Proceedings of the
International Conference on Mechanical Engineering 2005
(ICME2005) 28- 30 December 2005, Dhaka, Bangladesh

ICME05-FL-08

NUMERICAL INVESTIGATION OF FLUID FLOW CHARACTERISTICS IN A PCV VALVE

Md. Tajul Islam[1], Jong Hoon Lee[1], M. M. A. Sarker[2] and Yeon Won Lee[3]

[1]School of Mechanical Engg., Graduate School of Pukyong Natioanal University, Busan, Korea.
[2] Dept of Mathematics, BUET, Dhaka-1000, Bangladesh.
[3]School of Mechanical Engg., Pukyong National University, Busan, Korea.

ABSTRACT

Blow by gas emitted by automobiles causes air pollution and reduces engine efficiency. Automobile engines have the Positive Crankcase Ventilation (PCV) system to prevent air pollution. In this system, a PCV valve is used and it is the most important component to control the flow rate of blow by gas. But designing an effective PCV valve is not easy. It needs information of fluid flow characteristics through a PCV valve. But extracting flow information is very difficult because of interaction between fluid and solid motions. In this study, we investigated fluid flow characteristics using re-meshing method of CFD technique to simulate spool behavior. The spool motion is periodical and oscillatory with time. The objective of this study is to provide numerical results of spool behavior and flow characteristics in a PCV valve which may play an important role for designing an appropriate PCV valve.

Keywords: PCV valve, Blow by gas, Spool.

1. INTRODUCTION

Air pollution is becoming a serious problem all over the world in recent times. Taking various measures to prevent air pollution is a demand of time. Nowadays, the uses of automobiles are being increased rapidly and the necessity of using automobiles is unavoidable. For this reason, automobile companies are developing various techniques to increase its efficiency and to prevent air pollution problems. Exhaust gas of an automobile mainly consists of HC (Hydrocarbon) and CO (Carbon monoxide). Hydrocarbon is generated about 20~35% by the crankcase in the total system [1]. During normal compression stroke, a small amount of gases called blow by gas in the combustion chamber escapes through a small gap between the cylinder wall and the piston ring into the crankcase room. Approximately 70% of these blow by gases are unburned that can dilute and contaminate the engine oil cause corrosion to critical parts and contribute to sludge build up [7]. As a result engine efficiency is reduced. At higher engine speed, blow by gases increase crankcase pressure that can cause oil leakage from sealed engine surface. Consequently, blow by gases need to be eliminated to prevent air pollution problem and to increase engine efficiency.

Emission system of blow by gas is classified into two types as shown in figure 1. Open type emits blow by gas to the atmosphere through the draft tube. This type is not used nowadays because of air pollution. Closed type eliminates blow by gas by re-burning it in the combustion chamber through feedback loop. This type is called the Positive Crankcase Ventilation system (PCV system) [2].

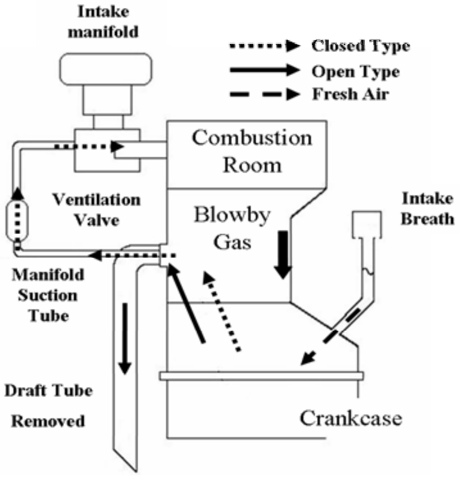


Fig 1. Crankcase ventilation system

There are lots of researches regarding automotive engines but much attention has not yet been reported in the literature concerning spool dynamics. The velocity of a spool is very high. So the experimental study of spool dynamics and flow characteristic in a PCV is very sparse. We conducted numerical study on the flow behavior in the PCV valve. The purpose of this study is to get information about flow behavior in a PCV valve which can be used to design an efficient PCV valve.

2. NUMERICAL SIMULATION

A PCV valve consists of a main body, spool, return spring and cushion spring. When the Blow by gas passes through the PCV valve, it creates pressure on the spool. So the spool **is** displaced and the flow area is reduced. But the return spring creates an opposite force on the spool and tries to return it to its original position. Thus the spool is moved by the force balance between the flow force of the gas and the elastic force of the return spring. As a result, the spool controls the flow rate of blow by gas by changing flow area. Because of the geometric similarity of each model, we have arbitrarily selected a commercial model of a PCV valve as shown in figure 2. A commercial CFD software FLUENT [8] has been used and MDM (Moving Dynamic Mesh) method has been applied to consider the spool dynamic behavior. For using this method, we generated tetra type mesh as shown in Figure 3. In this study, we assumed axisymmetry.

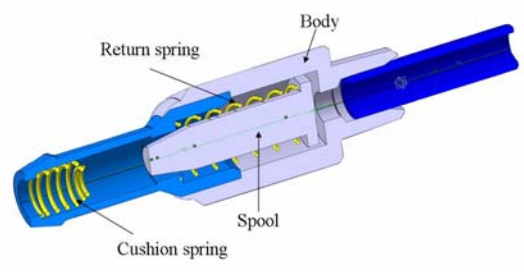


Fig 2. Sectional view of a PCV valve

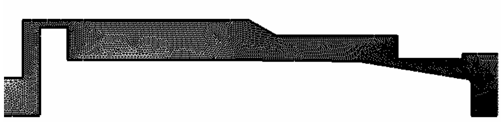


Fig 3. Mesh structure

Table 1: Boundary condition

Boundary Name	Momentum	Energy
Inlet	0 mmHg (By gauge pressure)	293 K
Outlet	-50 mmHg -100 mmHg -200 mmHg (By gauge pressure)	Nuemann condition
Wall	No slip	Adiabatic

Table 1 shows boundary conditions of this study. At the inlet, we fixed the atmospheric pressure. At the outlet, we selected three pressure boundary conditions -50, -100 and -200 mmHg.

2.1 Spool Behavior

This is occurred by force balance between fluid force and elastic force of spring. The force balance is shown in figure 4.

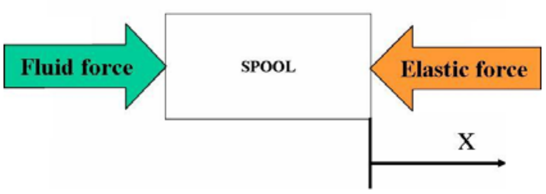


Fig 4. Free body diagram of a spool

The above free body diagram can be expressed mathematically. We formulized this relationship as follows; [3]

$$\frac{dv}{dt} = \frac{1}{m}\left(F - kx\right) \tag{1}$$

$$\frac{dx}{dt} = v \tag{2}$$

where, v is the spool velocity, m is the spool mass, $F(=PA)$ is the fluid force, P is the pressure, A is the area normal to moving direction and k is the elastic modulus and x is the spool displacement.

Eq. (1) and (2) can be expressed using Euler's explicit method as follows [4];

$$\Delta v = \frac{1}{m}\left(F - kx\right)\cdot \Delta t \tag{3}$$

$$v_{i+1} = v_i + \Delta v \tag{4}$$

$$x_{i+1} = x_i + \Delta v \cdot \Delta t \tag{5}$$

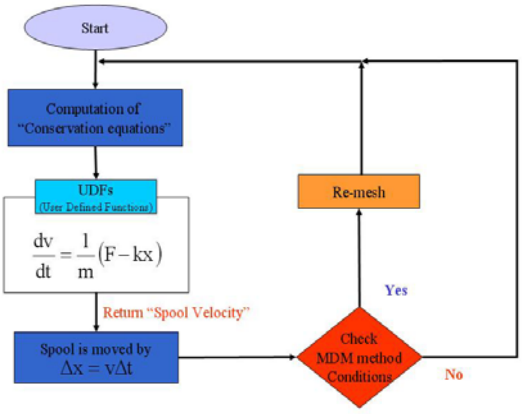


Fig 5. Flow chart of numerical computation

where, subscript i is current time step, subscript $i+1$ is next time step and Δt is time step.

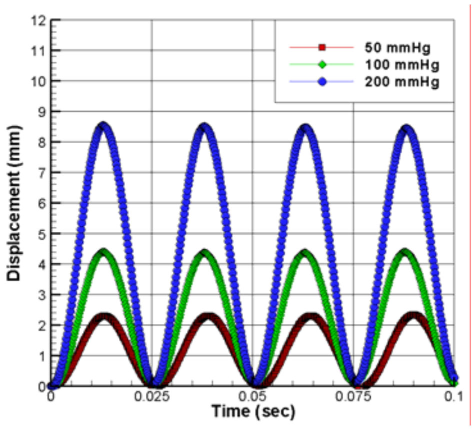
Figure 5 shows the flow chart of numerical computation. On current time step, governing equations of fluid flow is solved by the commercial solver. Then, we know the pressure force acting on the spool. In UDFs (User Defined Functions), F is equal to the solved pressure force. Spool velocity is solved by UDFs. This value is returned to the commercial solver. Then, the solver moves the spool as Δx . This work may influences on the mesh quality such as maximum or minimum cell volume and cell skewness. The commercial solver checks the quality and if the quality is not satisfactory then the computational domain is re-meshed.

3. RESULT AND DISCUSSION

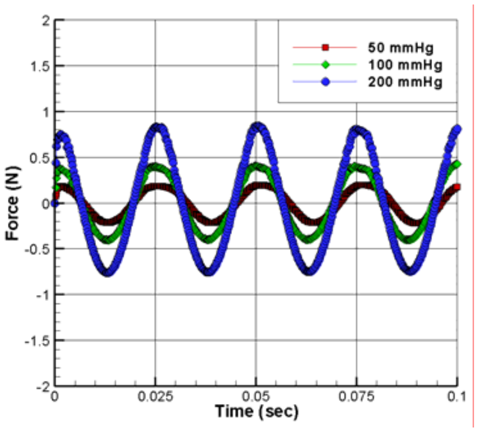
We simulated PCV valve to understand dynamic flow characteristics with spool behavior from 0 second to 0.1 second according to the differential pressures.

3.1. Spool Behavior

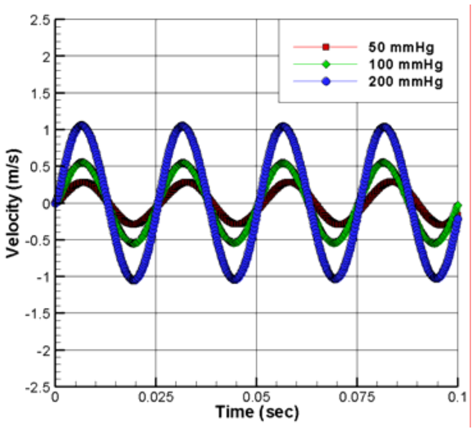
Figure 6 shows the change of various dynamic spool characteristics according to various differential pressures. Figure 6 (a) shows the oscillatory spool movement.



6 (a). Spool displacement



6 (b). Spool velocity



6 (c). Force

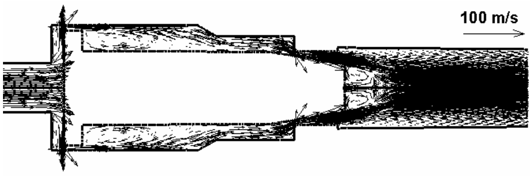
Fig 6. Spool dynamic characteristics w.r.t various differential pressures

From figure 6 (a), it can be claimed that the frequency of spool motion is similar for each case. But, the frequency of differential pressure 50 mmHg is getting faster in comparison to other cases. Also their magnitudes increased with the increase of the differential pressure. Figure 6 (b) and (c) show the spool velocity and the surface force of spool, respectively. These are similar to the characteristics of spool displacement.

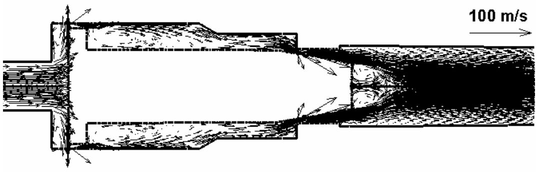
3.2. Flow Characteristic

The purpose of this study is to simulate and to understand flow characteristics according to changing spool motion. So, we indicated the change of flow in one cycle from 0.075 second to 0.1 second in figure 6(a).

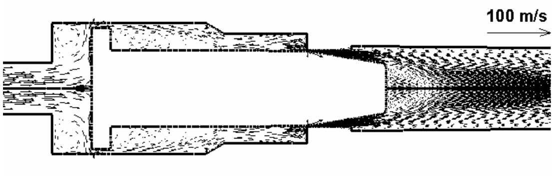
Figure 7 shows flow characteristics at differential pressure 50 mmHg during the last cycle of figure 6(a). According to passing time, spool is moved forward and backward by force balance between fluid force and elastic force of the spring. At 0.075 second, the maximum velocity is 93.5 m/s at orifice region. And at 0.0812, 0.0878, 0.0938, 0.1 second maximum velocities are 99.3, 122, 122 and 93.1 m/s at the orifice region respectively.



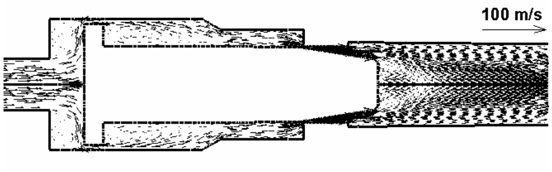
7 (a). 0.07500 sec



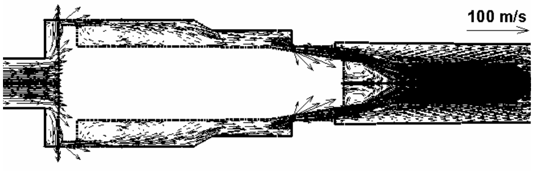
7 (b). 0.0812 sec



7 (c).0.0878 sec



7 (d). 0.0938 sec

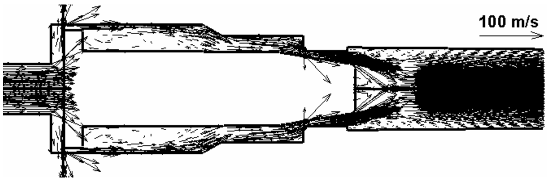


7 (e). 0.1000 sec

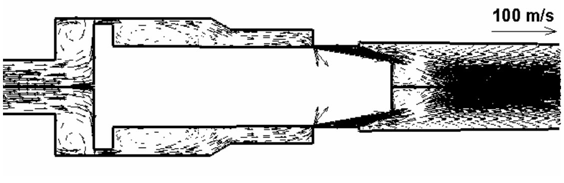
Fig 7. Velocity distribution of differential pressure 50 mmHg

Figure 8 shows velocity distribution of differential pressure 100 mmHg. Maximum velocities are 125, 173, 78.3, 107 and 148 m/s at the orifice region respectively. Although the flow area at the orifice region is decreased by spool behavior, maximum velocity decreases. Also, flow velocity is decreased when the volume of left side of the spool is the maximum.

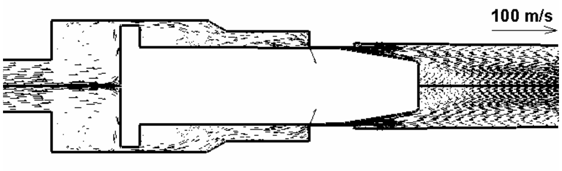
Figure 9 shows flow field at differential pressure 200 mmHg. Maximum velocities are 221, 223, 39, 103 and 233 m/s at 0.075, 0.0812, 0.0878, 0.0938 and 0.1 second respectively. At 0.0878 sec, velocity decreases rapidly. This phenomenon is caused by viscous effect because flow area of the orifice region is very small. In this case the flow velocity is decreased more than the differential pressure 100 mmHg on the left side of spool by larger increase of volume.



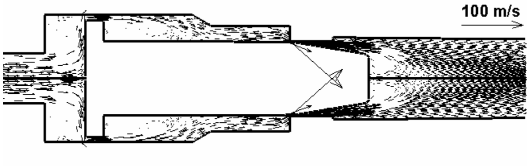
8 (a). 0.07500 sec



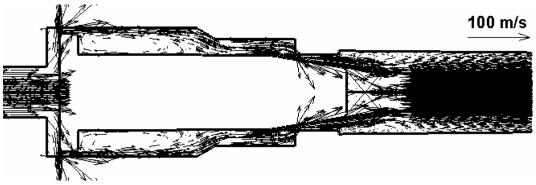
8 (b). 0.0812 sec



8 (c). 0.0878 sec

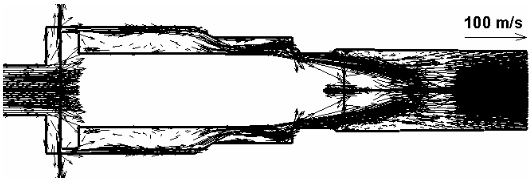


8 (d). 0.0938 sec

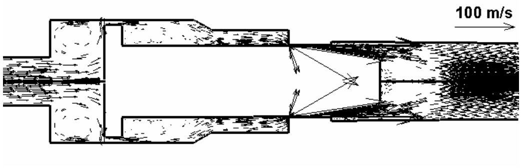


8 (e). 0.1000 sec

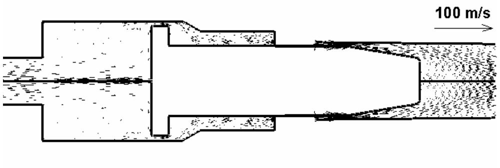
Fig 8. Velocity distribution of differential pressure 100 mmHg at different time span.



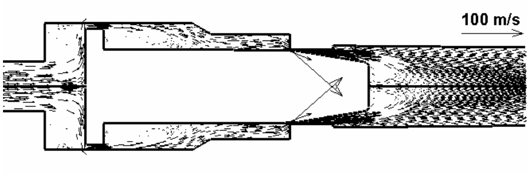
9 (a). 0.07500 sec



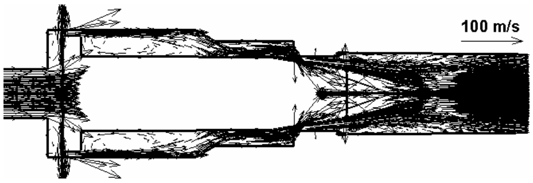
9 (b). 0.0812 sec



9 (c). 0.0878 sec



9 (d). 0.0938 sec



9 (e). 0.1000 sec

Fig 9. Velocity distribution of differential pressure 200 mmHg

Figure 10 shows the change of mass flow rate at the outlet in each differential pressure. In comparison to figure 6 (a), mass flow rate is inverse in proportion to spool displacement. At 0.075 second spool displacements are zero in all differential pressures whereas the mass flow rates indicate maximum values. Also at 0.0878 second, spool displacements have maximum values according to figure 6(a) but mass flow rates indicate minimum values in figure 10.

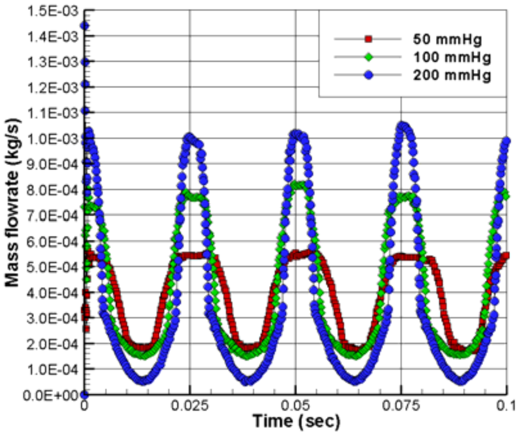


Fig 10. Change of mass flow rate of each differential pressure at outlet

4. CONCLUSIONS

The flow characteristics of PCV valve considering the spool dynamic behavior has been numerically simulated in this study. The findings from this investigation can be summarized as follows:

- Spool dynamic characteristics such as displacement, velocity and force are in simple proportion to the differential pressure.
- With respect to the increase of the differential pressure, the decrease of velocity at the orifice indicates a strong viscous effect.
- When the spool displacement is large viscous effect is increased by the decrease of flow area at the orifice which influences the flow rate at the outlet. In the other words, the flow rate is inversed in proportion to the spool displacement.

5. REFERENCES

1. Donald J. Patterson, *Emission From Combustion Engines and Their Control*, Butterworth-Heinemann, 1972
2. H. C. Dhariwal, “Control of blowby emissions and lubricating oil consumption in I.C. engines”, Energy Convers. Mgmt, Vol.38, No.10-13, pp.1267-1274, 1997
3. Chuen-Yen Chow, *An Introduction to Computational Fluid Mechanics*, pp.17-33, 1981
4. J. H. Lee, Y. H. Choi, Y. W. Lee,“Computational Analysis of Flow Characteristics of a PCV Valve”, Transaction of KSAE, Vol.13, No.4, pp.66-73, 2005
5. R. L. Hoekstra, K. Collier, N. Mulligan and L Chew, “Experimental study of a clean burning vehicle fuel”, Int, J. Hydrogen Energy Vol. 20, No. 9, (1995), pp 737-745.
6. Panorama 2002: “Energy, Oil, Natural Gas, the Automobile, the Environment”, Oil Gas European Magazine, v.28 no.1, 2002, pp.14-17.
7. Toyota motor sales, U.S.A. Inc. Emission sub system-positive crankcase ventilation system”.
8. FLUENT, User's Guide, Fluent Inc.

6. NOMENCLATURE

Symbol	Meaning	Unit
F	Force	(N)
v	velocity	(m/s)
x	displacement	(m)
m	Mass	(kg)