher Reynolds number over 35,000 at 22.2°of valve angle. Fig.10 shows the variation of the Reynolds number with respect to the change of valve angle on each difference(%) of the average horizontal velocity between compressible and incompressible flow. The valve loss coefficient(Kv) has the same trend in variation as the average horizontal velocity, shown in fig.5. The difference of the valve loss coefficient(Kv) between compressible and incompressible flow has been over 10% as the Reynolds number is 56,000 at 45°o f valve angle.

Compared with fig.5, the incline of the curves of fig.4 is much steeper. This means that the average horizontal velocity is more sensitive to the compressible effect in the flow field than the valve loss coefficient(Kv). When the valve angle is 22.2°, the loss coefficient difference has already reached on 10% at 19,000 of the inlet Reynolds number ; however, in the case of the average horizontal velocity, the Reynolds number is up to about 30,000 until it reaches to 10% difference(fig.4, 5).

The turbulent kinetic energy(T.K.E) at the back of the valve has very important meaning for an understanding of the quality of the flow field. As shown in fig.6, the compressible effect is not serious until the Reynolds number reaches 105,000 at 45° ; however, the compressible ef-

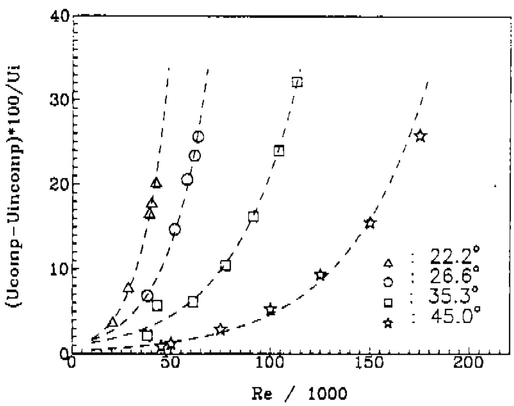

Fig.4 Comparison of the computed compressible and Incompressible averaged velocity at the 4D back of the valve on each valve angle, (22.2°, 26.6°, 35.5°, 45°)

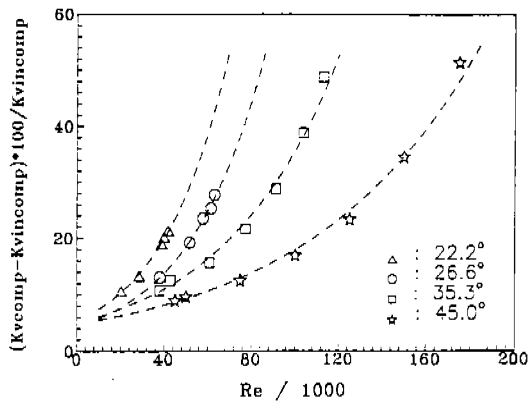

Fig.5 Comparison of the computed compressible and Incompressible valve loss coefficient on each valve angle, (22.2°, 26.6°, 35.3°, 45°)

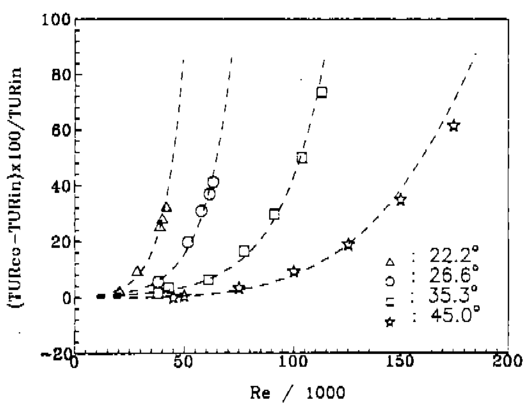

Fig.6 Comparison of the computed compressible and Incompressible turbulent kinetic energy(TKE) at the 4D back of the valve on each valve angle, (22.2°, 26.6°, 35.3°, 45°)

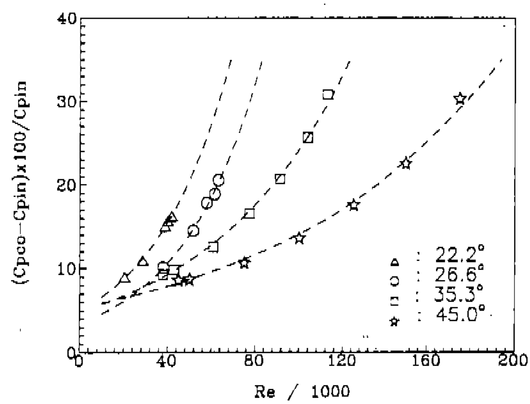

Fig.7 Comparison of the computed compressible and Incompressible pressure coefficient(Cp) at the lower valve tip on each valve angle,(22.2°, 26.6°, 36.3°, 45°)

fect appears earlier than this value at the lower valve angle. In the case of 22.2°, the compressible effect has already appeared at 31,000.

The pressure coefficient distribution in the flow field has an important meaning for an analysis of the characteristics of the flow field. This value is defined as ;

$$C_p = \frac{2(P_2 - P_{ref})}{\rho U_i^2} \qquad (1)$$

where P_2 is the static pressure on the lower surface of the duct at 4D back from the centre of the valve and P_{ref} is the local static pressure at the lower inlet of the duct. The C_p difference is not serious until the Reynolds number is 64,000 at 45°, but the compressible effect has already appeared seriously in the flow field as Re is 25, 000 at 22.2°as shown in fig.7.

When the inlet mass flowrate is very high, the valve tip velocity must be very high with a smaller valve angle and at times reaches supersonic speed. In that case, the flow around the valve will be choked and it must have a critical effect on the well-mixed gas generation for perfect combustion in cylinder. Fig.8 shows the variation of Mach number (M) with the change of the inlet Reynolds number on each valve angle(θ). Mach number varies linearly with the inlet velocity. Supersonic flow or choked flow for each valve angle and inlet Reynolds number can be anticipated from the fig.8.

Fig.9 shows the variation of computer CPU time difference between compressible and incompressible flow calculation with respect to change of valve angle. As mentioned earlier, the duct which has smaller valve angle with higher mass flowrate did have very strong compressibility around the valve tip.

It also should be noted that the compressible flow calculation requires longer computer CPU time than the incompressible flow one.

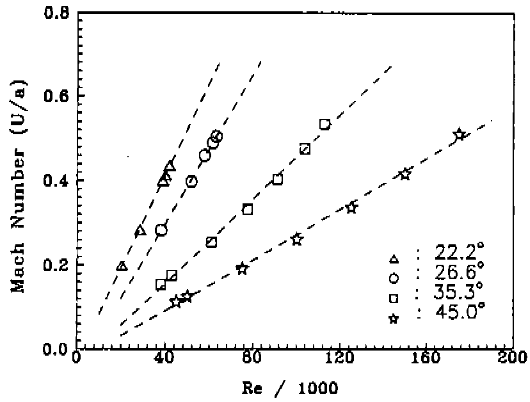

Fig.8 Variation of compressible Mach number (M) at the lower valve tip with respect to the change of Reynolds number on each valve angle(22.2°, 26.6°, 35.3°, 45°)

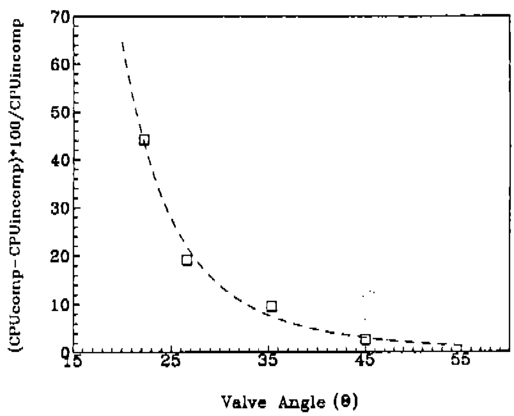

Fig.9 Comparison of the averaged CPU time between compressible and incompressible calculation on each valve angle, (22.2°, 26.6°, 35.3°, 45°)

Fig.10, 11, 12 and 13 show the variation of the difference ratios of the average horizontal velocity(U), the valve loss coefficient(Kv), the turbulent kinetic energy(TKE) and the pressure coefficient(C_p) which are the main characteristic components of the flow field with respect to change of a valve angle(θ) and an inlet Reynolds number (Re). These are very important results for the understanding of the flow field characteristics, especially for compressibility. The right lower side of each 10% line is a very stable region where compressible effect in airflow field can be ignored.

4. CONCLUSION

The base airflow around a butterfly valve has been simulated in order to examine the effects of compressibility of the flow field. Air mass flow-rate(m) and valve opeing angle(θ) have been chosen as primarily independent variables for this numerical experiment. The compressible effect in the flow field has critical effect on the generation of well-mixture gas for combustion in a S.I. engine. When the valve is closed, the flow field at the tip of the valve is very complicated and at times the flow is choked because of compressible effect. It is a serious obstacle on the well-mixture gas generation. As this condition is approached, the energy equation term for compressible flow simulation must be used to predict the flow field. For a larger valve angle and/or smaller mass flow rate, it is unlikely that the compressibility would play such an important role in the predictions. In this case, it would be more economical to use the incompressible simulation method to save computer CPU time. For example, if the valve angle is 20°using the incompressible flow calculation option, a saving of over 40% of CPU time is achievable(fig.9).

Fig.10, 11, 12 and 13 show an exact boundaries of the incompressible flow simulation region against that of the compressible flow. The lower right side of each line of 10% difference is stable zone where the compressible effect is not so critical in the flow field. These outputs have very important meaninig on decision of the existence of compressible effect on the flow field and to have economical simulation.

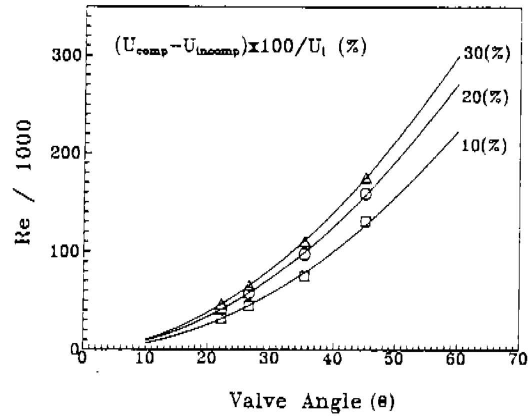

Fig.10 Variation in the difference ratios of the averaged horizontal velocity with respect to the change of valve angle and inlet Reynolds number

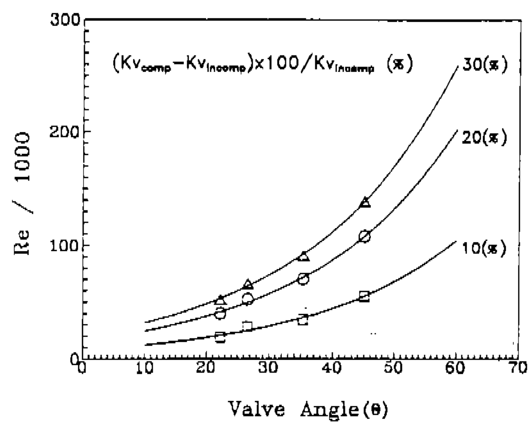

Fig.11 Variation in the difference ratios of the valve loss coefficient with respect to the change of valve angle and inlet Reynolds number

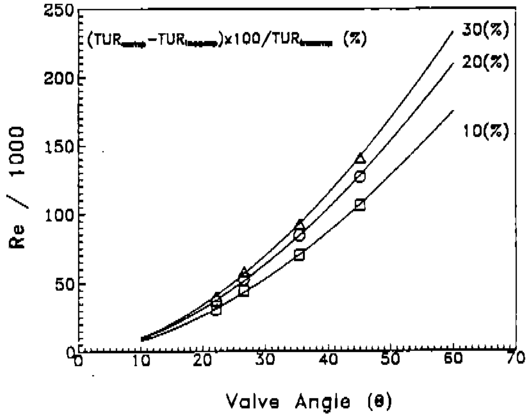

Fig.12 Variation in the difference ratios of the turbulent kinetic energy with respect to the change of valve angle and inlet Reynolds number

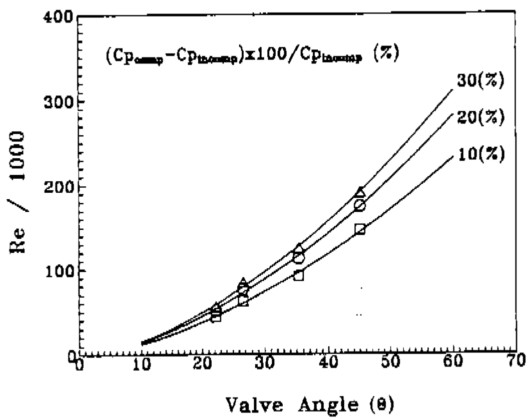

Fig.13 Variation in the difference ratios of the pressure coefficient with respect to the change of valve angie and inlet Reynolds number

5. REFERENCES

1. Harrington, D.L. and Bolt, J.A. "*Aanalysis and Digital Simulation of Carburetor Metering,*" SAE 700082, SAE Trans. Vol79, 1970.
2. Behnia, M., Jones, I.P. and Milton, B.E., "*Simulation of Flow Past an Engine Intake Manifold Throttle V alve*", 4th Asian Cong. on Fluid Mechanics, Hong Kong, August 1989.
3. Morris, M.J. and Dutton, J.D., "*Compressible Flowfield Characteristics of Butterfly Valves*", Trans. ASME, J. Heat Transfer, Vol.111, p. 400~407, 1989.
4. Harwell-FLOW3D User Manual(Release 2), Harwell Laboratory, July 1989.
5. Numerical Grid Generation Package, Harwell Laboratory, January 1989.
6. Kim, C.H., Choi, J.M., Behnia M. and Milton, B.E., "*Computation of Flow Field Around A Butterfly Valve,*" Trans. KSAE, J. Automotive Engr., Vol.14, No.1, 1992.
7. Patankar, S.V., "*Numerical Heat Transfer and Fluid Flow,*" Hemisphere Pub. Co., p.126~131, 1980.