Computation of the Flow Field around a Butterfly Valve

Butterfly밸브주위의 유동장 해석

김 철 호,* 최 정 명,** M. Behnia,*** B. E. Milton***
C. H. Kim, J. M. Choi,

초 록

대도시의 자동차 배기가스에 의한 대기오염문제는 날로 심화되어 가고 있으며, 그 규제 역시 매우 엄격해지고 있다.

배기가스의 문제는 연소실내에서의 고효율연소에 의해 개선될 수 있으며, 이를 위해서는 혼합가스 생성장치에서의 잘 미립화된 혼합가스의 생성에 관한 미시적 연구가 필수적이다.

본 연구에서는 범용 유체 유동해석, 프로그램을 이용하여 Butterfly밸브 주위에서의 공기의 흐름현상을 예측하였으며, 밸브의 손실계수(Kv)를 실험치와 비교하여 보았다. 또한 압축성과 비압축성 유체 유동해석의 비교, Residual Mass (R. M)의 최적치 정의에 의한 CPU 시간의 최소화, Numerical Grid Size의 해석결과에 미치는 영향 등에 관해 알아보았다.

1. INTRODUCTION

Flow around a throttle valve or damper in a duct is complex and multi-dimensional. Many examples exist in engineering practice but one of the most complex cases is that taking place in the intake manifold of a spark ignition (S.I.) engine [1,2] where the flow may be both two-phase and multi-component. Additionally, the fuel transport takes place in the form of an airborne flow and a liquid wall film [3]. This is known to produce operational problems in engines particularly during transient operating conditions [4,5]. In detail, the fuel film is instrumental in producing an air/fuel ratio imbalance during transient operation because its velocity is several orders of magnitude slower than the airborne fuel with a consequent lag before obtaining the new steady state conditions. Hence, an understanding of film formation and movement is essential in manifold design.

The experimental results of Behnia and Milton [6] have shown that the throttle plate is itself an important factor in providing a highly turbulent, recirculating region that can

* 정회원, 호주 New South Wales 대학교 기계공학과 박사과정
** 금성중앙연구소 제 4 연구실
*** University of New South Wales

direct the liquid fuel flow towards the wall. That is, it serves as a desposition region for droplets which can then be redirected as a steam onto the wall. For an understanding of the phenomena leading to film formation, it is essential that the base air flow behaviour around the throttle plate is known. To this end, a numerical study of the flow in this important region has been undertaken.

Once solutions of the base air flow are obtained for a number of different parameters, the model developed by Chen et al [7], based on a Monte Carlo technique, can be used to determine where the fuel droplets leading to the film formation are deposited. Preliminary results indicate that reasonable agreement with the experimental results of Behnia and Milton [6] occurs.

In the present study, a general purpose, multi-dimensiona, finite difference CFD code has been used to obtain solutions to the problem of two-dimensional, turbulent air flow in a duct around a throttle valve. A number of different flow conditions and throttle valve geometries have been simulated and the importance of flow compressibility has also been investigated. Several numerical grid sizes have been tested to determine an optimum, cost effective mesh for accurate simulation. The computed results, when compared with the experiments of Eom [8], show that the numerical predictions agree favourably with the experiments, indicating that the computer code can be successfully employed for simulation of the complex base air flow around a throttle valve.

2. NUMERICAL MODEL

The computer program, FLOW3D, used here was developed at the Atomic Energy Research Establishment, Harwell and is a general purpose, multi-dimensional, finite difference code. Details of the code structure may be found in the FLOW3D Manual [9]. This package solves the full Navier-Stokes, energy and continuity equations in eithe Cartesian or cylindrical co-ordinates. It may be used for either laminar or turbulent flow, the latter incorporating a ($k-\varepsilon$) model as a standard option, but it should be noted that any desired turbulence model may be specified instead. The turbulent no-slip condition near solid boundaries is modelled by the logarithmic law. There is an option for solving the compressible flow equations instead of the incompressible ones. The finite difference approximations are based on a non-staggered grid. Time differencing is either fully implicit backward or central whilst advection terms are hybrid differenced. Conjugate gradient techniques for pressure corrections in the transport equations are incorporated and the velocity/pressure coupling is solved via standard algorithms, such as 'SIMPLE'. The code uses body fitted co-ordinates in conjunction with non-orthogonal grids allowing irregular geometries to be handled without much difficulty. Boundary conditions may be designated either in terms of pressure or velocity, but the velocity boundary condition has been selected for the computations presented here. Through preliminary numerical experiments at the beginning of the research, it was found that there were no differences in the calculated results between the pressure and the velocity boundary condition. However, much less computer CPU time was required for the calculations using the latter. The code incorporates a pre-processing routine for the generation of the spatial grid [10]. Once the geometrical boundaries and the number of the mesh points are specified in the mesh generation routine, a grid is generated and stored for computational purposes. For improved resolution in the regions of interest, the computer generated mesh can be refined either manually or by using the adaptive grid routine.

3. COMPUTER SIMULATION

The geometry selected for the numerical modelling of the flow around the butterfly valve closely follows the experiments of Eom [8]. It should be noted that these experiments were performed in a circular duct, but the present computations are two-dimensional. Figure 1 shows the geometry of the experimental rig as well as a typical computational grid. The valve angle (θ) is measured from the horizontal and, for each angle, a separate mesh was generated by the grid generation routine. To match the geometry of the experimental valve, the same valve blockage ratio (R), defined as the area of the throttle plate divided by the duct cross-sectional area, was used. The value of R used in Eom's experiments [8] was 0.976. In the simulations, since it was necessary to have a mesh point located at each tip of the plate, this value could not be exactly matched for all angles. However, care was taken to ensure that the simulation blockage ratio did not differ from experiment by more than 0.1%. Ten different valve angles between 30° and 90° were considered. The inlet velocity was varied between 5.4 m/s and 45.4 m/s, corresponding to a Reynolds number (Re) range between 38,000 and 316,000. The inlet air was assumed to be at the US standard atmospheric condition. For large values of the valve angle (θ), the opening area of the valve is smaller leading to higher velocities in this region. At higher velocities, the effect of fluid compressibility becomes more important and it is necessary to take this

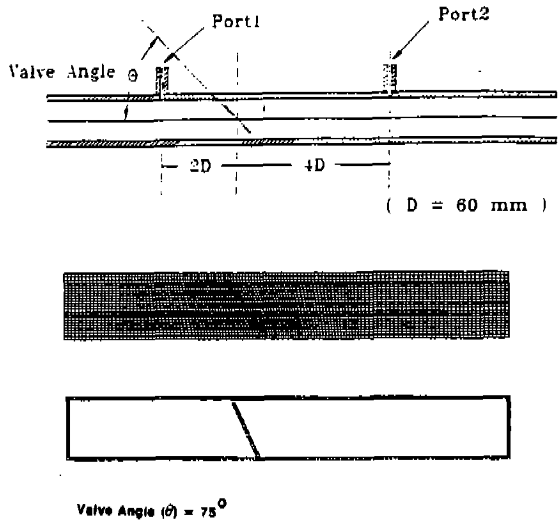

Fig. 1 Eom[8] experimental rig and a sample computational grid

into account in the numerical simulation [11]. Of course, it should be noted that the compressible flow computations generally require more computer CPU time. To examine the effects of the compressibility of air on the computed flow field as well as the required CPU time, numerical experiments were performed using both compressible and incompressible flow options. For these runs, a valve angle of 30° and the above-mentioned range of Reynolds numbers were chosen for the simulation conditions.

To determine the dependence of the computed flow field on the computational grid spacing, three different grids were tested. The number of grid points in the vertical direction was fixed at 32 with a non-uniform spacing. However, different grid spacings of 72, 84 and 96 were used in the horizontal direction. The grids were orthogonal at the entrance and exit regions while elsewhere they were deformed to provide a suitable fit for the throttle valve angle, as shown on Figure 1. The iterative solution procedure was stopped when the value of the convergence test criteria reduced to that specified by the user. This quantity represents the ratio of the integrated residuals of the continuity equation at all mesh points normalised with respect to the total mass flowrate. It is called 'residual mass' and it should be noted that the smaller its value, the higher the accuracy to which the continuity equation is satisfied. However, greater accuracy is at the expense of CPU time. A number of different residual mass quantities were examined and the accuracy of the solutions tested.

4. RESULTS

Simulations have been performed on the above geometry with different valve angles (θ) and inlet flow conditions. A number of preliminary runs were carried out to establish an optimum convergence criteria (i.e. residual mass). Different features of the converged solutions were checked for a residual mass range from 10^{-6} to 0.1. It was found that in most cases the solution field values changed very little for a residual mass of less than 10^{-3} It should be noted that an increase of the convergence test criteria from 10^{-6} to 0.1 will result in a 60% reduction in computer CPU time. Therefore as a compromise between accuracy and computational cost, a value of 10^{-4} was chosen for most of the simulations.

Sample results for different grid sizes were examined. It was noted that most flow field values varied by less than 0.5% as the horizontal mesh numbers were changed from 72 to 96. Therefore, most simulations were carried out using the more economical 72 x 32 grid.

Computed results for an inlet velocity of 5.4 m/s corresponding to a Reynolds number of 38,000 are shown in Figure 2. These are for different valve angles and assume that the flow is incompressible. The velocity vector plots clearly show the recirculation zone behind the throttle valve. This reverse flow forms two vortices which are symmetrical when the valve position is vertical. For a fully opened valve, very little disturbance exists. As the valve angle is increased from the fully open position, the length of the recirculation region grows until the valve angle reaches 85° after which a further increase of the angle to 90° causes its length to reduce. A comparison of different recirculation lengths for various angles is given in Table 1. Eom [8] presented pressure measurements at two positions, port 1 and port 2 as shown in Figure 1, in terms of a valve loss coefficient (Kv) defined as:

$$K_v = 2(P_1 - P_2)/(\rho U_i^2)$$

where ρ is the air desntiy and U_i is the inlet velocity to the duct. A comparison between

Table 1 Variation of the Recirculation zone length on each butterfly valve angle (θ)

VALVE ANGLE (θ)	30°	50	65	70	75	80	85	90
RECIRC. ZONE LENGTH	1.63D	2.25D	3.06D	3.12D	3.16D	3.18D	3.52D	3.01D

(D: duct diameter)

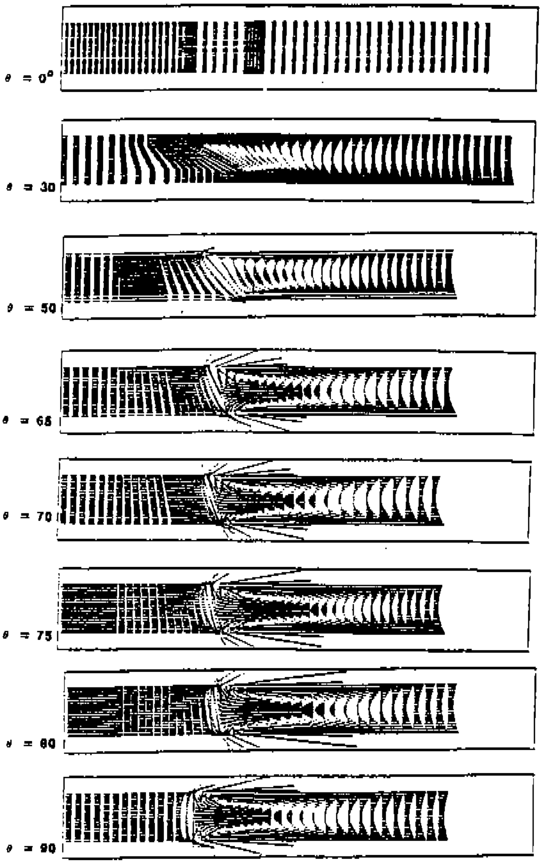

Fig. 2 Velocity vector fields for incompressible flow simulation, Re = 38,000

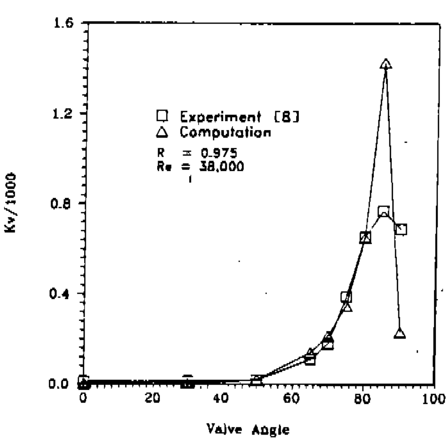

Fig. 3 Valve Loss Coefficient

the present computed valve loss coefficient and Eom's experimental results for different valve angles with a fixed inlet velocity of 5.4 m/s (Re = 38,000) is given in Figure 3. The agreement for valve angles of less than 80° is excellent. However, beyond this value the computed results first become higher than the measured one and then drop below it. The difference is possibly due to the fact that the computations are two-dimensional while the experiment was three-dimensional. As can be seen in Figure 2, the computed recirculation zone increases until the angle reaches 85°, after which it decreases. If it is assumed that the zone increases until the angle reaches 90°, the pressure will drop continuously to 90° and induce an increase of valve loss coefficient (K_V). That is, the valve loss coefficient (K_V) would then increase until the angle reaches 90°. Eom, in his conclusions, notes that there was some experimental error at high valve angles, which may explain the discrepancy between the experimental and computed results. The good agreement for valve angles of up to 80° is encouraging and strongly justifies the economical two-dimensional computations instead of costly three-dimensional ones.

As mentioned earlier, at higher mass flowrates (i.e. higher inlet velocity) the velocity through the valve opening will become very large. In fact, when the valve is nearly closed, the velocity through the valve will be high even at the lower mass flowrates. In such circumstances, fluid compressibility is important and needs to be properly accounted for in the computations. For a valve angle of 30°, compressible flow computations were performed for different mass flowrates. A comparison between compressible and incompressible computed average horizontal velocity $\overline{U}$ at port 2 is given in Figure 4. It can be

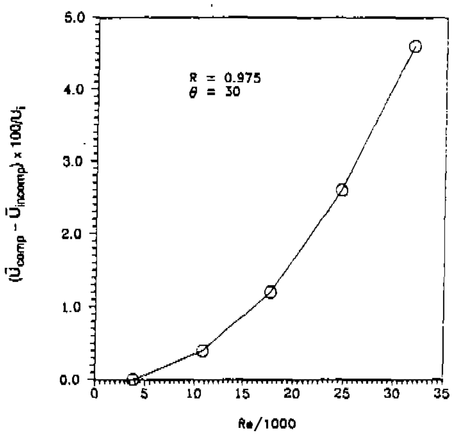

Fig. 4 Comparison of the computed compressible and incompressible averaged velocity at port 2

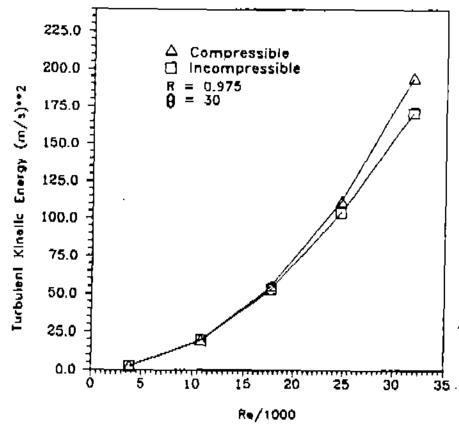

Fig. 6 Comparison of the computed averaged turbulent kinetic energy at port 2

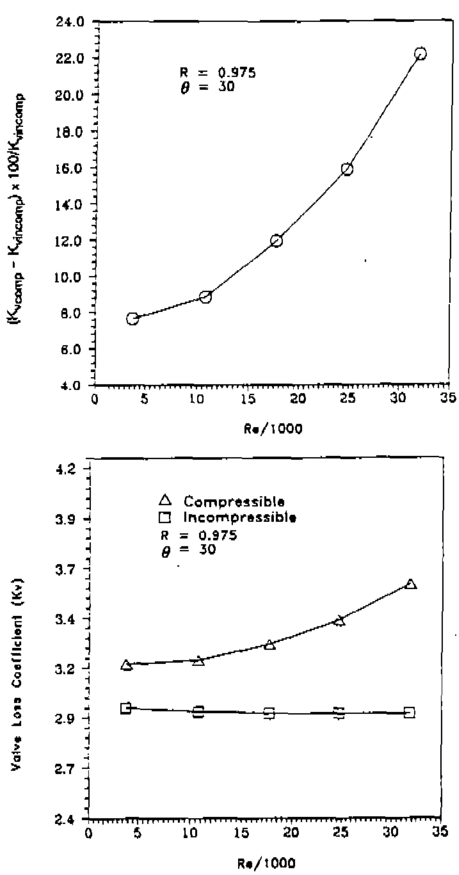

Fig. 5 Comparison of the computed compressible and incompressible valve loss coefficient

seen that the difference increases with the mass flowrate. For this valve angle the difference is not very significant and at Re = 316,000 the compressible average velocity is only 4.6% higher than the incompressible one. However, compressibility has a more significant effect on the valve loss coefficient as can be seen on Figure 5. The effect of compressibility on the computed turbulent kinetic energy at port 2 for a valve angle of 30° is shown in Figure 6. The comparison indicates that, as expected from the velocity differences, compressible computations yield a higher value of turbulent kinetic energy. At Re = 3.16 x 10^5 the difference is about 12%.

5. CONCLUSIONS

The base air flow around a butterfly valve has been computed using a finite difference computer code. Although the program is capable of three-dimensional computations, in this study a two-dimensional geometry was considered. Simulations have been performed for different air mass flowrates at several valve opening angles. The computations clearly show that a recirculation region exists the length of which varies with the valve angle. The flow patterns obtained computationally are qualitatively similar to the experiments.

Valve loss coefficients were computed for geometries identical to those used by Eom [8]. In general, for valve angles of less than 80° from the horizontal, the agreement was excellent. At higher valve angles the agreement deteriorated. This is due to the fact that computations were two-dimensional in contrast to the three-dimensional experiments as well as some experimental errors at larger angles. The effects of flow compressibility in the computations were examined. Even at moderate valve opening angles, for the higher mass flowrates, it was noted that compressibility must be properly accounted for in order to obtain accurate valve loss coefficients. For instance, at a valve angle of 30° and Re = 3.16×10^5, ignoring compressibility in the computations underpredicts the valve loss coefficient by 22%.

6. REFERENCES

1. Chapman, M., *"Two-Dimensional Numerical Simulation of Inlet Manifold Flow in a Four Cylinder I.C. Engine,"* SAE Paper 790244, 1979.
2. Benson, R.S., Baruah, P.C. and Sierens, I.R., *"Steady and Non-Steady Flow in a Simple Carburettor"*, Proc. IMechE, Vol. 188, No. 53/74, pp. 5370548, 1974.
3. Kay, I.W., *"Manifold Fuel Film Effects in an S.I. Engine"*, SAE Paper 780944, 1978.
4. Milton, B.E., *"Flows in i.C. Engine Manifolds during Transient Operation"*, 9th Australasian Fluid Mechanics Conf., Auckland, December 1986.
5. Behnia, M., Jones, I.P. and Milton, B.E., *"Simulation of Flow Past an Engine Intake Manifold Throttle Valve"*, 4th Asian Cong. on Fluid Mechanics, Hong Kong, August 1989.
6. Behnia, M. and Milton, B.E., *"The Behaviour of Fuel Films on the Walls of Engine Manifolds and Ports"*, Proc. MECH'88 Cong., IE Aust., Brisbane, March 1988.
7. Chen, P.Y.P., Milton, B.E. and Behnia, M., *"A Numerical Examination of Fuel Film Formation in Manifolds"*, 4th Int. Symp. on Transport Phenomena, Sydney, July 1991.
8. Eom, K., *"Performance of Butterfly Valve as a Flow Controller"*, Trans. ASME, J. Fluids Engr., Vol. 100, pp. 16-19, March 1988.
9. Harwell-FLOW3D User Manual (Release 2), Harwell Laboratory, July 1989.
10. Numerical Grid Generation Package, Harwell Laboratory, January 1989.
11. Morris, M.j. and Dutton, J.D., *"Compressible Flowfield Characteristics of Butterfly Valves"*, Trans. ASME J. Heat Transfer, Vol. 111, pp. 400-407, 1989.