Steady flow and dynamic analyses comparison of an air intake breathing capacity

Nur Adila Mohamad Shafie¹, Mohd Farid Muhamad Said^{1,}, Azhar Abdul Aziz¹, Zulkarnain Abdul Latiff¹, Ahmad Kamal Mat Yamin², and Noreffendy Tamaldin²*

¹Automotive Development Centre (ADC), Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM), Johor, Malaysia.

²Automotive Department, Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka (UTeM)

Abstract. Analysis of intake breathing capacity is important in order to determine the performance output for any respective engine. Flow coefficient and discharge coefficient are the common parameters used to define the breathing capability and different analysis have been applied by researchers to determine these parameters which include experiments and Computational Fluid Dynamics (CFD) analyses. This study aims to investigate the difference in breathing parameters obtained from steady flow simulation and dynamics simulation. Proton CAMPRO 1.6-litre engine was selected as the reference engine in this study. The experiment involved was the flow bench test, while CFD simulation carried out were the port flow analysis (steady flow) and the cold flow analysis (dynamics). Results obtained indicate that flow coefficient and discharge coefficient from cold flow simulation are always lower than both parameter values obtained from port flow simulation and experiment with large deviations of minimum 15.6% and maximum 27%. Meanwhile, the breathing parameter values from port flow simulation were very close to the experimental data with the minimum deviation of 1.6%. This study concludes that port flow simulation is very accurate for the analysis of defining intake breathing capacity, meanwhile cold flow simulation can be used to predict the trend and lower limit of these parameters especially at low valve lift.

1 Introduction

As the most important aspect of the automobile system, engine performance is analysed in details in fulfilling the important parameters which include geometrical properties and also different engine efficiencies. Aside from thermal efficiency and mechanical efficiency, volumetric efficiency is another crucial parameter which represents the effectiveness of air induction in engine [1, 2]. Analysis of suction capability of the engine system is important as it influences the performances output of any respective engine [3]. In the effort of improving engine system, CFD simulation has emerged from years ago as an alternative

* Corresponding author: mdfarid@utm.my

and has been used alongside other methods which include Particle Image Velocimetry (PIV) and flow bench test. In analysing the intake air suction of engine, questions have surfaced on whether the steady flow test can accurately determine the engine breathing parameters of a moving car. Based on previous study, it has been defined that discharge coefficient and flow coefficient from steady flow analysis can represent real operation of car engine as long as it is under normal speed range [1]. CFD simulation is used for the analysis for optimization of the cost and timing [3]. There are many researchers that have adapted the use of CFD simulation and proved the significance of this method for engine study. Research by Balasankar *et. al* [3] had discussed on the optimization of intake port in order to improve the engine performance. This research had investigated on the computation of port flow at different valve lifts by CFD solver. Intake flow data was compared with the flow bench test, and the research proceeded into optimization of port geometry by DOE analysis. Balasankar *et. al* [3] had presented good correlation between port flow simulation and flow bench test, leading to a conclusion that CFD simulation is a reliable method to replace the conventional method of engine intake analysis.

Ismail and Abu Bakar [2] had discussed with great details about engine induction by experiment of Superflow bench test. It was proven that the steady flow bench test can provide high accuracy on the flow analysis. In the research, detail design of the port and the valve were described and the sources of flow losses were listed which include the bend at the valve guide, bend to exit valve and push rod expansion [2]. Analysis by Ismail and Abu Bakar [2] concluded that increasing the test pressure and the valve lift allowed the increment in sucked flow. This study also concluded that the increment of flow does not occur and stabilized after it reaches the ratio of maximum valve lift per diameter (L/D) of 0.25. Research by Kumar and Nagarajan [4] also investigated on those non-dimensional parameters with the inclusion of the swirl ratio analysis. Analysis showed the correlation of the obtained data from the flow bench test and also from the modelling by the commercial CFD code. This research discussed on the swirl improvement by modification of the valve design. Kumar and Nagarajan [4] defined that valve shrouding and twisted tape insert can improve the swirl in engine. From the research, it was also concluded that although flow increased along the valve lift, this took a penalty on the increased friction.

Port flow simulation in internal combustion engine was further analysed by Himanth and Jayashankar [5]. The geometry model created in ANSYS Fluent was constructed as close as possible to the proposed theoretical model. The main stages for problem solving in ANSYS Fluent involved decomposition, set up the engine case and performed the moving simulation. Himanth and Jayashankar [5] proposed on the use of growing prismatic cells at the surface of the valve to solve the turbulent boundary layer. Meanwhile, the computational model surface was meshed with triangular element and the remaining decomposed region was meshed with tetrahedral cells. This research concluded that as the losses in flow increased along the valve lift, the separation of flow becomes critical. It was also defined that even though valve lift influenced the flow, there was exception for the upstream of port bend [5].

The complex-problem solving capability of the CFD simulation is emphasised more in the analysis of engine by Chaudhari *et.al* [6], Morauszki *et.al* [7] and Schmidt *et.al* [8]. Injection of fuel and combustion were reflected close to reality in the research where it is proven that CFD simulation is superior in engine analysis with respect to time and cost optimization. In the research done by Chaudhari *et.al* [6], analysis on a 2-dimensional spark ignition engine with the validation of experiment and zero-dimensional simulation had proven that deviation of data is larger at high engine speed compared to low one. Meanwhile, Morauszki *et.al* [7] had used CFD to provide greater details on the visualization of flow in engine, mixing, injection and combustion occurred in gasoline engine aided with fuel direct injection. This research had introduced the idea of reducing

mesh number while countering the loss of data by refining the wall properties and its solver equation. On the other hand, Schmidt *et.al* [8] had also investigated on the operation in gasoline direct injection (GDI) engine using CFD. This study had visualized the engine phenomenon which include injection and combustion in the engine, and finding had defined the design parameter that influence the performance of engine.

Eventhough the solving capability of CFD simulation is undeniably excellent, the full analysis of engine is always separated into different stages; port flow or experiment for defining initial breathing parameters values, and CFD moving simulation for the other related operations involved in analysis of flow and combustion. Based on the study of previous researches, there is none of the study shows the use of moving simulation in finding breathing parameter. The study of engine breathing is always either by experiment or CFD port flow simulation where both are under the assumption of steady flow. Meanwhile, dynamics simulation by CFD has always been used to investigate other parameter aside from engine suction capability which include engine combustion (heat release, peak temperature, pressure) and internal flow properties (swirl, tumble). This has caused the questions to surface on whether moving simulation can be used to predict the ability of engine suction and what is the accuracy of dynamics simulation over steady flow simulation for the engine breathing study.

Based on the study of previous researches, steady flow analysis by CFD simulation has shown good correlation with experimental data. However, the tremendous improvement of CFD has make the dynamic analysis on engine much easier, and this has picked the interest in determining whether engine breathing analysis in dynamic operation also provides good correlation with the experiment. Thus, this research aims to investigate the difference of flow coefficient and discharge coefficient obtained from dynamic analysis (cold flow) and steady flow analysis (port flow) with the experiment (flow bench test). Research objectives are mainly to investigate the ability of moving simulation in engine breathing analysis, and to determine the accuracy of dynamics simulation compared to the steady flow simulation in study of engine suction. Proton CAMPRO 1.6-litre engine was selected as the analysed engine and commercial CFD software, Fluent is used to simulate the steady flow and dynamics analyses.

2 Literature review

2.1 Governing equations

In order to represent the fluid flow using the CFD codes, the governing equations are required for defining the flow cases. For internal combustion engine, the governing equation for conservation of mass is defined as [5]:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho.u)}{\partial x} + \frac{\partial(\rho.v)}{\partial y} + \frac{\partial(\rho.w)}{\partial z} = 0 \quad (1)$$

The governing equation for the conservation of momentum can be written as [5]:

$$\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} + \rho w \frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \mu \frac{\partial^2 u}{\partial x^2} + \mu \frac{\partial^2 u}{\partial y^2} + \mu \frac{\partial^2 u}{\partial z^2} \quad (2)$$

$$\rho \frac{\partial v}{\partial t} + \rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} + \rho w \frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \mu \frac{\partial^2 v}{\partial x^2} + \mu \frac{\partial^2 v}{\partial y^2} + \mu \frac{\partial^2 v}{\partial z^2} \quad (3)$$

$$\rho \frac{\partial w}{\partial t} + \rho u \frac{\partial w}{\partial x} + \rho v \frac{\partial w}{\partial y} + \rho w \frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + \mu \frac{\partial^2 w}{\partial x^2} + \mu \frac{\partial^2 w}{\partial y^2} + \mu \frac{\partial^2 w}{\partial z^2} \quad (4)$$

Equation for conservation of energy is defined as [5]:

$$\begin{aligned} \frac{\partial}{\partial t} \left(\rho e + \frac{1}{2} \rho v^2 \right) + \frac{\partial}{\partial x} \left(\rho u e + \frac{1}{2} \rho u v^2 \right) + \frac{\partial}{\partial y} \left(\rho v e + \frac{1}{2} \rho v v^2 \right) + \frac{\partial}{\partial z} \left(\rho w e + \frac{1}{2} \rho w v^2 \right) = \\ k \left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) - \left(u \frac{\partial p}{\partial x} + v \frac{\partial p}{\partial y} + w \frac{\partial p}{\partial z} \right) + \\ \mu \left[u \frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x} \left(v \frac{\partial v}{\partial x} + w \frac{\partial w}{\partial x} \right) + v \frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y} \left(u \frac{\partial u}{\partial y} + w \frac{\partial w}{\partial y} \right) + w \frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z} \left(u \frac{\partial u}{\partial z} + v \frac{\partial v}{\partial z} \right) \right] \\ + 2\mu \left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial u}{\partial y} \frac{\partial v}{\partial x} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z} \frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} + \frac{\partial w}{\partial x} \frac{\partial u}{\partial z} \right] + \rho u g_x + \rho v g_y + \rho w g_z \quad (5) \end{aligned}$$

2.2 Flow coefficient and discharge coefficient

In defining the capabilities of engine suction, this analysis represents the breathing capabilities in term of flow coefficient and discharge coefficient. Based on study of previous researches, there are different definitions that have been used for flow coefficient and discharge coefficient. This study however rely on the definition indicted by Xu [9] as it has been used more often in the intake flow analysis by car manufacturer. Flow coefficient, C_F is defined as [9-13]:

$$C_F = \frac{\dot{m}}{\rho A_p V_o} \quad (6)$$

where $\dot{m}$ is air mass flow rate, ρ is air density at operating condition, V_o is the velocity of air and A_p is the area of valve inner seat. Valve inner seat area can be obtained by the following formula:

$$A_p = \pi D_v^2 / 4 \quad (7)$$

where D_v is the valve diameter. Closely related to the flow coefficient, discharge coefficient is defined as [9-13]:

$$C_D = \frac{\dot{m}}{\rho A_c V_o} \quad (8)$$

where A_c is the valve curtain area/gap area. Curtain area can be calculated by the formula below:

$$A_c = \pi D_v L_v \quad (9)$$

where L_v is valve lifts and D_v is diameter of valve. Air velocity can be obtained by the following equation [4, 9, 11, 13]:

$$V_o = \sqrt{2\Delta P / \rho} \quad (10)$$

where ΔP is the pressure difference.

3 Research methodology

This research focuses on the comparison between steady flow and dynamic analyses of CFD simulation with flow bench test. In CFD simulation for engine, it is divided into four

different simulations which are port flow, cold flow, in-cylinder combustion and full-engine full-cycle simulation. Each simulation type differs from each other based on the operations and also the output. For steady flow analysis, it was computed by means of port flow simulation while cold flow simulation was computed for dynamics analysis of breathing parameter.

3.1 Experimental method (flow bench test)

Methodology:

1. For the actual flow bench test, test orifice plate was removed from the flow bench and the engine cylinder head, cylinder adapter and valve lifter were installed.
2. The dial indicator was set to zero with the valves completely closed.
3. At the FlowCom panel, the test pressure value of 25" H₂O was inserted.
4. As an initial step, leakage test must be performed with all the valves closed. All the parts that may lead to leakage were sealed with the clay.
5. The motor was then turned on to define the leakage value. The leakage value was then either added into the data sheet separately or to the FlowCom.
6. The valve in the test head was then lifted to 1mm opening.
7. The motor was turned on until desired test pressure is reached and the FlowCom automatically displayed the measured flow (cfm).
8. Step 6-7 were repeated with the increment of 1mm until it reach 8mm (maximum opening for the selected CAMPRO engine) and the reading of flow for each valve lift was recorded. Figure 1 shows the setup of flow bench test conducted in the analysis of intake air suction.

Fig. 1. Flow bench test.

3.2 Steady flow analysis (port flow simulation)

In the computation of the fluid flow in engine, one of the most important steps is the construction of the computational model. The computational model must be accurate in order to simulate the real operations of engine. Engine parameters for CAMPRO 1.6-litre engine is shown in Table 1 and the computational model is shown in Figure 2.

Table 1. Engine parameters [14].

Engine Parameters	Value
Bore (mm)	76
Stroke (mm)	88
Crank radius (mm)	44
Valve diameter (mm)	29
Number of cylinder	4
Connecting rod length (mm)	131
Total displacement (cc)	1,597
Compression ratio	10:1
Intake valve open (cad)	348.2
Intake valve close (cad)	555.3
Exhaust valve open (cad)	149.3
Exhaust valve close (cad)	372.2

Fig. 2. Computational model of engine.

For port flow simulation, the original computational model was extended into “inplenum” and “outplenum” as shown in Figure 3. The inclusion of “inplenum” assisted the suction of air into the engine in the simulation. The box shape of “inplenum” was selected among other shapes which include spherical and hemisphere as the flow output for the CAMPRO engine is more accurate for the box “inplenum”. The dimension for the box “inplenum” is 152.14mm (length) x 152.14mm (width) x 152.14mm (height) and the bending radius between “inplenum” and the intake port is 10 mm. As the simulation only involved analysis on intake breathing capacity, the exhaust port and exhaust valve were detached from the computational model.

Fig. 3. Computational model for port flow.

The computational model was then decomposed mainly into “inplenum”, port, valve, combustion chamber, and the “outplenum”. At the intake port, three inflation layers were set for the intake wall and the rest of model parts were decomposed with tetrahedral cell. In port flow simulation, the model remains static thus the decomposition and meshing are much simpler than simulation involving dynamics process. The mesh quality was particularly refined near the valves in order to capture the properties of induced flow. In the setup of the engine case, the inlet pressure at “inplenum” was set to 0 and the outlet pressure at “outplenum” was set to -6225kPa.

3.3 Dynamic analysis (cold flow simulation)

Cold flow simulation is able to predict various phenomenons in engine without the reaction and combustion, for example the formation of swirl and tumble, wall impingement and turbulence under valves movement. For the analysis of only engine breathing parameters, cold flow is prior to in-cylinder combustion and full-engine full-cycle simulation which are much complex and time-consuming. Cold flow analysis is also adequate for this study because the simulation is only along the intake stroke which does not involve other critical process for example combustion and spark ignition. For cold flow simulation, as this analysis involves the transient process of the engine, the computational model is decomposed in different manner from the model in port flow simulation. Model was not extended into “inplenum” and “outplenum”, and the original model was decomposed into different regions as shown in Figure 4.

Decomposition and meshing in cold flow analysis determine which parts remains static or moving during the simulation. As the piston and valves move along the engine stroke, another layers consist of hexahedral cells were added for piston and valve. Meanwhile, other regions of the computational model were meshed with tetrahedral cells.

Fig. 4. Engine decomposition.

4 Results and discussion

In the conduction of flow bench testing, this experiment was repeated for a number of times in order to investigate the accuracy and the precision of the data obtained. Table 2 shows the volume flow rate (cfm) obtained for each valve lifts. From the data tabulated, it can be observed that flow data measured for each valve lifts are very close to each other. There is only a very small deviation of data for low valve opening (1mm) as the flow is more restricted at low valve lift [5].

Table 2. Flow (cfm) obtained from flow bench test.

Flow bench test	Valve lift (mm)							
	1	2	3	4	5	6	7	8
	Volume flow rate (cfm)							
1	12.3	26.6	39.7	52.6	65.4	75.3	82.3	83.4
2	12.2	26.6	39.7	52.6	65.4	75.3	82.3	83.4
3	12.4	26.6	39.7	52.6	65.4	75.3	82.3	83.4
Average	12.3	26.6	39.7	52.6	65.4	75.3	82.3	83.4

From the flow bench test, the flow (cfm) obtained were tabulated into the data sheet for the calculation of discharge coefficient and flow coefficient. Table 3 shows the calculation of both parameters with reference to other listed important parameters.

Flow coefficient obtained from flow bench test, cold flow and port flow simulation is shown in Figure 5 below. From 1mm valve lift until 8mm valve lift, flow coefficient increases as the valve lift increases. This is because the valve lift is critical in affecting the flow downstream as well as determining the separation of flow [5]. Based on previous research, the breathing parameter from flow bench test and port flow simulation are always in good correlation [5, 15]. This study has defined that for all of the methods; flow bench test, port flow and cold flow, the trend of the flow coefficient is the same. Comparing port flow data with flow bench test data, the maximum difference is as low as 5.1% which is very accurate. However, comparing cold flow data with experimental data, the deviation is quite large which is the maximum of 27% and the minimum of 15.6 %. This differences is expected as port flow closely resembled the flow bench in term of setup and boundary condition where engine is hold static at each critical point [3, 5, 12]. Meanwhile, cold flow simulation deals with the tremendous change of parameters during transition from one

critical point to another, thus causing more losses in induced flow for moving simulation [5, 6].

Table 3. Calculation of flow coefficient and discharge coefficient.

		Valve lift (mm)							
Parameter	Unit	1	2	3	4	5	6	7	8
Test Pressure	"H ² O	25	25	25	25	25	25	25	25
Volume Flow Rate	cfm	12.3	26.6	39.7	52.6	65.4	75.3	82.3	83.4
Valve Curtain Area	m ²	0.000 18	0.000 36	0.000 55	0.000 73	0.000 91	0.001 1	0.001 3	0.001 5
Valve Seat Area	m ²	0.001 3	0.001 3	0.001 3	0.001 3	0.001 3	0.001 3	0.001 3	0.001 3
Flow Velocity	m/s	102.8 5	102.8 5	102.8 5	102.8 5	102.8 5	102.8 5	102.8 5	102.8 5
Air density	kg/ m ³	1.18	1.18	1.18	1.18	1.18	1.18	1.18	1.18
L/D		0.034	0.069	0.10	0.14	0.17	0.21	0.24	0.28
Discharge Coefficient	t	0.310	0.335	0.334	0.332	0.330	0.317	0.297	0.263
Flow Coefficient	t	0.042 8	0.092 5	0.138	0.183	0.227	0.262	0.286	0.29

Even though the deviation is big, the tabulated trend shows that cold flow simulation can be used to investigate the trend and predict the lower limit for the flow coefficient. After doing repeated data collection and analysis, it is defined that the data is more precise for lower valve lift which is around 1-4mm. Beyond 4 mm, repeated cold flow simulation shows large differences.

Fig. 5. Flow coefficient versus lift/ diameter (L/D).

The study on comparison between cold flow (dynamic analysis) and port flow (steady flow analysis) with flow bench test is further discussed in term of discharge coefficient as shown in Figure 6.

Fig. 6. Discharge coefficient versus lift/ diameter (L/D).

Similar to flow coefficient, the trend for discharge coefficient obtained from flow bench test, port flow and cold flow are all the same. Port flow simulation data is recorded with minimum and maximum deviation as small as 1.6% and 5.1% respectively which show that port flow simulation can accurately predict the breathing parameter of engine. For cold flow simulation, the maximum difference defined is 27% and minimum deviation is determined as 15.6%. Eventhough the large deviation causes the difficulties to accurately predict the real breathing parameter value, the similarities in trend shows that cold flow can be used to predict the trend of discharge coefficient and the lower limit of this breathing parameter. As discharge coefficient is closely related to flow coefficient and only differs in term of referred valve area, it is also deduced that the differences of data obtained is because there is more losses in the valve movement from lower valve lift to high valve lift in cold flow moving simulation [5, 6] compared to port flow engine simulation where losses in volume flow rate is measured at static critical point [3, 5, 12]. In a similar manner to flow coefficient data, repeated simulation shows that discharge coefficient is more precise at 1- 4mm valve lift, thus cold flow simulation is suitable for the prediction of trend and lower limit at lower valve lift value.

Both flow and discharge coefficients discuss on the mass flow rate that pass through the valve. These parameters only differ by reference area, where flow coefficient refers to inner seat area of valve, while discharge coefficient is dependent on gap of valve lip and valve seat. Results in Figure 5 and Figure 6 show that flow coefficient increase along valve lift, while discharge coefficient decreases along the opening of intake valve. This is because as the valve opens, greater flow is induced into the system. As valve inner seat area remains the same along valve lift, the flow coefficient remain increased [3, 5, 9]. Meanwhile, along valve lift, the valve curtain area also increase, thus causes the reduction of discharge coefficient value [5, 9, 12].

5 Conclusions

Based on the obtained result, it can be concluded that as valve lift increases, flow coefficient increases while the discharge coefficient decreases. As discharge coefficient

refers on the valve curtain area, discharge coefficient should be used to perform analysis at low valve lift value while flow coefficient should be used for analysis at high valve lift. The port flow is proven to give good correlation with the experimental data of flow bench test. Meanwhile, cold flow simulation prediction on breathing parameter is not so close to the experiment and port flow simulation where even the smallest deviation is quite large, thus does not allow accurate prediction on the values of flow coefficient and discharge coefficient. Eventhough cold flow simulation give high deviation with the experiment, this research has defined that cold flow analysis has come out with similar trend of breathing parameter compared to port flow and flow bench test. This trend predicted by cold flow simulation is also more similar at lower valve lifts. Thus, it can be concluded that cold flow analysis can still be used to predict the trend and the lower limit of flow coefficient and discharge coefficient value.

This work acknowledges the financial support from Universiti Teknologi Malaysia (UTM) and Ministry of Higher Education Malaysia (MOHE) under the FRGS grant R.J130000.7809.4F174. Appreciation also goes to Automotive Laboratory, Universiti Teknikal Malaysia Melaka (UTeM) for assisting in the flow bench test.

References

1. J.B. Heywood, *Internal Combustion Engine Fundamentals*, (United States of America: McGraw-Hill, Inc., 1988)
2. A.R. Ismail and R. Abu Bakar, *Amer. J. Appl. Sc.*, **5**, 963 (2008)
3. P. Balasankar and A. Gokhale, *HyperWork Technology Conference*, 1-5 (2012)
4. C.R. Kumar and G. Nagarajan, *ARPN J. Eng. Appl. Sc.*, **7**, 180-186 (2012)
5. Himanth Kumar H.Y. and Jayashankar N., *IJIERT*, **2**, 1-9 (2015)
6. A.J. Chaudhari, N. Sahoo, and V.Kulkarni, *4th ICAER 2013*, 330-341, (Elsevier Ltd. 2014)
7. T. Morauszki, P. Mandli, Z. Horvath, and M.R Dreyer, *Hung. J.Ind.Chem.* **39**, 27-30 (2011)
8. S. Schmidt, M. Abart, O.Schogl, D. Jajcevid, A. Trattner, R.Kirchberger, and H.Eichlseder, *SETC2009*, 2009-32-0045 / 20097045 (2009)
9. H. Xu, *SAE Technical Paper Series* 2001-01-1308 (2001)
10. H. Bettes, *Engine Airflow: A Practical Guide to Airflow Theory, Parts Testing, Flow Bench Testing, and Analysing Data to Increase Performance for Any Street or Racing Engine*, (USA: HPBooks, 2010)
11. A. Mohammedbrahim, B. Shafiei, and S. K. Hannani, *The J.Engine.Research*, **26**, 11-19 (2012)
12. B. Soriano, and C. Rech, *Int. J. Mech. Eng. Autom.*, **1**, 236-241(2014)
13. A. Algieri, *Int.J.Eng.Tech.*, **2**, 140-148 (2013)
14. M. F. Muhamad Said, A. Abdul Aziz, Z. Abdul Latiff, and A.M. Andwari, *2014 SAE International*. 2014-01-2549 (2014)
15. H.Cui, M. Li, and Y. Han, *2010 WASE International Conference on Information Engineering*, 255-259 (2010)