Performance Analysis for 4-Cylinder Intake Manifold: An Experimental and Numerical Approach

First A. Jason D'Mello, Second B. Omkar S. Siras

***Abstract*— The primary function of the intake manifold was to deliver the air / air-fuel mixture to the engine cylinder through the intake port with least flow losses. Also, based on the engine cylinder firing order, the flow must be evenly split among the cylinders. This had been investigated in this work for a 4-Cylinder IC engine intake manifold for five flow rates – 4 kg/min, 4.5 kg/min, 5.0 kg/min, 5.5 kg/min and 6.0 kg/min. CFD simulations, using STAR CCM+, were carried out for estimating the flow losses, mass flow distribution between the engine cylinders, swirls inside the intake manifold. The Realizable k-epsilon turbulence model with All Y+ Two Layer Model was applied for these simulations. An experimental validation was also carried out. An innovative boundary condition method for the CFD simulations was suggested for improving the CFD simulation accuracy. The flow path for the cylinders 2 and 4 provide high flow losses. Also, un-even distribution of the mass flow between the ports had been observed.**

***Index Terms*— Intake Manifold, Swirl, CFD boundary conditions, IC Engine CFD**

I. Introduction

The primary function of the intake manifold was to deliver the air / air-fuel mixture to the engine cylinder through the intake port with least flow losses. Certain intake manifolds were designed to enhance the flow swirl in the intake manifold to improve the combustion in the engine cylinder.

Fig.1 Intake Manifold Geometry

Also, based on the engine cylinder firing order, the intake manifold must supply evenly split air flow among the cylinders. This had been investigated in this work for a 4-Cylinder IC engine intake manifold.

Jason D'Mello, is pursuing M.E from Flora Institute of Technology, Pune India.

Omkar S. Siras, is working with Flora Institute of Technology, Pune India as Assistant Professor.

Recent developments in the computer simulation based methods for designing automotive components had been gaining popularity. Even though the results obtained from these numerical simulations (CFD) were comparable with the experimental studies, there's been continuous research to improve the simulation accuracy.

In the available literatures, the intake manifold CFD simulations were performed by considering all the ports to be open. But, in actual conditions for the 4-cylinder engines, only two ports – based on the firing order- would be open. The other two port would remain closed.

A similar condition was imposed in this study. The flow outlet condition was imposed for the two ports and the wall boundary conditions for the remaining two ports.

II. Literature Review

Suresh Aaepu [1] had designed an intake manifold for a twin cylinder CI engine using CFD methods. They had combined the 1-Dimensional Engine simulation software with the 3-Dimensional Steady state CFD simulations. Y. K. Loong [2] had undertaken experimental and CFD simulation to improve the engine performance by optimizing the intake manifold port. With the k-epsilon turbulence model in their CFD simulations, the authors were able produce the results which were comparable to experimental data. M. A. Jemni [3] had applied the 3D CFD simulations with k-epsilon turbulence model for estimating the performance of the intake manifold. The conditions corresponding to crank angle of 130˚, and adiabatic conditions were assumed. Their results were validated using the experimental approach. The intake manifold for a 200 cc engine was analyzed using CFD simulations in ANSYS FLUENT by S. A. Sulaiman [4]. The authors had studied for various geometrical configurations such as runner length, surge tank parameters etc. The flow coefficient from the CFD simulations and the experimental work was compared. Per Ahlm [5] had developed a manifold for the diesel engine of marine application. The flow conditions were calculated based on the stoichiometric ratio of 14.3 for the diesel. Even distribution of mass flow was achieved in the V 12-cylinder engine. Long Xie [6] had developed an algorithm based on the thermodynamic and hydrodynamics from the experimental data that resulted in differential equations. The authors had solved these set of equations using fixed step Euler method. Steady and Transient

CFD simulations for the intake manifold was conducted by Negin Maftouni [7] in their research. Using the 1-D WAVE software, the auhors had obtained the boundary conditions for the transient simulations. The pressure drop across the manifold predicted from the steady state simulations were compared against the experimental work and found to be in the acceptable range. Rajesh Holkar [8] had performed steady-state CFD simulation studies using ANSYS FLUENT for intake systems. The authors had applied inlet conditions at the manifold inlet and the outlet conditions for all the 4 ports. K-epsilon turbulence model with the wall functions were employed in their study. S. Karthikeyan [9] had used CFD simulation methods to investigate flow field inside the intake manifold. From these simulations, the disproportionate quantity of air in the manifold was identified and the necessary optimization was suggested. P. Ragupathi [10] had applied an innovative method of inserting curved pipes from the plenum to the runners in the intake manifold. The authors had conducted their study using CFD simulations. In their research on intake port system analysis using ANSYS FLUENT, M. M. Khan [11] had compared the RANS turbulence models – Standard k-epsilon, Realizable k-epsilon, Re-Normalization Group k-epsilon, k-omega and the RSM. And, they conclude that the RNG k-epsilon model is better suited for the swirl dominated flows. In another innovative method to improve the engine performance, D. Ramasamy [12] had included the guide vane in the air intake system. The authors had observed nearly 12% reduction in flow losses by the inclusion of the guide vanes in the air intake system. G.Venkata Punna Rao [13] had compared the Cast Iron and Aluminum alloy as the material for the intake manifold by conducting the vibration analysis in ANSYS. In their study, they had included the effects of engine vibration and the pressure pulsation loads for consideration. The maximum deflection as well as maximum stress were predicted to be higher in the Cast Iron model than the Aluminum alloy model. Hence, the authors suggested Aluminum alloy as the preferable material for the intake manifold construction. Amit Kumar Gupta [14] had developed the intake manifold for a 6-Cylinder, 7 liter diesel engine, having a firing order of 1-4-2-6-3-5. They had developed three variants- removal of sharp corners, increased radius of walls to decrease the deflection near the wall- for optimizing.

III. PROBLEM DESCRIPTION

In this study, CFD simulations were carried out for estimating the flow losses, mass flow distribution between the engine cylinders, swirls inside the intake manifold. An experimental validation was also carried out.

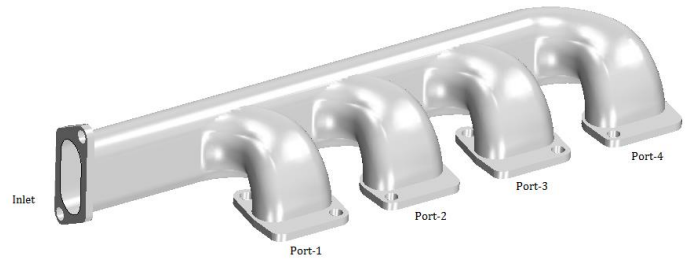

Fig.2 Geometry Details

The engine speed varies based on the engine load and accordingly the mass flow inside cylinder would also change. So, the performance investigation must be carried out various engine operating conditions. Based on the calculations, five flow rates – 4 kg/min, 4.5 kg/min, 5.0 kg/min, 5.5 kg/min and 6.0 kg/min – were considered for this investigation.

The engine firing order (FO) for this engine configuration was 4-1-2-3. So, at an instant, the ports 4 and 2 would be open and ports 1 and 3 would be closed. And the reverse condition shall apply. The following were the possible configurations.

Table 1 Intake Port Configurations

Configurations	Port 1	Port 2	Port 3	Port 4
FO:1-3	Open	Closed	Open	Closed
FO:2-4	Closed	Open	Closed	Open

These configurations were investigated for each engine operating conditions (varying mass flow rate) as mentioned below.

Table 2 Engine mass flow rate

Configuration	Mass flow rate, kg/min
Case A	4.0 kg/min
Case B	4.5 kg/min
Case C	5.0 kg/min
Case D	5.5 kg/min
Case E	6.0 kg/min

So, a total of 10 configurations were investigated in this study.

IV EXPERIMENTAL APPROACH

A blower with a capacity to cover this operating range with a flow control valve was connected with the intake manifold. The pressure sensors were mounted on the surfaces of the manifold surfaces. The quantity of the flow in to the intake manifold was controlled by the flow control valve.

Upon starting the blower, the necessary mass flow rate into the manifold. Steady state operations were ensured prior to note the readings. Multiple attempts were made to ensure the accuracy of the readings in the experimental study.

V CFD SIMULATION APPROACH

The CFD simulations were performed using STAR CCM+ software. The pre-processing activities that were needed for the CFD simulations like geometry clean-up, meshing, applying boundary conditions were completed in the STAR CCM+. The polyhedral mesh element type was chosen for meshing the computational volume of the Intake manifold.

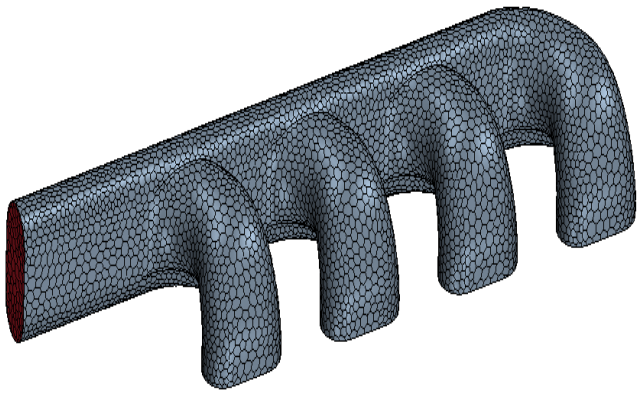

Fig.3 Polyhedral mesh for the Intake Manifold

The 4 layers of prism elements near the manifold wall surfaces were applied to capture the boundary layer effects. The total height of the prism layers was ensured to be higher than the boundary layer thickness.

The Realizable k-epsilon turbulence model with All Y+ Two Layer Wall Treatment option in STAR CCM+ was chosen for the simulations. The flow inlet was modeled using the 'velocity inlet' boundary condition. The intake manifold walls were modeled as no-slip, adiabatic, stationary walls. The iterative approach in the CFD simulations were continued till the mass balance was achieved between the flow inlets and outlets.

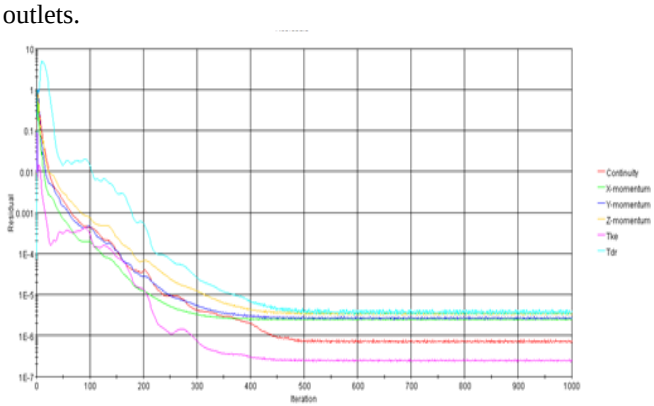

Fig.4 Governing equation residual plot

Also, the total pressure values at the outlets were monitored for deciding the steady state conditions. The wall Y+ values were in the range of 0-5, indicating near wall boundary layer effects had been resolved in the CFD simulations.

VI Validation

A comparative study between the experimental and the CFD simulations was performed in terms of the flow losses. This was done all the 10 configurations. The results were plotted in the following images.

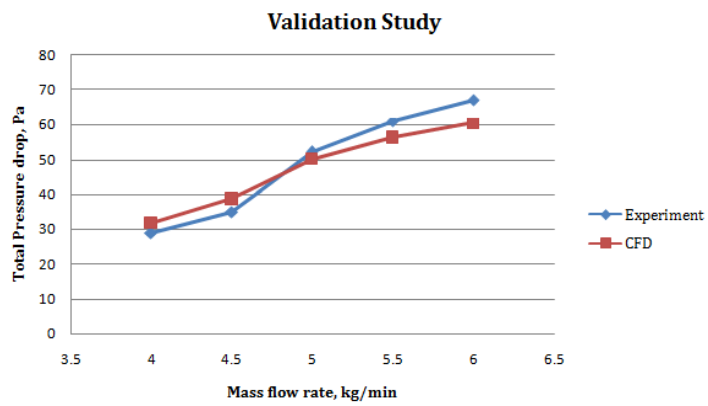

Fig.5 Flow loss comparison between Experiment and CFD Simulations [Firing Order 1-3]

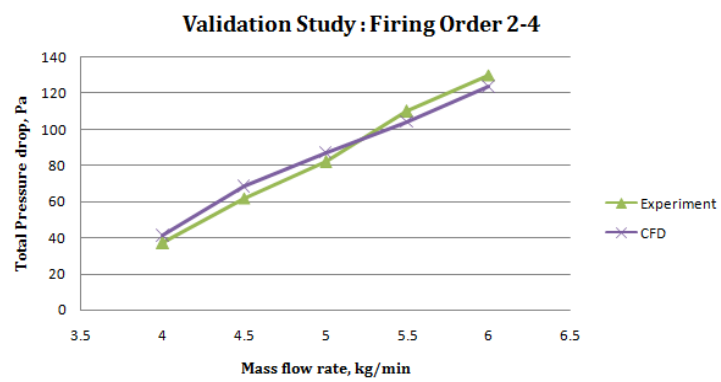

Fig.6 Flow loss comparison between Experiment and CFD Simulations [Firing Order 2-4]

From the above graphs, Fig 5 and Fig 6, the differences between the CFD simulations and the experimental methods could be concluded to be negligible. This serves as validation for the project.

VII RESULTS & ANALYSIS

The flow loss had been defined as the total pressure drop from the inlet to the port outlets. It was expected that the manifold must offer least flow losses for the high engine performance.

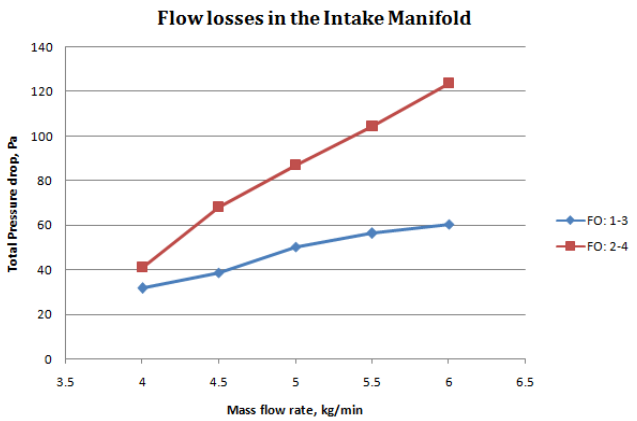

Fig.7 Flow loss comparison between Two Firing Order Configuration

From the above graph, it was observed that the flow path corresponding to Firing Order 2-4 provide significantly higher flow losses – as much as 100% - in comparison to the identical flow conditions of Firing Order 1-3.

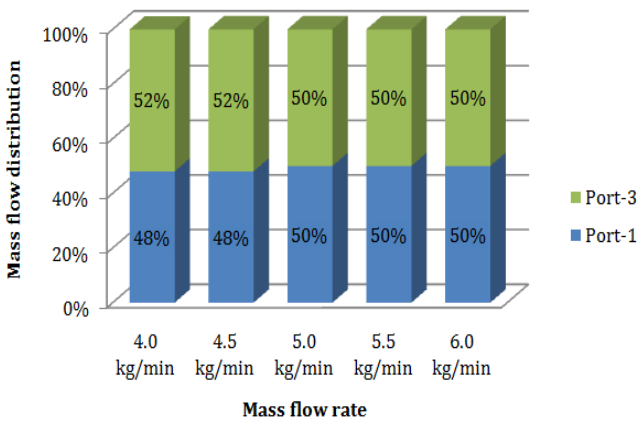

Fig.8 Mass flow distribution for the Firing Order 1-3

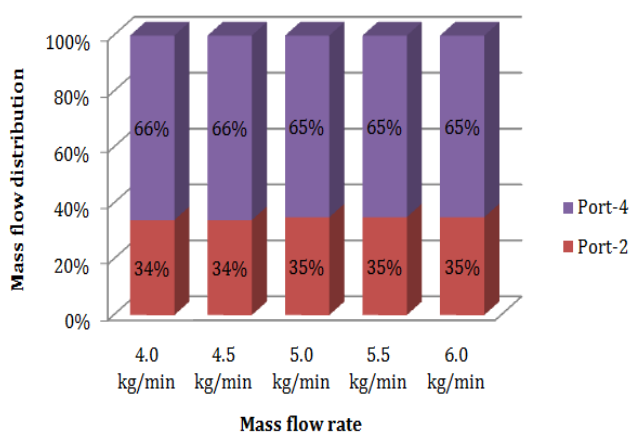

Fig.9 Mass flow distribution for the Firing Order 2-4

In the next analysis, the mass flow distribution among the ports for these two firing-order configurations was studied.
Here too, the flow path of Firing Order 1-3 resulted in even mass flow split. This was observed in all operating conditions i.e. 4 kg/min to 6kg/min.
But, un-even mass flow split was observed for the Firing-Order 2-4 configurations, with Port-2 was receiving lower mass flow rate (35%) as compared to the Port-4 for all operating conditions.

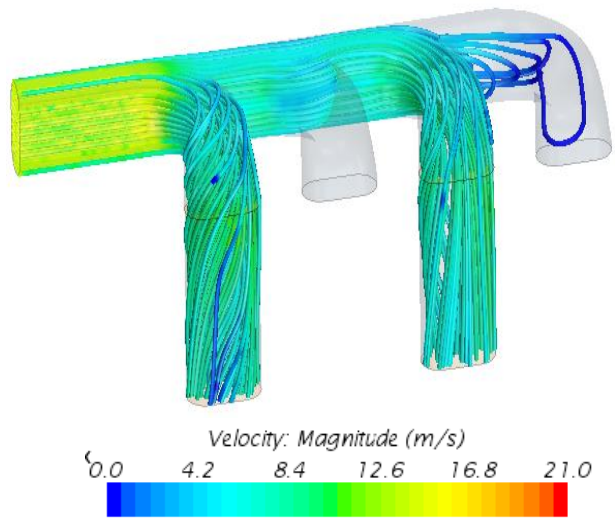

Fig.10 Streamlines for FO:1-3 for the Case A

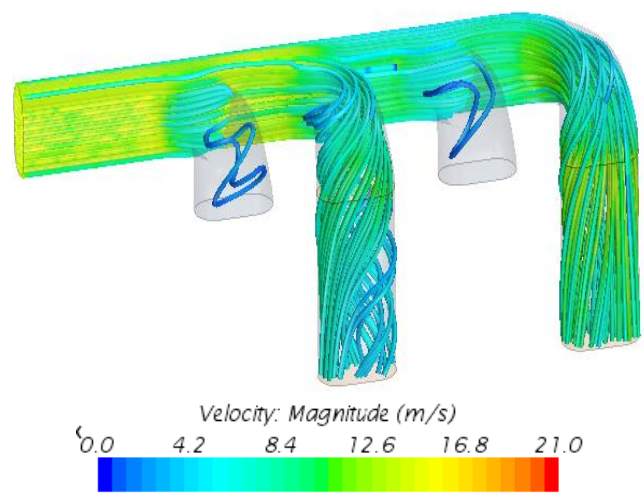

Fig.11 Streamlines for FO:2-4 for the Case A

The streamlines for Case A and D for both the firing order had been provided in images [Fig 10 - 13]. The flow swirl could be observed from these plots. The flow to the Port-2 appears to have the high swirling as compared to the remaining ports. An equally strong swirling in Port-1 could also be noted from these streamline plots.

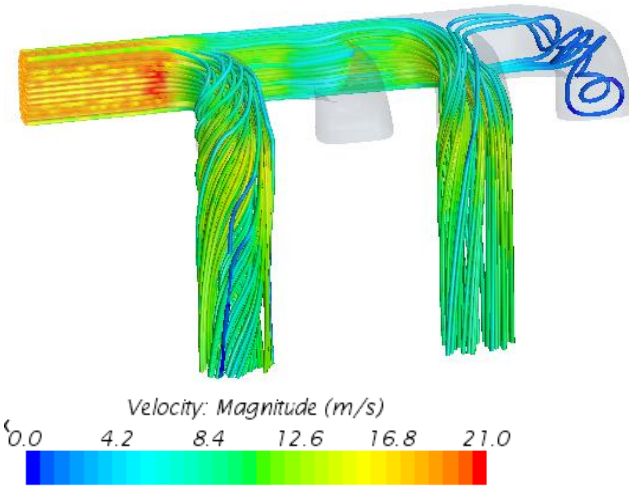

Fig.12 Streamlines for FO: 1-3 for the Case D

The flow swirl at the port inlet could enhance better engine combustion characteristics. Since the identical flow patterns with different velocity magnitude was observed for the remaining cases, those were not presented here.

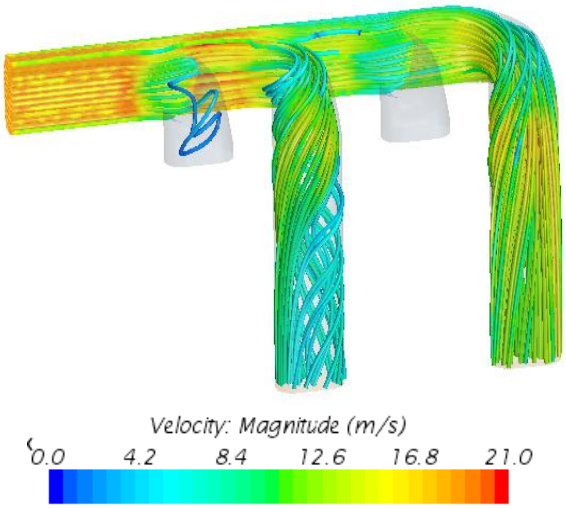

Fig.13 Streamlines for FO: 2-4 for Case D

Velocity contours at the manifold outlet – Inlet ports – for the Case D had been provided. Few low velocity regions had been predicted. This could be result of the high flow swirls.

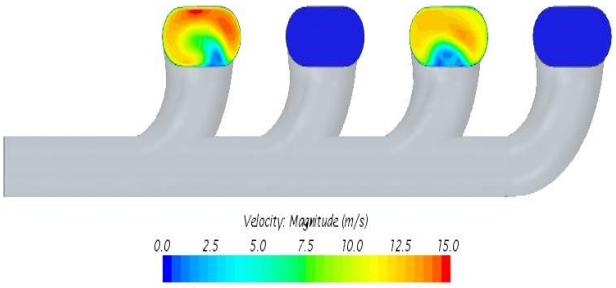

Fig.14 Streamlines for FO: 2-4 for Case D

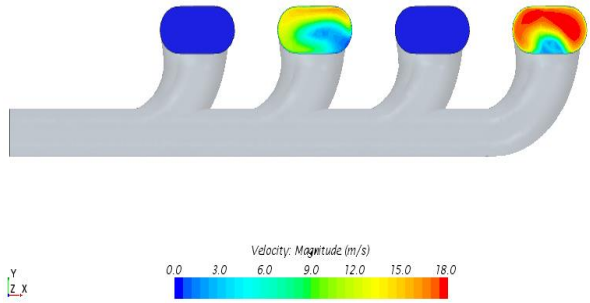

Fig.15 Streamlines for FO: 2-4 for Case D

VIII CONCLUSIONS

Based on these extensive research work, the following were the conclusions

1) A new CFD simulation methodology –in terms of boundary conditions - for the intake manifold was proposed. The results from this approach was in close agreement with the experimental data.

2) The flow path for the Firing Order 2-4 resulted in high flow losses. Also, the un-even mass flow split for the Ports were identified. An optimization would be required for better engine performance

3) However, the flow path for the Firing Order 1-3 provided even mass flow split to the ports.

4) High flow swirl had been noted for all the operating conditions which could enhance the engine combustion characteristics.

REFERENCES

[1] Suresh Aadepu, I.S.N.V.R. Prashant, Jarapala Murali Naik, (2014) "*Design of Intake Manifold of IC Engines with Improved Volumetric Efficiency*" International Journal & Magazine of Engineering, Technology, Management and Research. Vol-1, Issue-6; June 2014;

[2] Y. K. Loong, Salim M. Salim, "*Experimentation and Simulation on the Design of Intake Manifold Port on Engine Performance*" EURECA 2013

[3] M.A. Jemni, G. Kantchev, M. S. Abid (2012) "*Intake Manifold Design Effect on Air Fuel Mixing and Flow for an LPG Heavy Duty Engine*" International Journal of Energy and Environment, Volume-3, Issue 1;

[4] S. A. Sulaiman, S. H. M. Murad, I. Ibrahim and Z. A. Abdul Karim, (2010) "*Study of Flow In Air-Intake System for a Single-Cylinder Go-Kart Engine*" International Journal of Automotive and Mechanical Engineering;

[5] Per Ahlm, Erik Froode, (2011) "*Manifold Design for a Marine Diesel Air Intake System*" Master's Thesis, Department of Applied Mechanics, Chalmers University of Technology, Sweden.

[6] Long Xie, Harutoshi Ogai and Yasuaki Inoue, (2006) "*Modeling And Solving An Engine Intake Manifold with Turbo Charger for Predictive Control*" Asian Journal of Control;

vol-8, No-3;

[7] Negin Maftouni, Reza Ebrahimi (2006) "*Intake Manifold Optimization By Using 3-D CFD Analysis With Observing the Effect of Length of Runners On Volumetric Efficiency*" proceedings of the 3rd BSME-ASME International Conference on Thermal Engineering; Dhaka, Bangaldesh.

[8] Rajesh Holkar, Yogesh N Sule-Patil, Somnath M Pise, Yogesh A Godase, Vishal Satish Jagadale, (2015) "*Numerical Simulation of Steady Flow through Engine Intake System using CFD*" IOSR Journal of Mechanical and Civil Engineering; vol-12, issue 1, ver.II.

[9] S. Karthikeyan, R. Hariganesh, M.Sathyanadan, S. Krishnan, P. Vadivel, D. Vamsidhar, (2011) "*Computational Analysis of Intake Manifold Design and Experimental Investigation on Diesel Engine LCV*" International Journal of Engineering Science and Technology; vol-3, No-4

[10] P. Ragupathi, S. Rajkumar, R. Sameer Hussain, (2014) "*Design and Analysis of Diesel Engine Intake Manifold using CFD Simulation*" International Journal of Modern Trends in Engineering and Science; vol-1, issue-1

[11] M.M. Khan, S. M. Salim (2013) "*Evaluation of CFD sun-models for the Intake Manifold Port Flow Analysis*" EURECA 2013;

[12] D. Ramasamy, M. Zamri, S. Mahendran, S. Vijayan (2010) "*Design Optimization of Air Intake System (AIS) of 1.6L Engine by Adding Guide Vane*" Proceedings of the International Multi-Conference of Engineers and Computer Scientists; Hong Kong 2010; vol-II

[13] G. Venkatta Punna Rao, M. Vimal Teja, (2012) "*Finite Element Analysis of Automotive Intake Manifold using CAE Software*" International eJournal of Mathematics and Engineering;

[14] Amit Kumar Gupta, Abhishek Mishra, (2014) "Design and Development of Inlet Manifold for Six Cyinder Engine for Truck Application" Indian Journal of Research; Volume: 3, Issue: 7;