



Performance study of a diaphragm type crankcase pressure control valve

Sh. Alaviyoun¹, M.A. Ehteram^{2*}

¹K.N. Toosi University of Technology and Irankhodro Powertrain Company (IPCO), Tehran, Iran, sh_alaviyoun@ip-co.com

²Qom University of Technology, Qom and Irankhodro Powertrain Company (IPCO), Tehran, Iran, ali_ehteram@aut.ac.ir

*Corresponding Author, Phone Number: +98-253-66411601

ARTICLE INFO

Article history:

Received: 29 September 2013

Accepted: 18 December 2013

Keywords:

Pressure control valve

Engine crankcase pressure

Diaphragm

Spring stiffness

ABSTRACT

Pressure control valve (PCV) is implemented in internal combustion engine to regulate crankcase pressure. In this study, the performance of a diaphragm PCV valve is investigated numerically and experimentally. Firstly, numerical simulation is carried out using commercial code to predict the flow and pressure distribution inside the valve. Furthermore, downstream and upstream pressure of the valve is measured in a laboratory test rig. Finally, displacement and oscillation of the diaphragm are measured by a high speed laser displacement sensor. Comparison of experimental and numerical results show that proposed method is capable to properly predict performance curve of the valve.

1) Introduction

Pressure regulating valve is a part of many fluid system which is implemented to adjust the pressure of system. This type of valve is widely used in crankcase ventilation system of internal combustion engines. When engine operates, a small amount of working gas passes through the cylinder walls and enters the crankcase in the form of blow-by gas flow (leak gas flow). Without ventilation, these gases can create a pressure build-up in the crankcase and contribute to premature contamination of the oil. Therefore, in order to avoid unwanted excess pressures in the crankcase, the blow-by gases must be continuously purged. In closed crankcase ventilation system these harmful gases are released to the combustion chamber via the vacuum of the intake manifold. The excessive vacuum of intake manifold could damage the sealing of the crankcase. In addition, the positive gauge pressure in the crankcase is restricted by emission regulations. Therefore, advanced pressure control is necessary to meet environmental requirements [1].

On some early vehicles, a fixed orifice is used to adjust the pressure of the crankcase. The orifice valve is simple in design and construction, and provides crankcase ventilation based on the size of the orifice and vacuum of the intake manifold. The biggest drawback of these valves is that blow-by production does not always match intake manifold vacuum characteristics. Unlike fixed orifice valves, systems that use a PCV valve could match ventilation flow more accurately. The PCV valve uses intake manifold vacuum (downstream pressure) to regulate the flow of the blow-by gases. This design permits a large blow-by flow through the PCV valve when the intake manifold vacuum is low (full-load operating point), while restricting flow when the intake manifold vacuum is high (idle or part-load).

The principal structure of a pressure control valve is shown in Figure 1. Plunger and diaphragm types of pressure control valve are widely used in vehicular engines. In a plunger valve, a spring moves a taper plunger inside a fixed valve. In the diaphragm type, a leak proof diaphragm separates the upper and lower chambers of valve. The diaphragm is loaded by a pre-stressed spring and a hole connects the atmosphere to the upper chamber. The lower chamber has two connections, one to the intake manifold and the other to the crankcase. While the engine is off, there is no flow and the spring holds the diaphragm up and the valve is open.

Several researchers have been involved in the study and development of PCV valves, which have focused on the computational fluid dynamics (CFD) techniques. For instance, the simulation and experimental verification were carried out on the design of variable flow plunger PCV valve [2].

In another study, the flow characteristics inside a PCV valve were simulated considering the spool (plunger) dynamic behavior [3]. The study could show that the spool motion was periodical with time based on re-meshing method of CFD technique. Some other notable work on PCV valve simulation includes a study on a model to predict the dynamic behavior of valve [4]. The model consisted of valve, crankcase, and intake manifold that are connected. According to CFD analyses for a pressure regulating valve, it was found that within the operating zone, two-dimensional axisymmetric formulation can capture the flow parameters quite well [5]. Flow unsteadiness caused by changes in demand flow has a negative effect on ventilation system. Therefore, an investigation was performed on a specific case of PCV valve vibration, or fluttering [6]. It was found in various tests that flutter occurs to varying degrees of severity. Fast Fourier Transform (FFT) analysis of the pressure traces filters out the frequency levels and can more accurately show whether or not a valve has flutter issues. In order to reduce flutter instabilities, a nonlinear spring system was developed for automotive positive crankcase ventilation valve [7]. This work developed design principles for a spring, that nonlinear stiffness characteristics was specified for a specific precision fluid flow control.

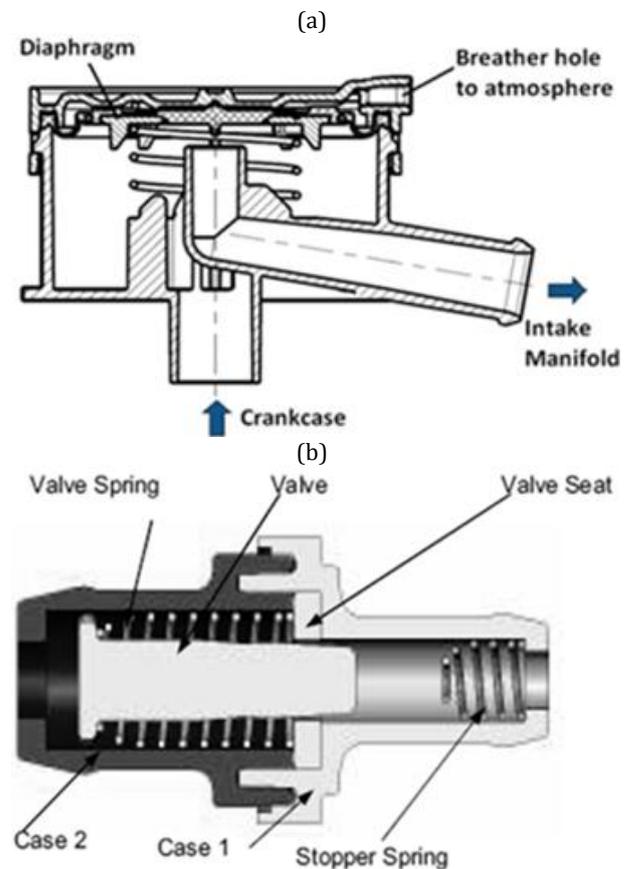


Figure 1: Structure of pressure control valve (PCV); (a) diaphragm type and (b) plunger type

Experimental tests in turbocharged diesel engine showed that the blow-by emissions can be controlled by installing positive crank case ventilation (PCV) system [8]. The cross-section area of the PCV pipe has to be adjusted according to the amount of the blow-by gases and the flow of ventilating air through the PCV valve. During the fabrication and working of the PCV valve, some cracks may occur and may lead to valve failure and an inconsistency of the blow-by gas flow. The cracks effect on the stress and strain variations on the PCV valve was analyzed using the ANSYS code [9]. From the results, the behavior of the crack extension for a safe condition of plunger PCV valve could be explained.

From the literature survey, it can be observed that much attention has been reported in the literatures on the plunger PCV valve; however, study of the diaphragm PCV valve is very sparse. Therefore, this article investigates numerical and experimental study on the flow characteristic inside the diaphragm PCV. In particular, such a simulation can offer a mechanism for visualizing the flow through the diaphragm PCV valve.

2) Simulation

In the present work, behavior of the diaphragm has been investigated by computational flow analysis using a commercial package FLUENT. The dimension of spring loaded valve is taken and the flow model is simulated. The PCV valve model consists of a main body, a moving diaphragm and the spring. The quasi-static motion for the diaphragm was assumed. In each steps of calculations, diaphragm has been closed about 5mm and it is assumed that the diaphragm force is equal to the spring force. Figure 2 shows the cross section of model. Maximum distance of the diaphragm from top of the valve is 3.16 mm. Hexahedral mesh was generated which has been depicted in Figure 3. The mesh quality has been checked for parameters such as skewness and aspect ratio and a reasonable balance between mesh size and quality has been attempted. As the mesh sizes in the orifice opening is very important, number of cells varies depend on the position of the diaphragm from 51,000 to 67,000.

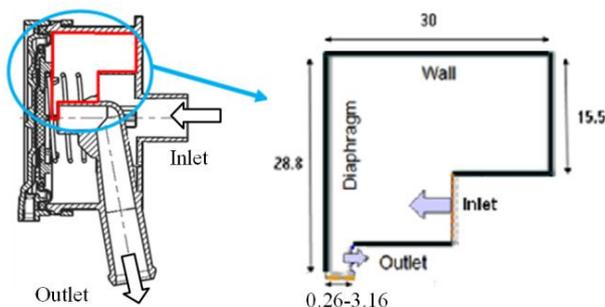


Figure 2: Cross section and dimensions of model and boundaries of computational domain (dimensions in mm)

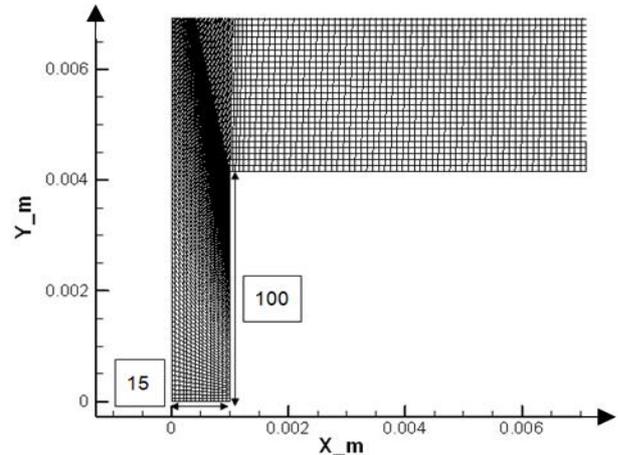


Figure 3: Mesh in the mid-Z-plane section and constant number of cells in the outlet of valve

The boundary conditions for the model consist of the inlet and outlet boundaries. The Inlet boundary is a fixed flow rate and the outlet is maintained at a pressure below atmosphere depending on the operating condition. The average pressure on inlet and the diaphragm has been calculated using the CFD code. The force exerted on the diaphragm due to pressure is equal to the force of the spring with static assumption. The CFD simulation of the PCV valve is a static analysis. A steady state procedure is used to solve the Navier-Stokes equations. A standard K- ω turbulence model is used to account for the turbulence. The fluid flow in the present investigation is in the compressible regime. Hence, compressible flows as described by the standard continuity and momentum equations; however density of the flow has to be computed by Perfect Gas Law. Blow by gas is dilute mixture of air and oil mist. It is known that in dilute gas/particles flow, the dispersed phase cannot affect on the continues phase flow. Hence in this paper it is assumed that air enters PCV valve since we just want to investigate fluid flow. For oil separation investigation it is important to choose air and oil mist.

There are two ways to simulate the movement of the diaphragm and its interaction with the spring force: The first method is to use spring characteristic curve as an input and change the diaphragm position till gas pressure on the diaphragm equals spring force. An iterative process is needed in this model to establish the valve equilibrium position. The vacuum is applied to the projected area of the valve, creates a force on the diaphragm. This leads to the movement of the diaphragm by compressing a spring. This movement of the diaphragm will change the projected area, which changes the force on the valve. This iteration will continue until the valve reaches its equilibrium position. Another method is the quasi-static solution. In this method, independent of the spring stiffness, the distance between the outlet port and the diaphragm valve is decreased while the valve is closed. In each diaphragm position, the

pressure of downstream of the valve (manifold) changes from maximum vacuum to atmospheric pressure to estimate the diaphragm and downstream (crankcase) pressure. Finally, the diagram of the diaphragm pressure versus the diaphragm position is plotted for different downstream pressure. Also the exerted pressure of the spring is plotted versus the diaphragm position. When the spring force on the diaphragm is equal to gas force, the position of the diaphragm is stable. Hence, the diaphragm positions are interaction points of spring curve with the gas curve. In this investigation the later method is implemented. Figure 4 shows the flow chart of numerical computation. Volumetric flow is considered 30 lit/min and maximum displacement of the diaphragm from top of the valve is 3.165 mm.

3) Experimental Setup

The implemented apparatus in the test has been shown in Figure 5. The air passes through the flow control valve and the vessel, and then it is entered the PCV valve. To simulate the engine operating conditions, two vessels with the same volume of crankcase and intake manifold were mounted in the test setup. A bypass valve was used to control the downstream pressure. The desired upstream pressure is achieved in a constant flow rate by adjustment of flow control and bypass valves. Finally, the downstream pressure and the position of the valve at equilibrium condition were measured, respectively, by using a pressure transducer and the laser displacement sensors.

The absolute Pressure before and after PCV valve were measured using a Kistler piezoelectric pressure transducer. To avoid any turbulence at the tip of the probe a so-called quiet length is introduced both before and after the probe. The AVL Blow-by Meter which determines the flow rate using the orifice measurement principle is used to measure the air flow rate in the test rig. The flow meter measurement range was 3 to 150 l/min with 1% accuracy. KEYENCE's laser displacement sensor has been used to measure height and position of PCV's diaphragm. The displacement measurement sensor has been performed at 50 kHz and high accuracy of $\pm 0.05\%$.

4) Results and Discussion

4.1) Numerical Results

Numerical simulation was carried out for different distances of the diaphragm from inlet, d. The details of simulation for d=1 mm has been shown in Figure 6. According to this figure, relative pressure distribution inside the half of PCV valve area has been shown. It is observed that the pressure is significantly decreased in the opening area of valve. It is also found that the pressure in the opening is less than the outlet pressure.

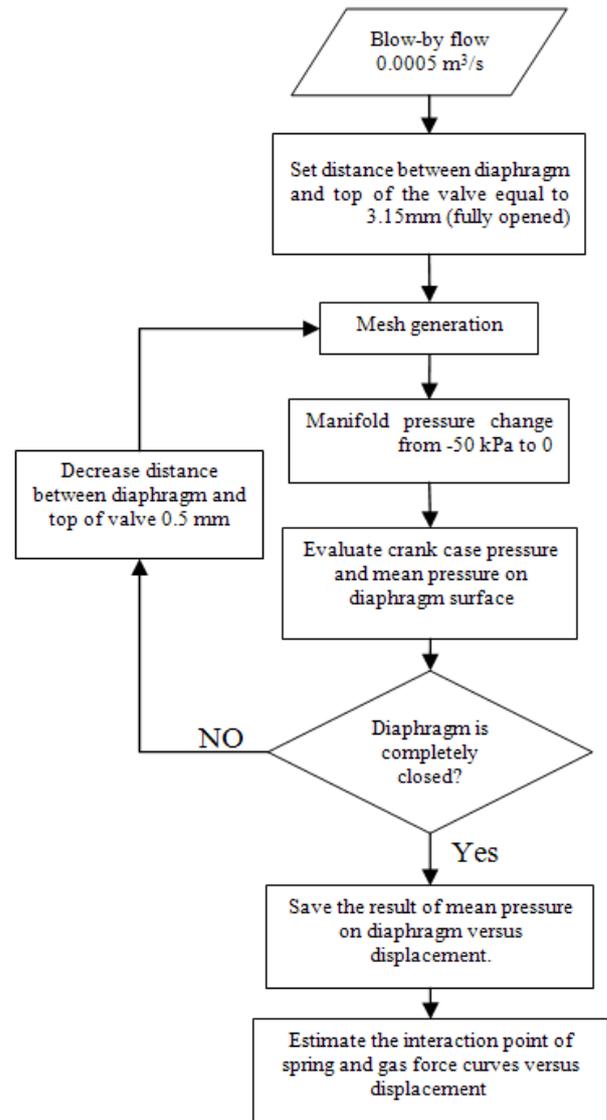


Figure 4: Flow diagram of simulation procedure

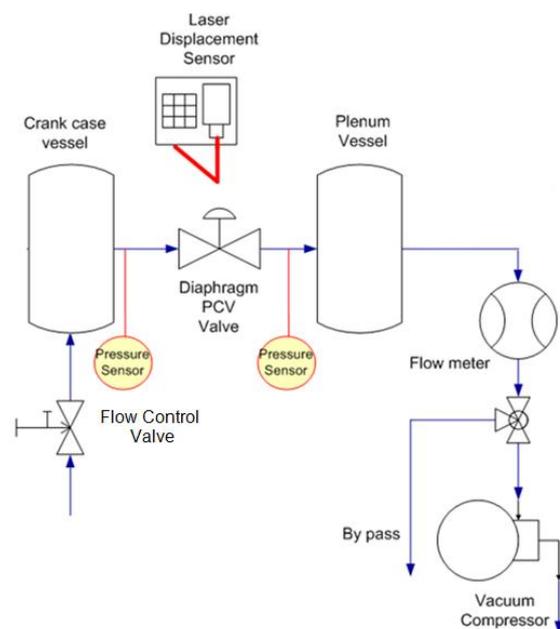


Figure 5: Schematic of experimental setup

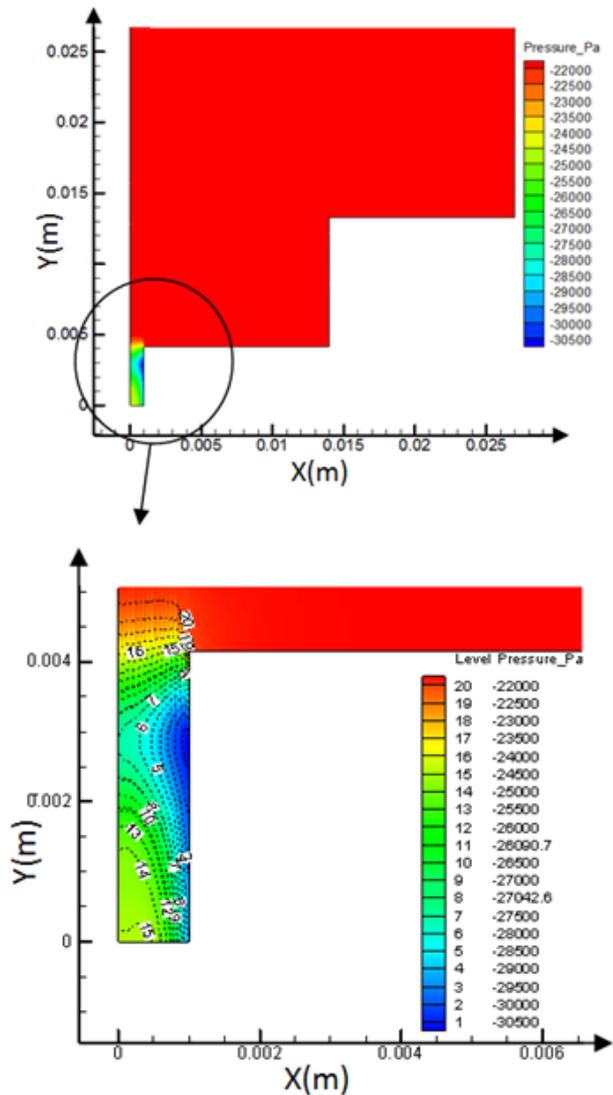


Figure 6: Contours of the gauge pressure variations in Pascal

Figure 7 shows the distributions of gas temperature. The flow area at the outlet region was decreased by the diaphragm behavior. In this condition the high velocity of gas causes the temperature to reduce approximately 10°C.

Figure 8 shows the velocity vector. Maximum velocities are 130 m/s at the outlet region. This phenomenon is important when the working fluid is a saturated gas, for example saturated mixture of water and fuel vapor. The reduction of temperature leads to condensation of mixture. The deposit of condensed liquid could damage the downstream equipments. Figure 9 shows the averaged forces on the diaphragm versus its position for different downstream pressures (Intake pressures). There are two type of forces on the diaphragm; spring force and differential pressure of gas on the diaphragm. The fluid pressure was calculated using CFD and the spring force on the diaphragm in different displacement was estimated using a simple linear springs equation. When the spring force on the

diaphragm is equal to the gas force, position of the diaphragm is stable. Hence, diaphragm positions are interaction points of spring curve (dotted curve) with gas curve (solid curve). For instance, when a -5.2 kPa partial pressure is applied, diaphragm and spring are displaced 2.1 mm. Therefore, valve is opened 1 mm. Also, -2 kPa gas pressure does not intersect spring force. It means that at this position, the diaphragm is not displaced due to the spring force.

Finally, the crankcase and intake pressure in each interaction point are extracted from the numerical result for that point. Figure 10, show the extracted characteristic curve.

4.2) Experimental Results

The performance curve of the PCV valve is achieved by plotting the intake manifold pressure versus crankcase pressure in a constant gas flow rate. In Figure 10, the predicted crankcase pressure is compared with the experimental results. The uncertainty of predicted pressure is below 10%. Also, results show that as downstream pressure (intake manifold) decreases to -50 mbar, upstream pressure (crankcase) decreases to -40 mbar. However, when the intake manifold pressure decreases more than -50 mbar, valve is closed by the diaphragm and the crankcase pressure is increased. To validate the calculated diaphragm displacement and the numerical simulations; an experimental verification for the diaphragm movement was done. The diaphragm displacement was measured using a laser displacement sensor. The measurement setup was implemented to measure the distance between laser head and the diaphragm. The laser beam was focused on the center of the diaphragm. This setup enables a contactless measurement of the surface displacements of the diaphragm during the test.

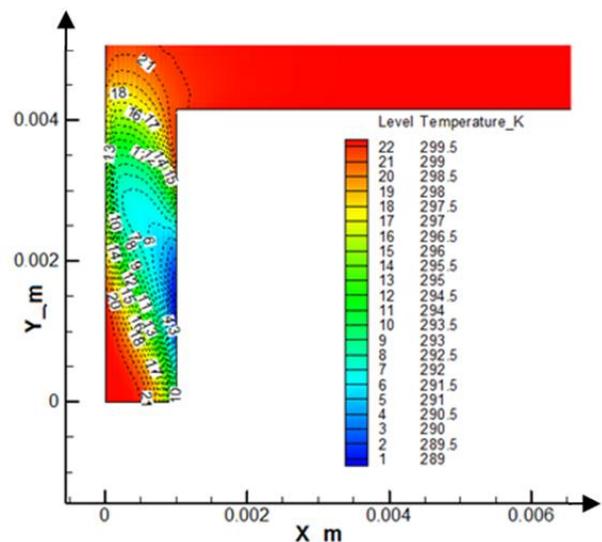


Figure 7: Temperature distribution in Kelvin

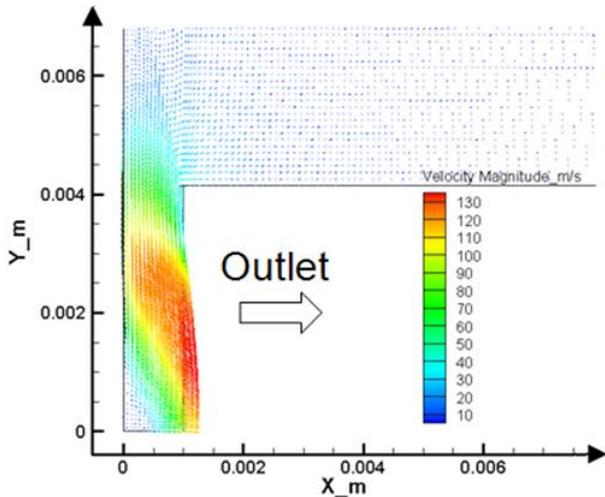


Figure 8: velocity profile in m/s

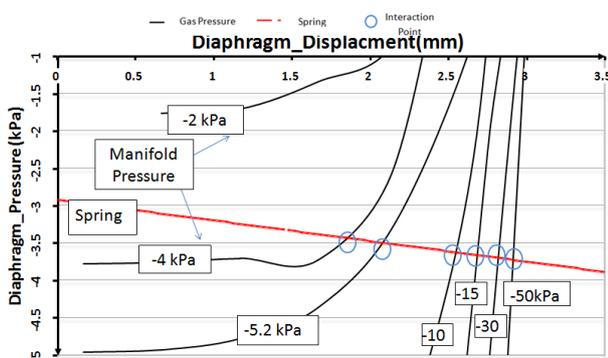


Figure 9: The example of diaphragm pressure versus its displacement for different intake manifold pressure

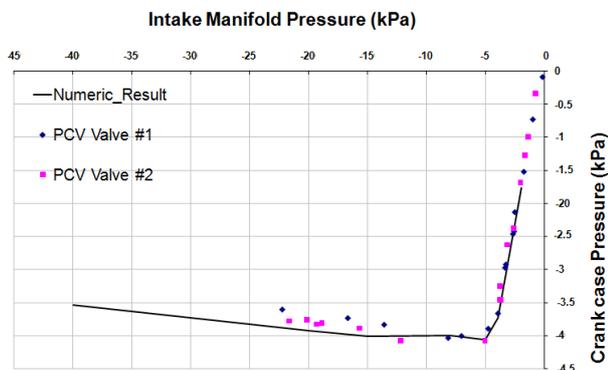


Figure 10: characteristic curve of PCV valve

The downstream pressure, Intake manifold pressure, was applied to test rig and the position of the diaphragm was measured at equilibrium position by laser sensor. Displacement of the diaphragm in relation to pressure is depicted in Figure 11. The experimental results match numerical results. However, within the range of 2.5-2.7 mm, diaphragm was oscillating (when valve outlet is approximately closed). When small shift in displacement is happened, the gas pressure on the diaphragm varies rapidly and causes unsteady motion of the diaphragm. Because the numerical simulation was semi-steady, this phenomenon could not be

considered by simulation. As the displacement of the diaphragm exceeds 2.7 mm, the damping of the diaphragm will diminish vibration.

Fluctuation in the diaphragm displacement leads to poor performance of PCV valve. The oscillating data was analyzed in order to identify the diaphragm vibration (flutter). In certain cases, oscillating pressure signal result in audible noise and in severe cases, physical vibration.

The Fast Fourier Transform (FFT) algorithm was used to analyze displacement data in order of amplitudes and frequencies. The frequency spectrum is important because it determines different frequency vibrations of the PCV. Figure 12 shows the frequency distribution for the diaphragm oscillation in case of -10 kPa pressure of intake manifold. It can be seen that the amplitude of the diaphragm motion at 20 Hz frequency was 0.016 mm.

5) Conclusions

In this work, flow characteristic of a diaphragm PCV valve is analyzed experimentally and numerically. The commercial code is used to simulate the flow inside the valve. This simulation can offer a mechanism for visualizing the flow through the diaphragm PCV valve. The valve function obtained by this methodology is verified using experimental results.

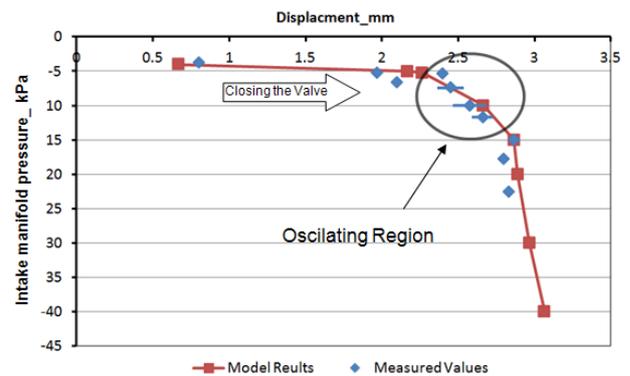


Figure 11: Effect of Intake manifold pressure on the diaphragm displacement

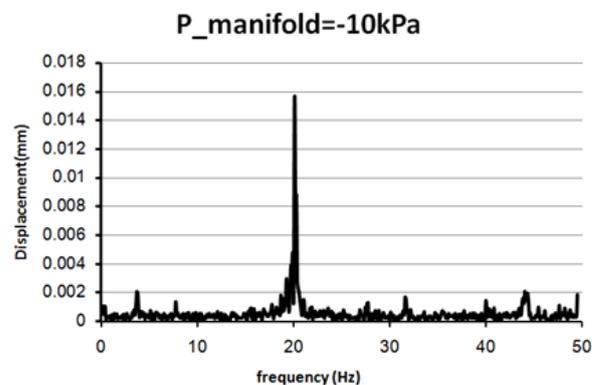


Figure 12: The frequency distribution for the diaphragm oscillation in case of -10 kPa pressure of intake manifold

The purpose of this article is to develop a new procedure for designing PCV valve. Based on the valve performance, a method is proposed to predict the effect of spring stiffness on the performance curve of the valve. Comparison of experimental and numerical results indicated that proposed method properly predicts performance curve of the valve.

Acknowledgment

The authors would like to acknowledge the support extended by Irankhodro Powertrain Company (IPCO) in carrying out this study.

References

[1] P. Avergard, F. Lindstrom, Modeling of crankcase gas behavior in a heavy-duty diesel engine, MSc Thesis, Department of Automatic Control, Lund University, Sweden, 2003
[2] P.J.P Singh, P. Lingeswaramurthy, B. Sirnivasan, S. Govindarajan, Design of positive crankcase ventilation valve for automotive gasoline engine - validation by simulation using CFD and by experiments, SAE International, Paper No. 2009-28-0017, 2009
[3] M.D. Tajul, J.H. Lee, M.M.A. Sarker, Y.W. Lee, Numerical investigation of fluid flow characteristics in a PCV valve, Proceedings of the International Conference on Mechanical Engineering, 2005

[4] M. Foerg, F. Borchsenius, F. Pfeiffer, Simulation of air pressure control valves for crankcases, Proceedings of Conference on Computational Fluid and Solid Mechanics, 2003
[5] H. Chattopadhyay, A. Kundu, B.K. Saha, T. Gangopadhyay, Analysis of flow structure inside a spool type pressure regulating valve, Energy Conversion and Management, Vol. 53, pp. 196-204, 2012
[6] N. Gomez, PCV valve flutter: Vibration characterization through pressure and flow, BSc Thesis, Massachusetts Institute of Technology, 2005
[7] D.C. Freeman, Nonlinear springs with applications to flow regulation valves and mechanisms, PhD Thesis, Massachusetts Institute of Technology, 2008
[8] H.C. Dhariwal, Control of blow-by emissions and lubricating oil consumption in I.C. engines, Energy Conversion and Management, Vol. 38, No. 10, pp. 1267-1274, 1997
[9] S.M. Song, O.H. Kwon, Y.W. Lee, Effect of the differential pressure by the blow-by gas flow on the PCV valve with a crack, International Journal of Automotive Technology, Vol. 8, No. 2, pp. 219-224, 2007



فصلنامه علمی - پژوهشی تحقیقات موتور

تارنمای فصلنامه: www.engineersearch.ir



بررسی عملکرد شیر دیافراگمی تنظیم فشار محفظه لنگ

سید شهاب‌الدین علویون^۱، محمد علی احترام^{۲*}

^۱دانشگاه صنعتی خواجه نصیرالدین طوسی و شرکت تحقیق، طراحی و تولید موتور ایران خودرو (اییکو)، تهران، ایران، sh_alaviyoun@ip-co.com

^۲دانشگاه صنعتی قم، قم و شرکت تحقیق، طراحی و تولید موتور ایران خودرو (اییکو)، تهران، ایران، ali_ehteram@aut.ac.ir

*نویسنده مسئول، شماره تماس: ۰۲۵-۳۶۶۴۱۶۰۱

اطلاعات مقاله

چکیده

تاریخچه مقاله:

دریافت: ۰۷ مهر ۱۳۹۲

پذیرش: ۲۷ آذر ۱۳۹۲

کلیدواژه‌ها:

شیر تنظیم فشار

موتور احتراق داخلی

فشار محفظه لنگ

دیافراگم

سختی فنر

در این پژوهش، رفتار شیر تنظیم فشار دیافراگمی محفظه لنگ به صورت عددی و تجربی شبیه‌سازی شد و منحنی عملکرد شیر بر حسب سختی فنر استخراج شد. این شیر با استفاده از یک دیافراگم لاستیکی و فنر و با توجه به شکل ورود و خروجی آن، فشار محفظه لنگ را بر حسب فشار چندراهه هوا تنظیم می‌کند. شبیه‌سازی عددی برای جریان تراکم‌پذیر و در حالت تقارن محوری انجام شده است. در حل عددی، جابجایی دیافراگم به صورت شبه‌تصادفی در نظر گرفته شد. به منظور بررسی تجربی شیر، فشار قبل و بعد از شیر به همراه شار جریان اندازه‌گیری شد. در عین حال جابه‌جایی و نوسانات دیافراگم بر حسب فشار با استفاده از فاصله‌سنج نوری اندازه‌گیری شد. مقایسه نتایج تجربی با نتایج شبیه‌سازی نشان‌دهنده قابلیت مناسب شبیه‌سازی ارائه شده برای پیش‌بینی منحنی فشار میانگین محفظه لنگ بر حسب فشار چندراهه هوا به همراه مقدار جابه‌جایی دیافراگم در فشارهای مختلف است.

تمامی حقوق برای انجمن علمی موتور ایران محفوظ است.