Numerical Simulation of Novel Nozzle Intake Manifold

Samyak Jain and Ranjit S Patil
Department of Mechanical Engineering, BITS-Pilani – K K Birla Goa Campus, Zuarinagar -403726, Goa, India

*Email: ranjitp@goa.bits-pilani.ac.in

Abstract. The design of automobile intake plays a vital role in the performance of the engine. Any abrupt change in the contour of the intake manifold affects the performance and emission characteristics of the engine. The objective for this research work is to study the flow through restrictor, plenum, and runner for a normally aspirated engine for motorsports application. Restrictor placed in the intake path to achieve the desired value of peak power and torque. In this study, a 20 mm restrictor is used before plenum and runner in compliance with FSAE competition rules conducted by SAE India. In the present study, two novel models of the intake manifold are considered with different plenum geometries. The geometries are modeled using SOLIDWORKS 2014. Meshing is done using ICEM CFD 15.0 and CFD simulations are done using ANSYS- Fluent 15.0. The K-epsilon (RNG) Model is used for simulation with scalable wall function. The two proposed novel intake manifolds were compared on the basis of static pressure, velocity magnitude, total pressure, and turbulence kinetic energy to decide the better model. The effect of the restricted path on the engine efficiency was also analyzed for the better intake manifold model.

1. Introduction
The air flow through the manifold is important as it directly affects the volumetric efficiency, engine emissions, combustion performance and output of the engine. Typically it is believed that the intake valve opens at Top Dead Centre (TDC) and closes at Bottom Dead Centre (BDC) during the suction stroke. However, in the actual scenario, the intake valve opens before the piston reaches the TDC during the exhaust stroke and it closes after the piston starts to move towards TDC during the compression stroke. Because of the early opening of the intake valve, it is observed that a certain interval when both the intake valve and exhaust valve are open causing overlapping, it allows drawing the fresh air into the cylinder which enables to replace the gases at TDC thereby increasing the volumetric efficiency [1]. During the cylinder filling, the pressure loss is observed due to rise in temperature of the combustible mixture when it flows through the intake manifold pipe into the cylinder, therefore the charge density is lesser than the ambient charge density. These losses account for the decrease in volumetric efficiency of the cylinder. It is also affected by throttle pipe construction, size and design of the intake port, throttle valve position and combustion chamber shape [2]. The decrease in volumetric efficiency of the engine and due to flow resistance in the air intake manifold decreases the peak power output of the engine [3]. The nozzle that converges to the throat and diverges afterward is called a convergent-divergent nozzle. It has a high expansion ratio i.e. divergent portion produces a stream of high velocities [4]. Volumetric efficiency for the engine is between 85-90% (generally for SI engines) [5]. K-Epsilon RNG model was used for the simulations as it is verified with computational and experimental results for air flow through the intake manifold [6].
Wave dynamics can be involved in analyzing the air flow through an intake manifold. However, the actual wave dynamics is complex but the intake system can be modeled as a simple Helmholtz resonator (with pressure waves) [7]. When the intake valve opens and piston descends, a rarefaction or expansion wave travels upstream in the runner towards the plenum. At the same time, a part of the original expansion wave is reflected as a compression wave heads back to the cylinder. If this compression wave arrives at the intake valve before it closes then an increased pressure will effectively force more air into the cylinder. This effect leads to an increase in volumetric efficiency and power. Through proper valve operation timing and tuning the runner length, we can employ pressure wave dynamics to increase the mass of air transferred from the intake manifold to the combustion chamber [8].

Literature is available for plenum design [7, 9] and runner design [8] in order to improve the volumetric efficiency of the engine. Very limited literature is available on the restricted intake manifold. It is reported that volumetric efficiency and hence the power of the engine decreases [10]. The pressure and velocity variation at the restrictor is not significant during the intake stroke, but the effect of turbulence in the plenum had a significant effect [11]. Hence present paper focuses on two novel designs (models) for the restricted intake manifolds along with novel design of plenum in order to improve the volumetric efficiency and turbulence inside the combustion chamber of the engine having restricted intake manifold since it was mandatory to use the restricted manifold in the race car manufactured for SAE SUPRA competition held in July 2018.

Selected two novel models were simulated using Ansys 15.0. The better model was selected based on static and total pressure, the velocity of air flow, and turbulence kinetic energy observed in the proposed models. Subsequently, volumetric efficiency was calculated for the better model of the restricted intake manifold designed for a 2014 KTM RC 390 engine used in SAE SUPRA competition 2018.

2. Methodology
Restricted intake manifolds with volume (total volume of divergence and plenum section as shown in figures.1-2.) beyond 8 times the displacement volume of the engine have experimentally shown significant improvement in performance parameters [9]. The volume of the plenum for model 1 (Fig. 1) is 3254 cc and for model 2 (figure 2) is 3128 cc which is about 8.7 and 8.4 times of the combustion chamber volume respectively. The restrictor was designed in shape of a convergent-divergent nozzle.

![Figure 1: Model of Restricted Intake Manifold 1](image1.png)  ![Figure 2: Model of Restricted Intake Manifold 2](image2.png)

For both the models intake diameter was 46 mm as per engine throttle body, runner length and the minimum diameter in the cross-section was 305.16 mm and 20 mm respectively. The overall length of the model 1 and model 2 manifolds were 807.45 mm and 812.4 mm and the diameter of the largest cross-section was 80 mm and 60 mm respectively. The front part of the restricted intake manifold is attached with the throttle assuming throttle was at the fully open position.

![Figure 3a: Mesh](image3a.png)  ![Figure 3b: Mesh near the wall](image3b.png)
Figure 3a and 3b illustrate the meshing details for the proposed models. Meshing was done using ANSYS ICEM CFD with the hexahedral element at the core of the design, trigonal element at the wall to capture the boundary physics and tetrahedral elements in between them. The mesh was made finer near the walls subsequently for good resolution on near-wall effects. The flow was also analyzed at the smallest cross-section of the manifolds that is at the restrictor neck by making a finer mesh because it’s the major section where it was observed a sharp change in pressure and velocity of air flow. Similar meshing was done for intake manifold model 2. The average mesh quality for Model 1 was 0.784 and Model 2 was 0.787.

2.1 Engine and Simulation Details

For simulations of both the models, at intake a constant velocity boundary condition was considered and air with velocity 30 km/h (8.33 m/s) was going inside the manifold. Spatial discretization gradient was Green-Gauss Node based. The partial differential equation for Momentum, Pressure, Turbulence kinetic energy, and Turbulence dissipation rate was calculated to second order upwinding. Pressure-velocity coupling was achieved using Coupled Algorithm. At the runner outlet, constant pressure conditions with pressure as 1atm were considered. The under-relaxation factor for momentum was 0.3 and under-relaxation factor for pressure was 0.3. The convergence criteria for both the models was $10^{-4}$. A pressure based solver was used for simulation.

### Table 1: Geometrical details for engine [12]

| Parameters             | Values       |
|------------------------|--------------|
| Displacement           | 373.2cc      |
| Bore                   | 89mm         |
| Stroke                 | 60mm         |
| Compression Ratio      | 12.6         |
| Power(RPM)             | 43hp(9600)   |
| IVO                    | 9.5 BTDC     |
| IVC                    | 54.5 ABDC    |
| Intake Valve Overlap   | 113°         |

2.2 Equations Discretized

The flow through the intake manifold was treated as an incompressible and steady state. The following equations were discretized using finite volume techniques.

**Continuity equation,**

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

**Momentum equation,**

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} + \frac{1}{\rho} \frac{\partial p}{\partial x_i} = \nu \Delta u_i$$

The continuity equation is then combined with the momentum equation and divergence-free constraint becomes an elliptic equation for pressure.

$$\frac{\partial}{\partial t} \left( \frac{\partial p}{\partial x_i} \right) = -\frac{\partial}{\partial x_j} \left[ \frac{(\rho u_i u_j)}{\partial x_j} \right]$$

In this study, RNG k- epsilon model was used as it provides a more general and fundamental approach and is expected to yield improved predictions of near wall flow, separated flows, flows in curved geometries and flows that are strained by effects like impingement or stagnation. These features make RNG k- epsilon model more accurate.

RNG k- epsilon model,
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon + Y_m + S_k
\]  

(4)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \varepsilon \left( G_k + C_3 \varepsilon G_b \right) - C_{2\varepsilon} \rho \varepsilon^2 k^{-1} + R_\varepsilon + S_\varepsilon
\]  

(5)

The value of constants are \( C_\mu = 0.0845 \), \( C_{1\varepsilon} = 1.42 \) and \( C_{2\varepsilon} = 1.68 \).

2.3 Grid Independence

Grid independence test was conducted to study the effect of varying number of nodes on the results. For this purpose, four different nodes were created for intake manifold geometry. The number of nodes was 83684 (83K), 179699 (179K), 387766 (387K) and 531820 (531K). Figure 4, displays the rake made with 100 partitions to collect the data on Turbulence Kinetic Energy (TKE) for grid independence test. The red arc portions are parts of the circular outlet of the manifold and the points marked between the arcs are the points where TKE data value was taken to test grid independence.

\[
\text{Figure 4: The rake line made at the outlet}
\]

\[
\text{Figure 5: Grid Independence Plot}
\]

The variation percentage in TKE value at the outlet for the nodes 83K and 179K was 4.39%, for the nodes 179K and 387K was 1.76%, and for the nodes, 387K and 513K is 1.14%. So, based on grid independence test, all results were calculated at 387K meshing. The \( Y^+ \) value measured is 244.77 and is suitable for the RNG K-epsilon model. A similar test was done for the intake manifold design 2, grid independence was found in the mesh with 360343 (360K) nodes.
3. Results and Discussion

Static pressure plots for the intake manifold geometries were as shown in figure 6. It shows that the air entering the intake manifold was at a high pressure but because of the restrictor placed in the path, the velocity of the flow increases drastically attaining the maximum value of 51.3 m/s and 48.07 m/s at minimum cross-section (figure 7 at 0th m) and the pressure reduces proportionally. It can also be seen that model 2 has higher pressure at inlet then model 1 though, the pressure at the outlet is greater for model 1 (figure 8). However, the velocity of flow at the outlet is nearly the same for both models around 8.60 m/s.

![Figure 6: Static Pressure Plot](image)

![Figure 7: Velocity Magnitude contour for models 1 and 2](image)
Figure 8: Total Pressure Contour for models 1 and 2

Figure 8 shows the total pressure (gauge pressure) contours for both the intake manifolds. It can be observed that in some regions for model 1, the negative pressures readings were observed. It was happened due to the formation of a vortex in the plenum chamber (figure. 9). The advantage of such a flow field is that it creates more turbulence in the air flow which helps in the better mixing of the air-fuel mixture. Also, it creates an extra swirling of the airflow going inside the combustion chamber. Figure 9 shows the path lines of air flow inside the intake manifold. Vortex formation can be seen near to the cross-section with the largest diameter. Whereas, there was no such flow in model 2, Fig. 10.

The total pressure (gauge pressure) at the outlet of the intake manifold for the model 1 and 2 were 59.27 Pa and 56.62 Pa, mass-weighted average turbulent kinetic energy at the output were 0.82 m²/s² and 0.81 m²/s² and velocity magnitude at the outlet was 8.58 m/s and 8.57 m/s respectively.

4. Conclusions

It is observed that the total pressure at the outlet is greater for the model 1 (59.27 Pa gauge). Hence a higher pressure difference between air at intake valve and inside the combustion chamber can be maintained which will facilitate better flow of air going inside the combustion chamber.

Velocity magnitude and turbulence kinetic energy are nearly the same for both the models. However, vortex formation was observed in model 1 which ensures better mixing of air with fuel since fuel is injected into the flow at the throttle which is placed before the intake manifold. Hence, it can be concluded that model 1 is better than model 2.

Mass flow rate at the output of the intake manifold was 0.017 kg/s, using the engine specifications mentioned in table 1, volumetric efficiency at 7250 RPM was calculated to be 53% [13]. The volumetric
efficiency of the engine was decreased because restrictor restricted the amount of air going inside the intake manifold towards the combustion chamber. The future scope is to develop techniques to increase volumetric efficiency and turbulence for restricted flow engine for the better combustion, performance, and lower emission characteristics. Among such ways is to use turbocharger and use of dual or variable intake manifolds to maximize airflow can be done with the restricted intake manifold.

Acknowledgment
The authors would like to thank Mr. Mahesh Dasar for helping us at various points in due course of the study. We would also like to thank Prateek Nagavani, Niranjan Gyanchandani and Urval Shah for their support.

References
[1] Ganesan V 2012 IC Engines (McGraw Hill) p 140.
[2] Lumley J L 2009 Engines (New York: Cambridge university press) p 245.
[3] Melaika M, Rimkus A and Vipartas T 2017 Air Restrictor and Turbocharger Influence for the Formula Student Engine Performance 10th International Scientific Conference Transbaltica 2017: Transport Science and Technology 187 402-7.
[4] http://nptel.ac.in/courses/Webcourse-contents/IITKANPUR/machine/ui/Course_homelec19.htm
[5] V Ganesan 2012 IC Engines (McGraw Hill) p 24.
[6] Paul B and Ganesan V 2010 Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds International Journal of Engineering, Science and Technology 2 1(1) 80-91.
[7] J B Heywood 1988 Internal Combustion Engine Fundamentals (McGraw Hill) p 205.
[8] Benajes J, Reyes E, Galindo J and Peidro J 1997 Predesign Model for Intake Manifolds in Internal Combustion Engine, SAE Paper-970055 1-12.
[9] Hamilton L J and Lee J E 2008 The Effect of Intake Plenum Volume on the Performance of a Small Normally Aspirated Restricted Engine SAE International Journal for Engines 1 (1) 1-7.
[10] Vora K C, Agrewale M R B and Jain S 2017 Design and Experimental Validation of Intake System to Improve Performance of Race Car International Journal of Scientific and Engineering Research 8(2)
[11] Ling J and Tun L 2006 CFD Analysis of Non- Symmetrical Intake Manifold for Formula SAE Car SAE Technical Paper 2006-01-1976.
[12] http://www.ktmforums.com/forums/rc8-rc8r/56490-cam-timing-numbers.html
[13] https://www.otetools.com/ve

Abbreviation
IVO: Intake Valve Open
IVC: Intake Valve Close
BTDC: Before Top Dead center
ABDC: After Bottom Dead Centre