

CFD Analysis of 4 Cylinder Intake Manifold using STAR CCM+

Lokesh R. Dhumne¹, Aniket S. Kulkarni², Raghavendra G. Daramwar³, Shruti Vedpathak⁴

^{1,2,3,4}Assistant Professor, Department of Mechanical Engineering, Dr. D Y Patil Institute of Technology, Pimpri, Pune, Maharashtra, India

Abstract - This paper deals with the improvement of the performance and to achieve the even distribution of flow at each cylinders, to select the best turbulence model for the analysis of intake manifold using computational fluid dynamics, to achieve the maximum mass flow rate through the runners, to maintain the equal pressure throughout the plenum, to propagate back the higher pressure column of air to intake port within the duration of the intake valve's closure. 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 - 2.2kg/min, 4.5 kg/min, 5.0 kg/min, 5.5 kg/min and 6.0 kg/min. In this analysis plenum size is kept constant to show the effects of shape of runners on output. 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.

Key Words: Intake Manifold, Cylinder, Volumetric Efficiency, Computational Fluid Dynamics, Star CCM+.

1. INTRODUCTION

The main purpose of the intake manifold is to evenly distribute the combustion mixture to each intake port of the engine cylinder, unless the engine has direct injection [1]. Even distribution is important to optimize the volumetric efficiency and performance of the engine. There are various factors that influence the engine performance such as compression ratio, atomization of fuel, fuel injection pressure, and quality of fuel, combustion rate, air fuel ratio, intake temperature and pressure and also based on piston design, inlet manifold, and combustion chamber designs etc.

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.

2. 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. 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.

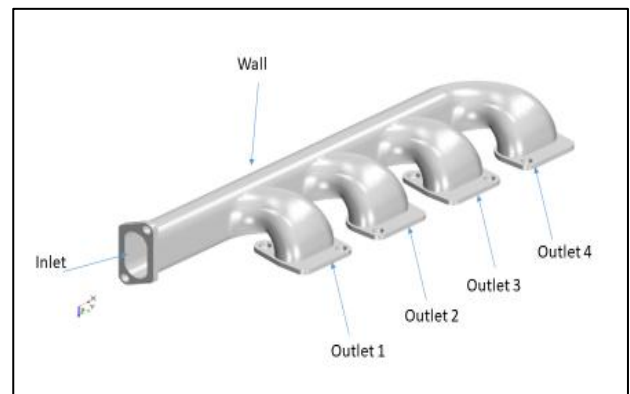


Fig.1: Geometry Details

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

Configuration	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
Case A	4.0 kg/min
Case B	4.5 kg/min
Case C	5.0 kg/min

3. 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.

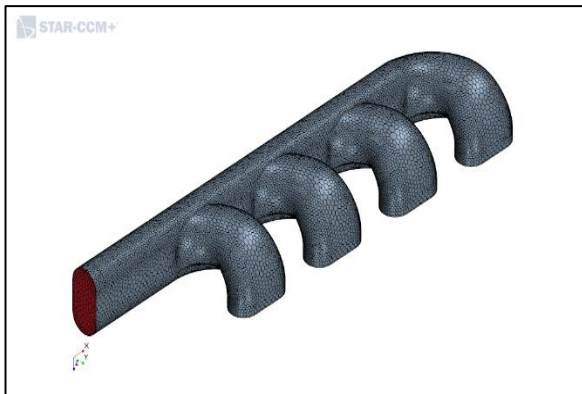


Fig.2: Intake Manifold Wrapped and Remeshed

To obtain computational volume, the inlet and outlets are closed in surface repair and then it is wrapped. To get better meshing, it is remeshed. To avoid the reversed flow at outlet, all outlets are extruded keeping outlet boundary condition same. The remeshed condition is shown in Fig. 2 and extrusion is shown in Fig. 3.

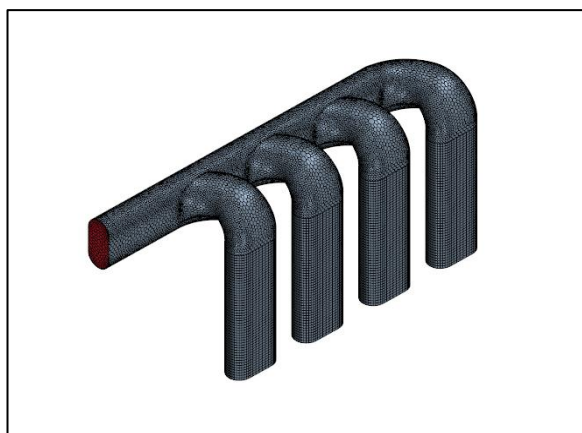


Fig.3: Intake Manifold Remeshed with Extrusion

The 3 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.

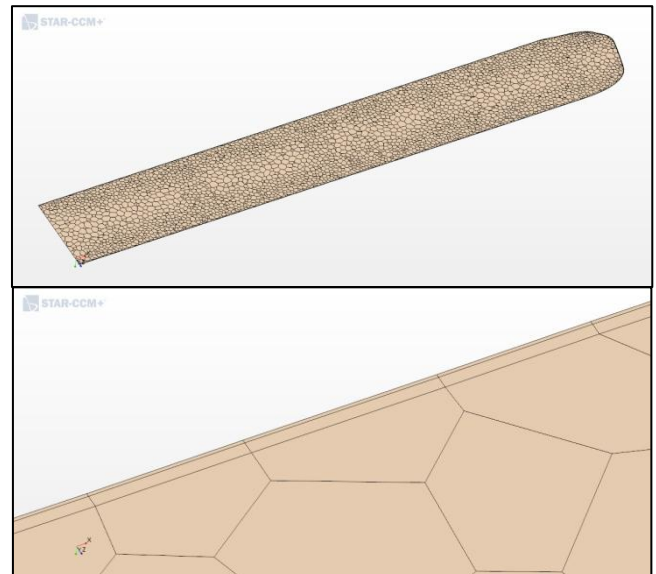


Fig.4: Sectional view Normal to X Axis and magnified view showing prism layers at boundary

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.

4. BOUNDARY CONDITIONS:

Boundary conditions for simulation are shown in Fig 5. The wall temperature is kept 65°C and temperature at inlet is 25°C. Ambient conditions are kept at NTP ie atmospheric pressure is 0 Pa gauge and temperature is 25°C.

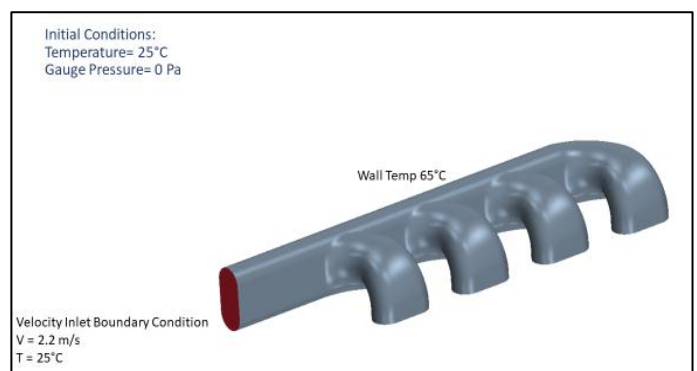


Fig.5: Boundary Condition

6. Solver Settings for simulation

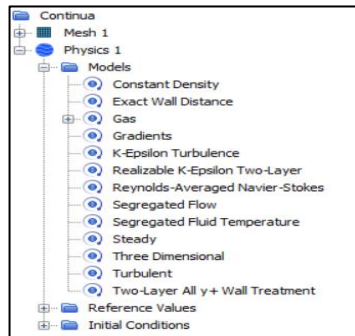


Fig.6: Solver Settings

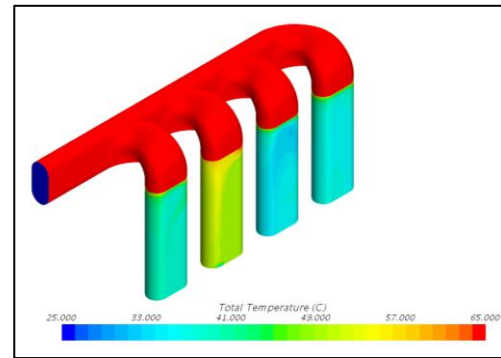


Fig.8: Temperature Color Plot

7. Convergence Criteria

Simulation is solved for 1000 iterations. Converged the solution when very low value of Energy balance line is observed which looks quite steady after 800 iterations

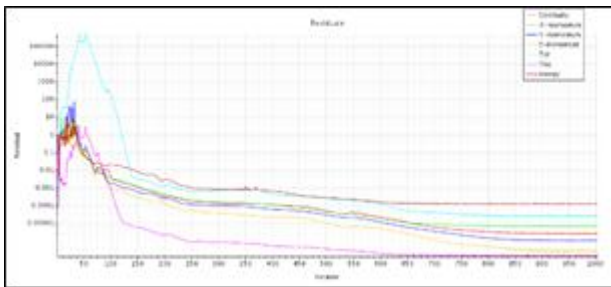


Fig.7: Residual Plot

8. Result & Discussion

Mass balance is obtained as

Mass Flow at Inlet = -1.026575×10^{-2} kg/s

Mass Flow at

Outlet 1= 1.714206×10^{-3} kg/s

Outlet 2= 4.116282×10^{-4} kg/s

Outlet 3= 3.772219×10^{-3} kg/s

Outlet 4= 4.367695×10^{-3} kg/s

Hence, Total Mass Flow at Outlet = 1.026575×10^{-2} kg/s

Temperature variations of fluid at various Outlet are shown in Table 3 and can be viewed in Fig. 8.

Table 3: Temperature variations of fluid at various Outlet

Part	Value(°C)
Outlet1	38.60857
Outlet2	49.9570
Outlet3	35.47663
Outlet4	37.32943
Total	37.89819

Pressure variations of fluid at various Outlet are shown in Table 4 and can be viewed in Fig. 9.

Table 4 Pressure variations of fluid at various Outlet

Part	Value(Pa)
Outlet1	0.4429775
Outlet2	0.0474973
Outlet3	0.9264718
Outlet4	0.8730748
Total	0.7516122

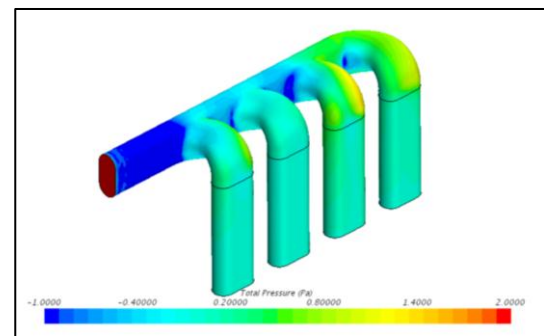


Fig.9: Pressure Color Plot

5. 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.

7. REFERENCES

- [1] Suresh Aadepe, 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, Bangladesh.
- [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 Cylinder Engine for Truck Application" Indian Journal of Research; Volume: 3, Issue: 7