# Design, Analysis and Development of 4-Cylinder IC Engine Exhaust Manifold

Nikhil Kanawade, Omkar Siras

***Abstract*—Exhaust manifold of an IC Engine has a crucial function of removing the exhaust gases from the engine cylinders effectively and efficiently. Computer based design and analysis methods had been extensively used for designing the exhaust manifolds. In this work, the exhaust manifold for the 4 cylinder IC Engine was developed based on the empirical co-relations. Then, a three step method was adopted for optimizing the manifold geometry. The study was conducted using the CFD simulations and the experimental approach. The flow pattern inside the manifold was studied using the streamlines and the necessary optimization was performed. STAR CCM+ was used for performing the CFD simulations. The Realizable k-epsilon turbulence model with All Y+ Wall Treatment was applied for the CFD simulations. An experimental study was conducted for the optimized geometry as part of the project validation. The optimized exhaust manifold provided ~26% less flow losses as compared to the base model.**

***Index Terms*— Manifold, Optimization, Exhaust Manifold, CFD simulations**

## I. INTRODUCTION

The exhaust manifolds of the IC engine collect the exhaust gases at the end of the combustion stroke and discharge to the atmosphere. The design of exhaust manifold plays a crucial role in the engine efficiency, combustion characteristics.

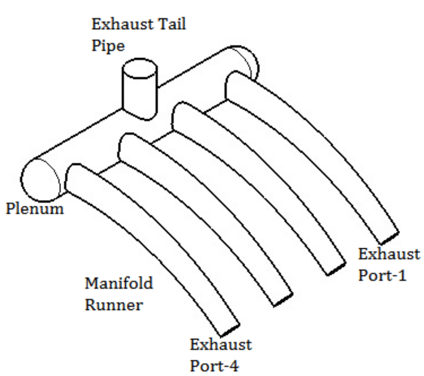


Fig.1 Components of Exhaust Manifold

The exhaust manifold would contain three major components, namely the manifold runner, plenum and the exhaust tail pipe. The manifold runners were connected to the engine exhaust ports at one end and the plenum at the other end. They collect the gases from the engine cylinders and deposit in the plenum. This flow path must offer least flow losses to have high efficient gas removal. In the plenum, there would be flow physics such as flow swirling, stagnation zones, re-circulation zones. These may not be avoided considering the nature of the flow however; these effects must be minimized for the efficient operation. The exhaust tail pipe delivers the gases from the plenum to the exhaust system (catalytic converter). So, the design of exhaust manifold must take in to consideration of these components.

This research work makes an attempt to develop an optimizing method for the exhaust manifold using the computational fluid dynamics (CFD) methods and by the experimental approach as well.

## II LITERATURE REVIEW

Jafar M Hassan [1] had analyzed the performance of the manifolds with a tapered longitudinal section. Authors had used the numerical simulations (CFD) for this research work. The results were analyzed in terms of uniformity coefficient. M.Usan [2] had applied a multi-disciplinary optimization approach for the exhaust system, exhaust manifold and catalytic converter, in highly integrated concurrent engineering software framework. HessamedinNaemi [3] had employed numerical simulations (CFD methods) for estimating the flow loss coefficient in manifolds. The results from different turbulence models – standard k-ε, standard k-ω, SpalartAllmaras model and RNG k-ε model – were compared in terms of flow loss coefficient against the experimental data. Based on their results, the authors had observed that the RNG k- ε turbulence model predictions were in close agreement with the experimental data.The design of exhaust manifold for a 4-stroke high power medium –speed diesel engine was carried out by Kyung-Sang Cho [4]. The typical operational range of the medium-speed diesel engine was in the range of 700 – 1500 rpm and has power outputs up to 6000 kW. Masahiro Kanazaki [5] had developed a multi-objective optimization method for the exhaust manifold by using Divided Range Multi-objective Genetic Algorithm. The three-dimensional fluid dynamics inside the manifold was simulated using transient, Euler flow solver. Hong Han-Chi [6] had used GT-Power, 1-dimensional software, for estimating the engine performance of a single cylinder IC engine. In their study, the authors had considered four parameters – the sphere style plenum diameter, the intake runner diameter, the exhaust runner lengths and the position of restrictor. The plenum for

Nikhil Kanawade, is pursuing M.E. from Flora Institute of Technology, Pune India.
Omkar Siras, is working as Professor at Flora Institute of Technology, Pune India.

the intake and exhaust manifold was designed using Helmholtz theory. In the research work carried out using numerical simulations (CFD), K. S. Umesh [7] had investigated the exhaust manifold performance for eight variants in terms of back pressure, exhaust velocity e.t.c. The flow conditions were varied from 2 kg to 12 kg, for every 2 kg, in their studies. The manifold configuration studied by the authors were Short Bend Center Exit (SBCE), Short Bend Side Exit (SBSE), Long Bend Center Exit (LBCE), Long Bend Side Exit (LBSE), Short Bend Center Exit with Reducer (SBCER), Short Bend Side Exit with Reducer (SBSER), Long Bend Center Exit with Reducer (LBCER), Long Bend Side Exit with Reducer(LBSER). These configurations were based on the location of the manifold pipe and the radius of the bend. Based on their results, the authors conclude that the Long Bend Center Exit (LBCE) configuration provide better performance.

XueyuanZhang [8] had conducted coupled thermo-fluid-solid analysis of an exhaust system with the consideration of welding stresses. The operating condition of 302 kg/hour flow rate of exhaust gas at 870 C was considered by the authors. Simon Martinez-Martinez [9] had performed CFD analysis to estimate the performance of the exhaust manifold while placing the catalytic converter near to it (Close-Coupled Catalytic Converter). They had considered three types of manifold – Cast manifold, 4-2-1 manifold, L-Shaped manifold. In their research work, S. N. Ch. Dattu. V [10] had performed thermal analysis for the tubular type IC-Engine exhaust manifold for various operating conditions. The authors had also considered four different manifolds – Radius 48 mm Exhaust Valve at Extremely Left, Radius 48 mm Exhaust Valve at Center, Radius 100 mm Exhaust Valve at Extremely Left, Radius 100 mm Exhaust Valve at Center. Benny Paul [11] had conducted CFD simulations on manifold of direct injection diesel engine. They had used the RANS (Reynolds Averaged Navier Stokes) solver approach with RNG k-ε turbulence model for the simulations. The various factors to be considered while designing an exhaust manifold were discussed by Gopaal [12]. They note that the smaller tubes offer higher flow resistance and thus the engine power will be needed to push the exhaust gases.

MohdSajidAhmed [13] had applied CFD methods to identify the optimum exhaust manifold for a 4-stroke 4-cylinder SI engine. They had considered five variants of exhaust manifold, based on the manifold pipe geometry, - convergent inlet pipe, divergent-straight-convergent, reduced convergent length and increased divergent length, reduced divergent length and increased convergent length, identical convergent and divergent and reduced straight length. The CFD simulations were performed using ANSYS FLUENT with un-structured meshes. I.P. Kandylas [14] had developed an exhaust system heat transfer model that included the steady state and transient heat conduction as well as convection and radiation.

## III PROBLEM DESCRIPTION

The project work dealt with designing the exhaust manifold for the 4-cylinder IC engine. Based on the literature, the exhaust port diameter had been chosen as 30 mm[7].

Since the exhaust manifold would have to deliver high performance for varying operating conditions, the design calculations were performed for multiple engine speeds. The engine speeds of 1200 RPM, 1500 RPM, and 1800 RPM were considered for designing the exhaust manifold. The exhaust gas flow rate from John B Heywood[16] for these speeds was listed below.

Table 1 Exhaust gas flow for various engine speeds

| Engine Speed, RPM | Exhaust gas, g/s |
|---|---|
| 1200 | 42 |
| 1500 | 44 |
| 1800 | 45 |

The optimization for the manifold was performed for the runners, the plenum and the exhaust pipe. Each of these components was optimized with different constraints as explained in the following sections. Initial studies were performed using CFD simulations while the validation for the project was carried using the experimental approach for the optimized exhaust manifold geometry.

The flow losses, defined in terms of Total Pressure drop from exhaust port to exhaust pipe outlet, and the flow streamlines inside each manifold configurations were analyzed for deciding the optimized shape.

## IV DESIGN CALCULATIONS

Typical engine parameters as provided in Heinz Heisler[17] were listed in Table 2.

Table 2Engine design parameters

| | | | |
|---|---|---|---|
| Piston Diameter | D | mm | 80 |
| port diameter | d | mm | 30 |
| piston stroke | S | mm | 85 |
| Crankshaft speed | N | rpm | 1800 |

Now, the mean piston speed was calculated based on the following relation

$$V_p = \frac{2\,S\,N}{60 * 1000}\ m/s$$

The manifold length (L) was calculated [17] as

$$L = \frac{\theta_t\,C}{0.012 * N}\ m$$

Where, C is the speed of sound and $\theta_t$ was the crankshaft angular movement. The speed of sound for an exhaust temperature of 400° C was 518 m/s.

So, the manifold length was calculated to be ~2887 mm. The CAD model based on these dimension was built using CATIA software.

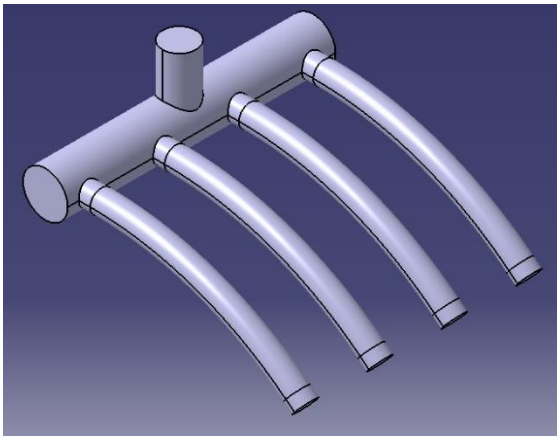

Fig.2 CAD Model of Exhaust Manifold

## V ASSUMPTIONS & LIMITATIONS

1) The flow field inside the manifold could be time-dependent and instantaneous. However, considering the scope of the project, Steady-state approach had been followed.
2) The exhaust gases would typically be of multi-component mixture of combustion products. The material property under these scenarios would be of highly anisotropic. This study assumes single component and the averaged iso-tropic material properties were applied during the simulations.
3) The exhaust gases will typically be at much higher temperature – in the range of 800 C. These high temperature gases would results in thermal stresses and related physics. Since the focus of the project was to optimize the flow path, reduce the flow losses and reduce the back pressure, the thermal effects had not been included in this study.

## VI EXPERIMENTAL APPROACH

The experimental study for the exhaust manifold was conducted using a high capacity blower that can supply air at 45 g/s flow rate. This approach was followed rather than fitting the exhaust manifold in to an actual IC engine because of the availability of the IC engine as well as associated cost for execution.

The flow rate was controlled by the flow valve and was monitored by the anemometer. The pressure values at the Port Inlets and the Tail Pipe Outlet was monitored after ensuring the steady state conditions were obtained.

## VII CFD SIMULATION APPROACH

The commercially available STAR CCM+ software was used for performing the 3-dimensional steady state CFD simulations for this project work.

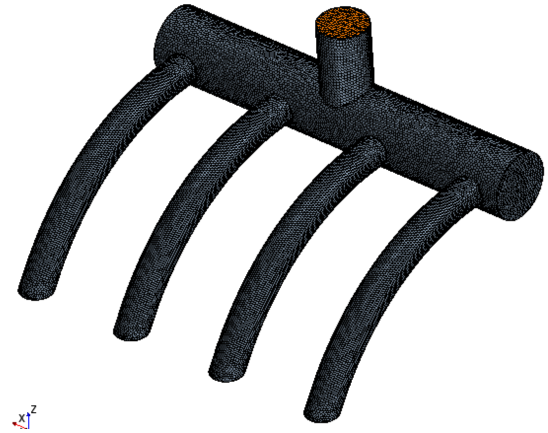

Fig.3 Polyhedral mesh for the Exhaust Manifold

The flow filed inside the manifold was expected to be highly turbulent and the boundary layers will have dominating influence. In order to resolve these flow gradients, prism layers elements near the manifold walls were generated. Overall there were ~300,000 mesh elements in the model.

Reynolds Averaged Navier Stokes (RANS) approach was used with Realizable k-epsilon turbulence model was chosen.

During the engine operation, only one exhaust port would be opened and the remaining ports would be closed. This condition was reproduced by applying the inlet boundary condition for a port and wall boundary condition for the remaining exhaust ports.

Most research had considered [7] all the exhaust port to be opened all the time. However, the approach followed in this study resembled to the actual conditions and was expected to provide accurate results. The simulation convergence based on the mass balance between the flow inlets and the outlets as well the obtaining steady values of Total Pressure at outlet.

## VIII OPTIMIZATION

The design optimization for the manifold was carried out in three steps. The CFD simulations were utilized for conducting the optimization study.

### VIII.1 Phase 1) Runner Optimization

In the first phase, the manifold runner was optimized by considering the following parameters.

Variant 1) Base Model with uniform cross-sectional area (Fig 2)

Variant 2) Providing Positive Taper on the Runners and

Variant 3) Providing the Negative Taper on the Runners

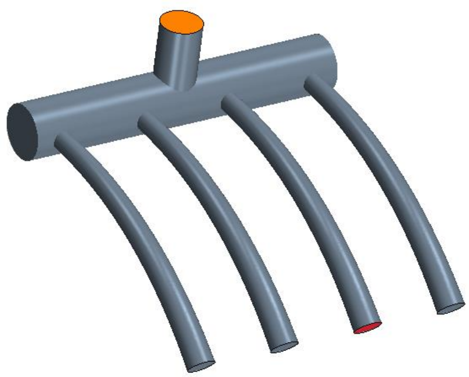

Fig.4 Variant-2 Exhaust Manifold

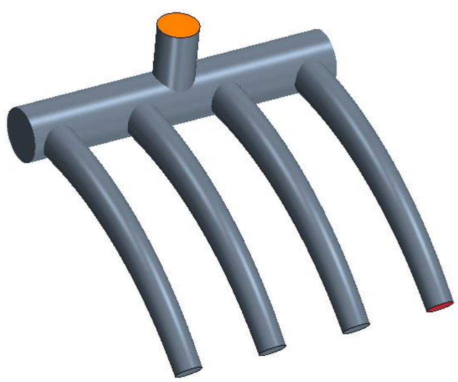

Fig.5Variant-3 Exhaust Manifold

In variant-2, the runner diameter was varied 30 mm from the port to 20 mm to the plenum side, while the variant-3 had 40 mm diameter on the plenum side. The CFD simulation modeling as explained earlier was followed.

The fluid flow pattern when the exhaust port-4 open would be identical. So, only two simulations for the exhaust port-1 and exhaust port-2 at open condition was performed. The streamlines from these simulations were provided in Figure.

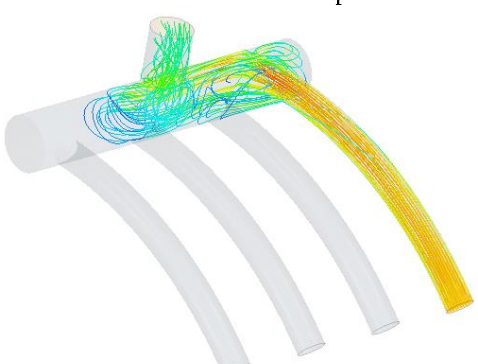

Fig.6 Streamlines for Exhaust Port-1 Open [variant 1]

From the streamlines, the high swirling flow in the plenum while the exhaust port-1 open could be noted. The swirl flow pattern considerably reduces for the exhaust port-2 open configuration. There is however, flow re-circulation in the plenum.

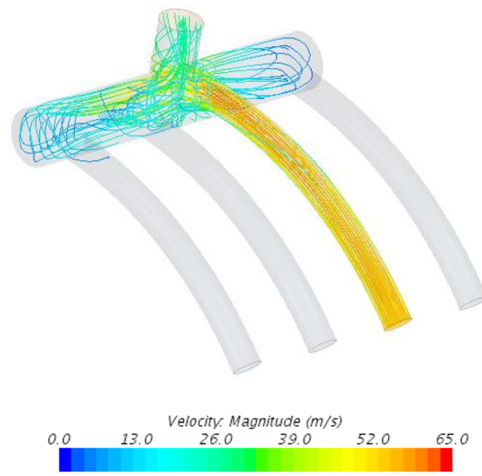


Fig.7 Streamlines for Exhaust Port-2 Open [variant 1]

The pressure drop for each flow path had also been compared in Figure 8.

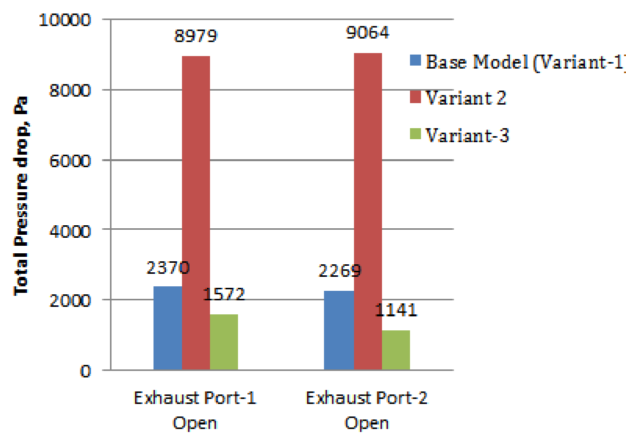


Fig.8 Flow loss comparison

Based on the streamlines and the pressure drop values, variant-3 which had 'negative taper on the runner' was chosen as the optimized runner shape.

VIII.2 Phase II) Plenum Optimization

In the second phase, two shapes for the plenum were considered for the optimization. The objective for this was to reduce the flow swirl in the plenum.

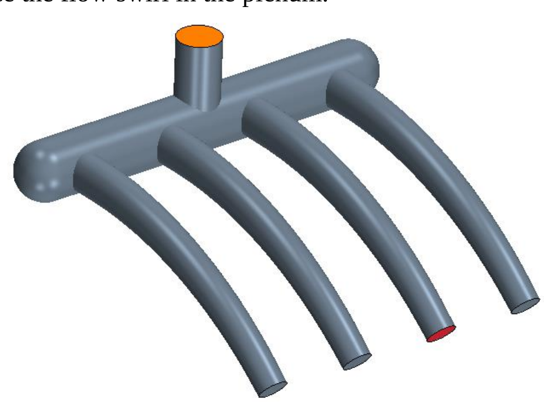

Fig.9 Plenum Modified with Cap [Variant-4]

Variant-4) circular cap at both ends (Figure 9)
Variant-5) elliptical plenum (Figure 10)

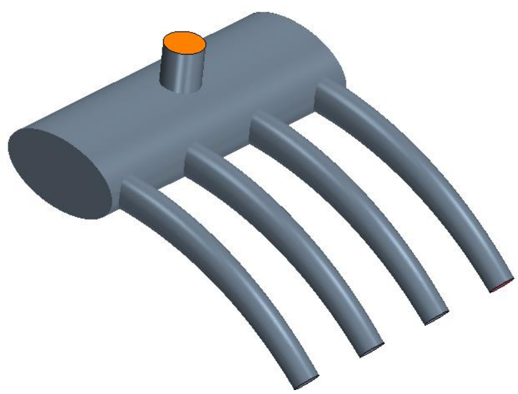

Fig.10 Exhaust Manifold with Elliptical Plenum [Variant-5]

For these simulations, the runner from the Variant-3 geometry was used. This was because of the less flow losses that were observed in Phase I. The flow streamlines, coloured with flow velocity, for both these configurations had provided below.

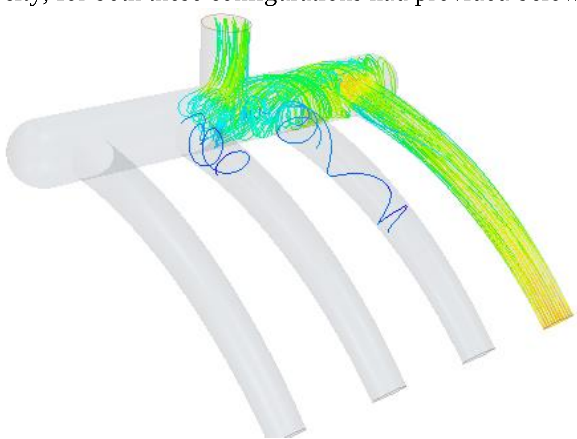

Fig.11 Streamlines for Exhaust Manifold with Modified Plenum Cap [Variant-4]

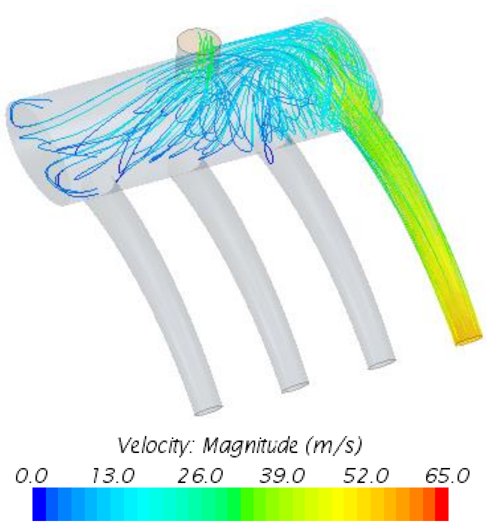


Fig.12 Streamlines for Exhaust Manifold Elliptical Plenum [Variant-5]

For the Variant-4, high swirling was observed in the plenum (Fig 11). The circular cross section of the plenum also assists the flow to obtain higher swirl.
In the case of Variant-5, even though flow swirl was observed in the plenum, it was much less severe than the variant 4. The elliptical shape plenum provides more length for the flow to move around as compared to a very narrow passage offered by the circular cross-section in Variant-4.

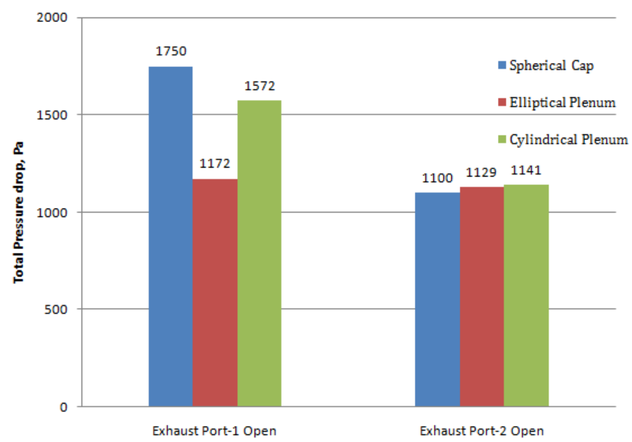


Fig.13 Flow loss comparison for various plenum geometries

Based on these, the elliptical plenum which provided lesser flow swirl had been chosen as the optimized shape.

VIII.3 Phase III) Exhaust Tail Pipe Optimization

In the final phase of the study, the exhaust pipe of the assembly was optimized by introducing taper. By providing the positive taper, the exhaust pipe had become a converging nozzle, which helped in accelerating the flow. This flow acceleration would ensure the faster removal of the exhaust gases from the engine cylinders.
The taper angle that was considered was 2.5°, 5° and 7.5°. The optimized geometry for the manifold runner and the plenum were taken and the exhaust tail pipe with the respective taper angle was attached, Figures 14 – 16.

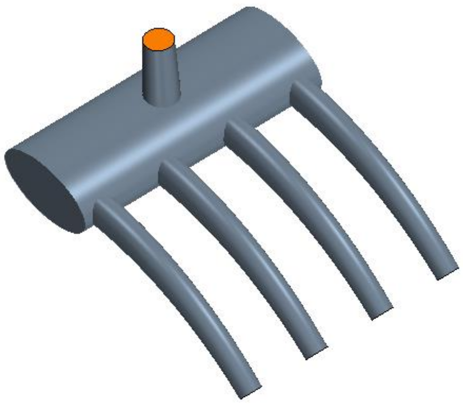

Fig.14 Exhaust Tail Pipe with 2.5 degrees

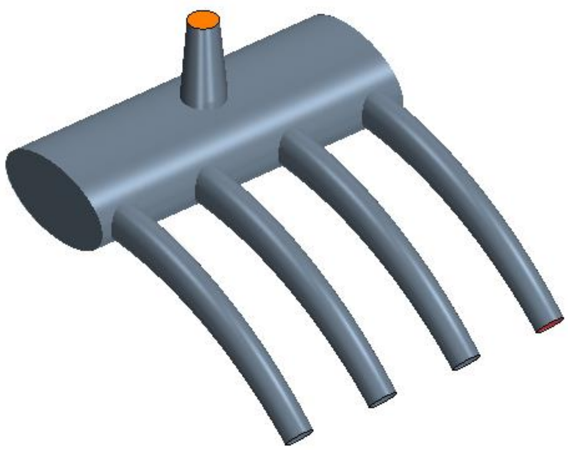

Fig.15 Exhaust Tail Pipe with 5.0 degrees

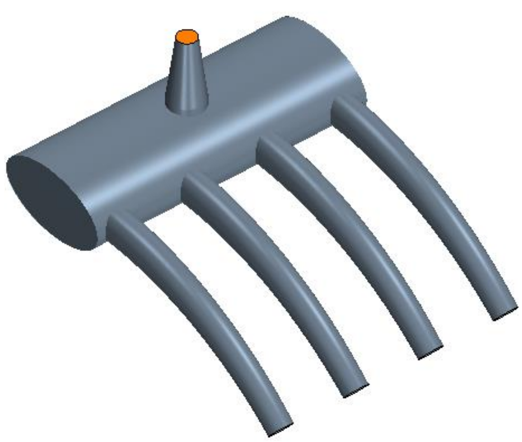

Fig.16 Exhaust Tail Pipe with 7.5 degrees

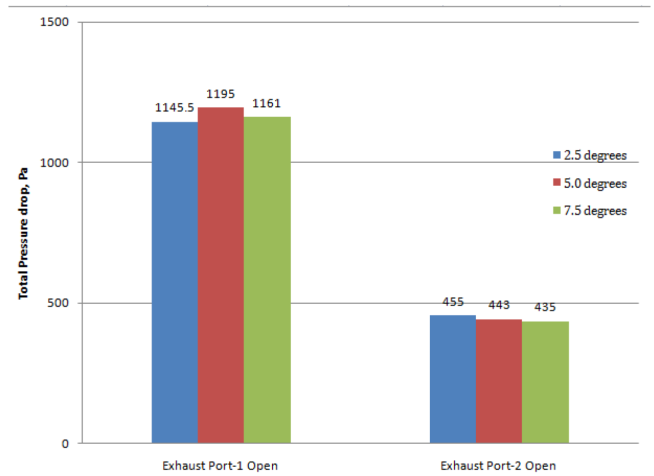


Fig.17 Flow loss comparisons for various exhaust tail pipe geometries

From the graph on Figure 17, its quite evident that the exhaust tail pipe with the taper angle of 2.5° provide least flow loss while the exhaust port-1 was open. However, when the exhaust port-2 was open, the tail pipe with 7.5° was providing least flow losses. Considering the higher flow losses observed in the flow path of exhaust port-1 open variants, its been suggested to select the tail pipe with 2.5° as the optimized shape. With this, the optimization of the exhaust manifold was completed.

## IX VALIDATION

The optimized exhaust manifold geometry was manufactured. The experimental procedure as explained in the earlier section was followed to obtain the pressure drop across the manifold. The Fig 17 provides the comparison between the Experimental methods and the CFD simulations.

Table 3 Comparison of flow losses

| Flow losses, Pa | | |
|---|---|---|
| | Experiment | CFD |
| Exhaust Port-1 Open | 1060 | 1161 |
| Exhaust Port-2 Open | 405 | 435 |

Since the difference between the results from these methods were negligible (~10%), this can be considered as the necessary validation for the project work.

## X CONCLUSIONS

Based on the work carried out, the following conclusions can be drawn

1) The three step method of optimizing the exhaust manifold provided favorable results.

2) Computer based simulation approach provided results that were in good agreement with the experimental studies.

3) The flow path for the exhaust port located on the extreme corners was having higher flow losses. This was observed for all plenum configurations. And, it could be attributed to the flow swirl in the plenum.

## XI REFERENCES

[1] Jafar M Hasan, Wahid S Mohammad, Thamer A Mohamed, Wissam H Alawee, "CFD Simulation for Manifold with Tapered Longitudinal Section" International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 2, February 2014

[2] M. Usan, O. de Weck, D. Whitney, "Exhaust System Manifold Development Enhancement through Multi-Attribute System Design Optimization", American Institute of Aeronautics and Astronautics;

[3] HessamedinNaeimi, DavoodDomiryGanji, MofidGorji, GhasenJavadirad and MojtabaKeshavarz, "A Parametric Design of Compact Exhaust Manifold Junction in Heavy Duty Diesel Engine Using Computational Fluid Dynamics Codes" Thermal Science, Volume-15, No. 4, 2011;

[4] Kyuang-Sang Cho, Kyung-Bin Son, Ue-Kan Kim, "Design of Exhaust Manifold for Pulse Converters Considering Fatigue Strength due to Vibration", Journal of the Korean Society of Marine Engineering, vol-37, No.7, 2013;

[5] Masahiro Kanazaki, Masashi Morikawa, Shigeru Obayashi and Kazuhiro Nakahashi, "Exhaust Manifold Design for a Car Engine Based on Engine Cycle Simulation" International Conference Parallel Computational Fluid Dynamics, Japan, May 2002;

[6] Hong Han-Chi, Huang Hong-Wu, Bai Yi-Jie, "Optimization of Intake and Exhaust System for FSAE Car Based on Orthogonal Array Testing" International Journal of Engineering and Technology, Volume 2, No. 3, March 2012;

[7] K.S. Umesh, V.K. Pravin, K. Rajagopal, "CFD Analysis of Exhaust Manifold of Multi-Cylinder SI Engine to Determine Optimal Geometry for Reducing Emissions" International Journal of Automobile Engineering Research and Development, Vol-3, Issue 4, October 2013;

[8] Xueyuan Zhang, Yu Luo, Jianhua Wang, "Coupled Thermo-Fluid-Solid Analysis of Engine Exhaust Manifold Considering Welding Residual Stresses" Transactions of JWRI, 2011L;

[9] Simon Martinez-Martinez, Ruben D. Leal-Garza, Fausto A. Sanchez-Cruz, Esteban Baez Villareal, "CFD Analysis of the effect of the Exhaust Manifold Design on the Close-Coupled Catalytic Converter Performance" Journal of KONES Powertrain and Transport, vol-17, No.4, 2010;

[10] S. N. Ch. Dattu. V, M. PradeepVarma, B. ShyamSundar, "Thermal Analysis on 4-1 Tubular Type IC-Engine Exhaust Manifold through Analysis" International Journal of Advanced Mechanical Engineering, volume-4, Number 7, 2014;

[11] Benny Paul, V. Ganesan, "Flow field Development in a Direct Injection Diesel Engine with Different Manifolds" International Journal of Engineering, Science and Technology, Vol-2, No.1, 2010;

[12] Gopaal, M.M.M Kumara Varma, L. Suresh Kumar, "Exhaust Manifold Design – FEA Approach" International Journal of Engineering Trends and Technology, Volume-17, Number 10, November 2014;

[13] MohdSajid Ahmed, Kailash B.A., Gowreesh,"Design and Analysis of a Multi-Cylinder Four Stroke SI Engine Exhaust Manifold Using CFD Technique" International Research Journal of Engineering and Technology, volume-2, Issue-9, December 2015;

[14] I.P. Kandylas, A.M. Stamatelos,"Engine Exhaust System Design based on Heat Transfer Computation" Energy Conservation & Management, 1999;

[15] Damien Kennedy, Gerry Woods, Darragh Forrest, "Development of a New Air Intake and Exhaust System for a Single Seat Race Car" Proceedings of the ITRN, September 2011;

[16] John B Heywood, (2011) "Internal Combustion Engine Fundamentals" Tata McGraw Hill Publications

[17] Heinz Heisler (2002) "Advanced Engine Technology" Butterworth-Heinemann Publications