Design, Analysis and Comparative Study of Exhaust Manifold for a Multi-Cylinder Four Stroke SI Engine using CFD

Vivek Chaudhary[1], Dr G. Amba Prasad Rao[2]

[1]*Post Graduate Student, Dept. of Mechanical Engineering, NIT Warangal*
[2]*Professor, Dept. of Mechanical Engineering, NIT Warangal, Telangana, India*

---***---

Abstract - *Gas exchange phenomena of an IC Engine take place through intake and exhaust manifolds. The engine's performance is influenced by the gas exchange process. The engine's performance improves when the exhaust manifold is in good condition. In the present work, an attempt is made to the modify exhaust manifold by upgrading its design for improved performance for a chosen multi-cylinder engine configuration. The primary objective of the present work is to enhance the design to reduce backpressure in the exhaust manifold for enhanced engine performance. Computational fluid dynamics (CFD) is a popular and current operating software that is widely used. CFD is largely used in the automobile industry to lower the cost of designing and analyzing various models in the fluid flow area. The flow through the exhaust manifold is performed using commercially available software. Efforts are made through CFD software to analyze various shapes of exhaust manifolds. Five distinct models were built and thoroughly analyzed using velocity and pressure contours to get the optimum shape for minimal backpressure and high exhaust velocity. The velocity contour and pressure contour are used to compare six distinct models, and the best potential model for low backpressure and high exhaust velocity is given. Among the six models designed, model 6 yielded satisfactory performance.*

Key Words: Multi-Cylinder Engine, Exhaust Manifold, Back Pressure, Exhaust Velocity, Exhaust Temperature.

1.INTRODUCTION

One of the most significant components of an IC engine is the exhaust manifold. Exhaust manifold design is a challenging technique because it is dependent on many parameters such as exhaust velocity, backpressure, mechanical efficiency, and so on. The importance of any of these criteria is determined by the designer's requirements. Backpressure, exit exhaust velocity, and power required are some of the exhaust manifold design considerations. The exhaust manifold is constructed of cast iron or stainless steel. It collects gases from several cylinders and routes them to the exhaust manifold. Many studies and research have been conducted in this topic. The study's goal is to gain a thorough understanding of the manifold operation, flow property distribution, and heat transfer details. Due to the inclusion of so many components, it is impossible to replicate the actual scenario in a model. The output gases from more than two cylinders are collected into a single pipe in an exhaust manifold of any multicylinder engine. The outlet of the multicylinder heads is connected to the inlet of the multicylinder exhaust manifold. In multicylinder engines, the exhaust manifold is attached to the engine and is the principal portion where the stream from multiple cylinders is to be collected into a single pipe. The accumulated hot exhaust gases exit through the single exhaust manifold outlet from the cylinder header. For the engine to work optimally, the backpressure of the exhaust manifold at the output should be reduced. For domestic automobiles, the higher the backpressure, the poorer the performance or economy. Higher backpressure is required for race cars to reach higher speeds in less time. To achieve optimal mechanical and thermal efficiency, the exhaust gases should have a high velocity at the exhaust manifold's exit.

Table -1: Engine Specifications

Engine	4 Stroke 4 Cylinder SI Engine
Make	Maruti-Suzuki Wagon-R
Calorific Value of Fuel (Gasoline)	45208 KJ/KG-K
Specific Gravity of Fuel	0.7
Bore and Stroke	69.05 mm x 73.40 mm
Swept Volume	1100 cc
Compression Ratio	7.2:1
Dynamometer Constant	2000
Diameter of Orifice	29 mm
Coefficient of Discharge of Orifice	0.65

Backpressure is a typical issue connected with the exhaust manifold. According to the literature research, much work has been done to improve the exhaust manifold in order to improve the engine's performance. The CFD approach saves manufacturing costs and production time. According to a survey of the literature, many exhaust manifold studies have been conducted utilizing the CFD technique. The following are some excerpts from the literature review.

Dr. M. Senthil Kumar, Dr. S. Sendilvelan, PL.S. Muthaiah [1] have examined the exhaust manifold in order to reduce backpressure in the exhaust manifold and improve exhaust gas filtration. Modification of the geometry of exhaust manifold was done by shifting the size of the conical region of the exhaust manifold as well as the grid size of the meshed wire packed through the exhaust manifold. When the grid size is increased, the filtration level lowers; when the filtration level decreases, the exhaust system emits more exhaust. When the grid size decreases, backpressure is formed in the exhaust manifold, causing engine efficiency to decline. CFD is used to evaluate the exhaust manifold and select the best-design model with the least amount of backpressure and the most amount of PM filtration.

Siddaveer Sangamad, Vivekanand Navadagi [2] used CFD to study the flow through two distinct versions of an exhaust manifold. They analyzed two models: one with the base geometry of the exhaust manifold and one with the modified geometry of the exhaust manifold. Both models were analyzed utilizing pressure and velocity contours. The major goal of this work is to enhance the exit velocity when the exhaust manifold has a low back pressure, which boosts volumetric efficiency and engine efficiency. They concluded that the present model is modified by modifying its geometry while maintaining the same boundary constraints. The new model's results outperform the current model because they analyzed velocity and pressure contours to find the lowest back pressure while boosting the volumetric efficiency of the engine.

KS Umesh, VK Pravin, K Rajagopal [3] performed both the CFD study and experimental verification of the influence of exhaust manifold geometry. They designed and analyzed two different models, an existing model and a modified model. They changed the geometry of the existing model of an exhaust manifold, which was then analyzed using parameters such as velocity contours and pressure contours, and both models were constructed and tested. In CFD analysis, the redesigned model yielded the best results, with little backpressure in the exhaust manifold and increased volumetric efficiency of the engine. The same results were achieved after conducting the experiments with both models. It is easy to deduce that the experimental results match the CFD analytical results. Experimental analysis validated the CFD analysis results.

Mohdsajid Ahmed, Kailash BA, Gowreesh SS [4] constructed five alternative exhaust manifold models and analyzed in this paper. The primary task is to enhance the design of an exhaust manifold since minimizing backpressure in the exhaust manifold improves engine efficiency and performance. They analyzed the simple flow of an exhaust using velocity and pressure contours for five distinct developed models, compared the five models using pressure and velocity contours, and then chose the best one model. They found that model number five is the best model because it has the lowest backpressure and smooth flow of an exhaust without recirculation. It also has the lowest turbulent kinetic energy and the highest output velocity when compared to other models. Using this exhaust manifold, higher exhaust velocity and lower back pressure can be achieved.

2. METHODOLOGY

2.1 CAD Modelling of Exhaust Manifold

The computer aided drawing software named design modeler is used to create the five models. A commercial CFD tool is used to analyze six different models. Meshing software is used to name boundaries and discretize the models. Dimensions in mm are provided for the basic CAD models that were used.

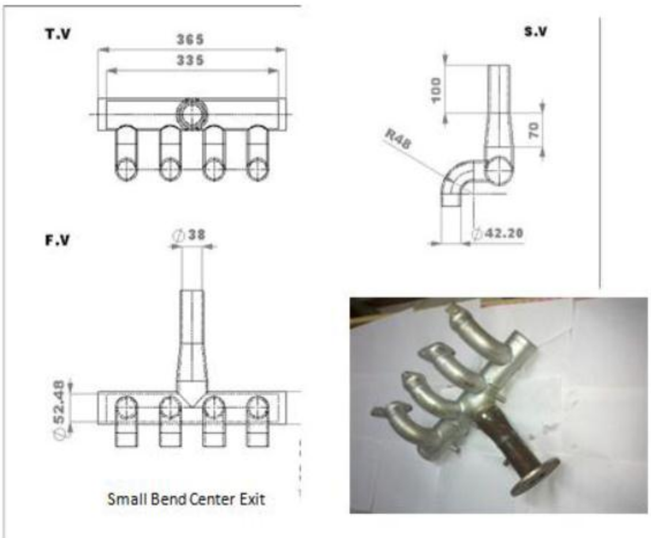

Fig -1: CAD dimensions of exhaust manifold

2.2 Exhaust Manifold Designs using CAD Software

Model 1

In the model 1 shape of the inlet has been modified from straight inlet to convergent inlet and 3D model and its 2-D sketch is shown.

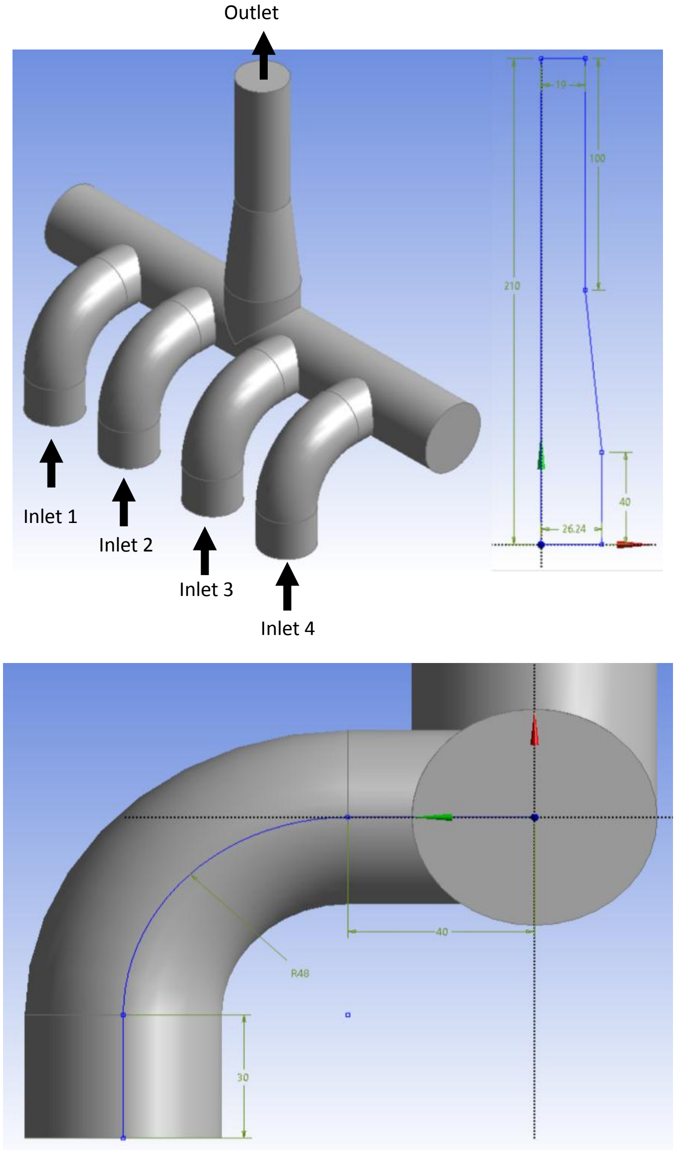

Fig -2: CAD and sketch for Model 1

Model 2

In model 2 outlet of exhaust manifold is modified from converging outlet to divergent-straight-convergent outlet.

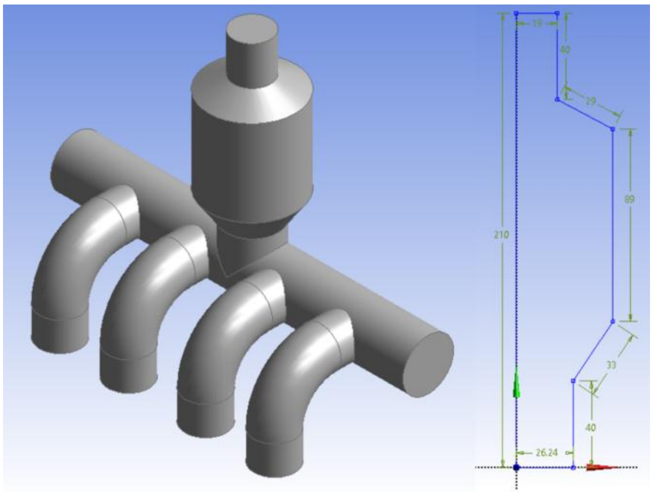

Fig -3: CAD and sketch for Model 2

Model 3

In model 3 the divergence length of the outlet is increased and convergence length is decreased.

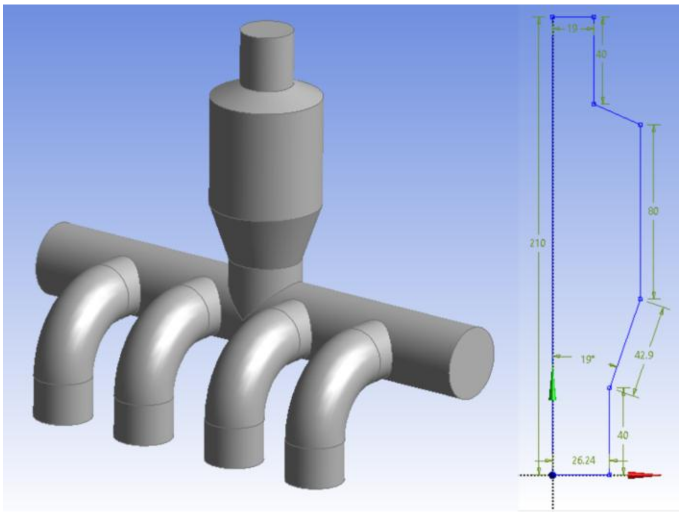

Fig -4: CAD and sketch for Model 3

Model 4

In model 4 the divergence length of the outlet is decreased and the convergence length is increased.

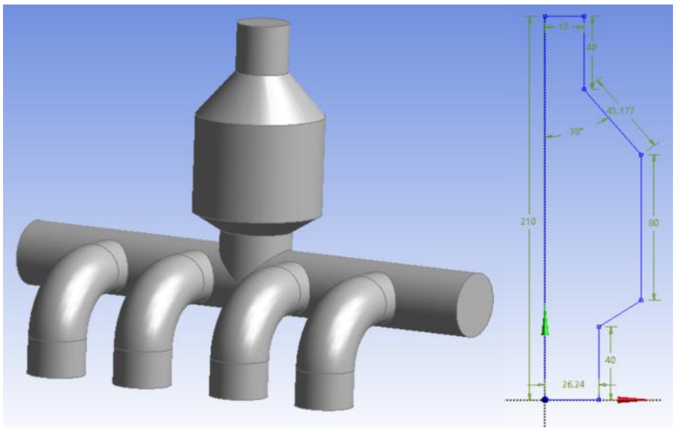

Fig -5: CAD and sketch for Model 4

Model 5

In model 5 the divergent area and convergent area of theoutlet are kept equal and straight area is decreased.

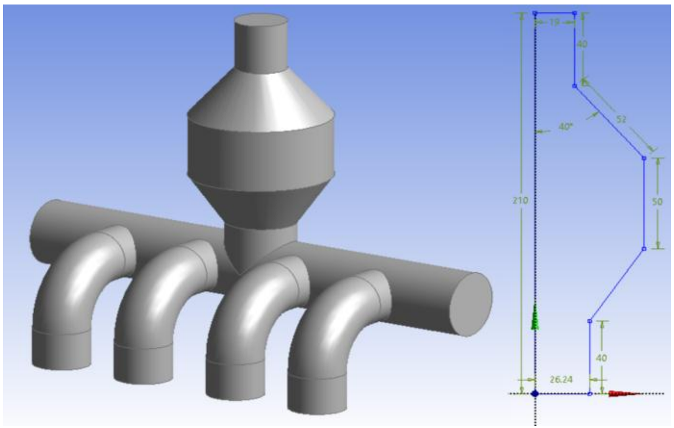

Fig -6: CAD and sketch for Model 5

Model 6

In model 6 the the elliptical cross-section has been used with sharp expansion and convergence shape towards outlet.

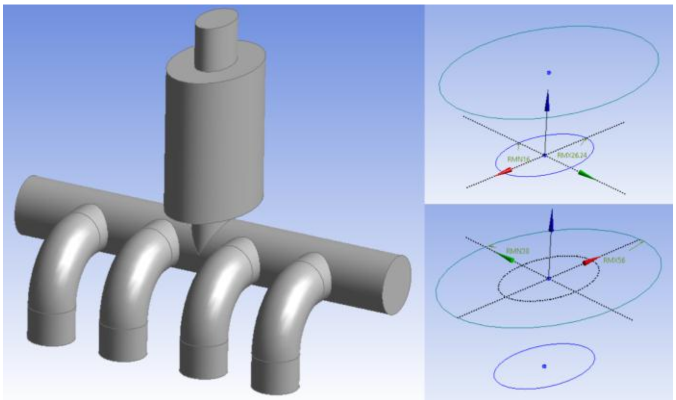

Fig -7: CAD and sketch for Model 6

2.3 Boundary Conditions

Table -2: Boundary Conditions

Named Selection	Boundary Type	Boundary Condition
Inlet	Patch	Mass flow rate inlet
Outlet	Patch	Pressure outlet
Walls	Wall	No Slip and Convection

The fluid considered is air and default material properties for air have been imported from Fluent library. The flow was assumed to be steady state, turbulent along with heat transfer phenomenon. The boundary conditions used at inlet and outlet boundaries are mass flow rate inlet and pressure outlet boundary conditions. Air temperature at inlet is used as 313 ^{o}C. No slip boundary condition was used on the walls along with convection value of 10W/m^{2}K for heat dissipation through walls to atmosphere.

Inlet 1= 0.00188kg/s, Inlet 2 = 0.00188kg/s, Inlet 3 = 0.00188 kg/s, Inlet 4 = 0.00188 kg/s, Outlet= 0Pa.

2.4 Meshing Details

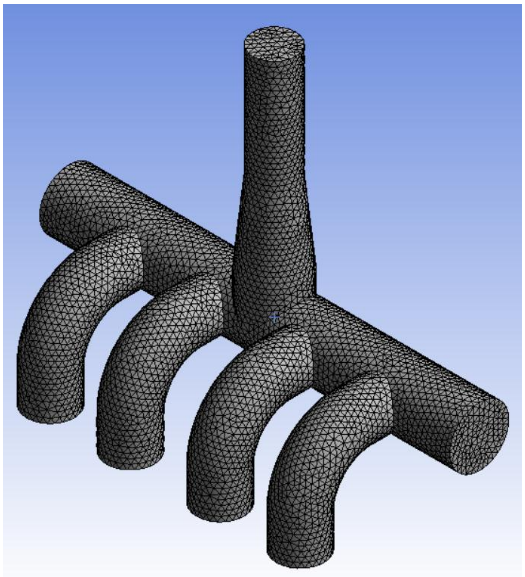

Fig -8: Mesh for Model 1

Total number of nodes - 23052
Total number of nodes - 113977
Maximum Skewness - 0.67
Average Skewness - 0.222

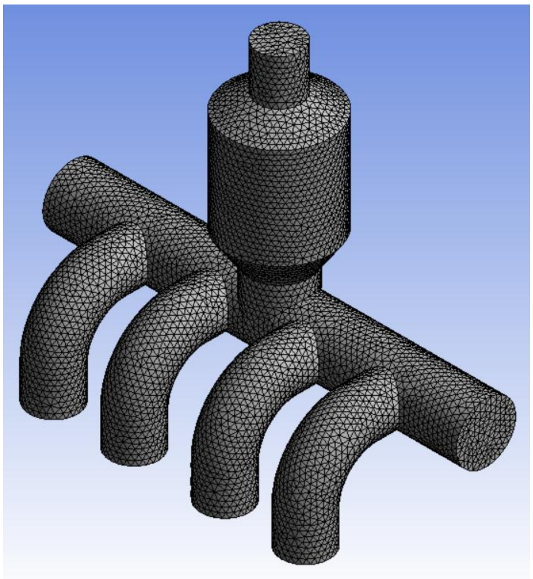

Fig -9: Mesh for Model 2

Total number of nodes - 29329
Total number of nodes - 148220
Maximum Skewness - 0.698
Average Skewness - 0.221

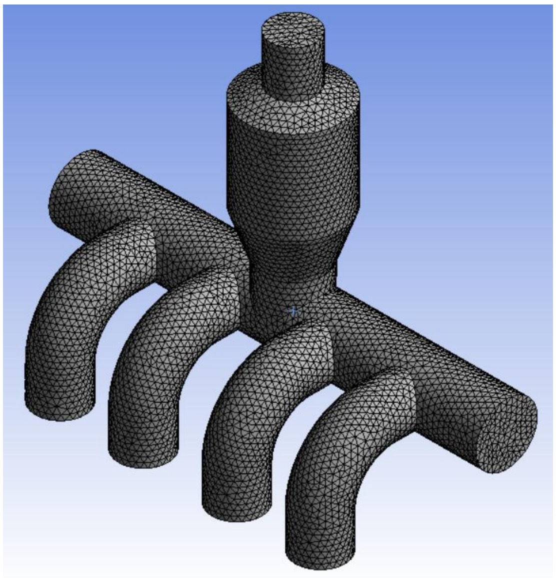

Fig -10: Mesh for Model 3

Total number of nodes - 27608
Total number of nodes - 138885
Maximum Skewness - 0.697
Average Skewness - 0.222

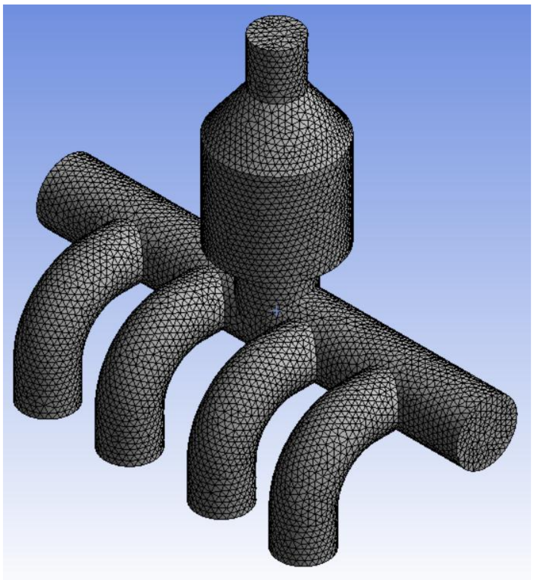

Fig -11: Mesh for Model 4

Total number of nodes - 29650
Total number of nodes - 150082
Maximum Skewness - 0.698
Average Skewness - 0.219

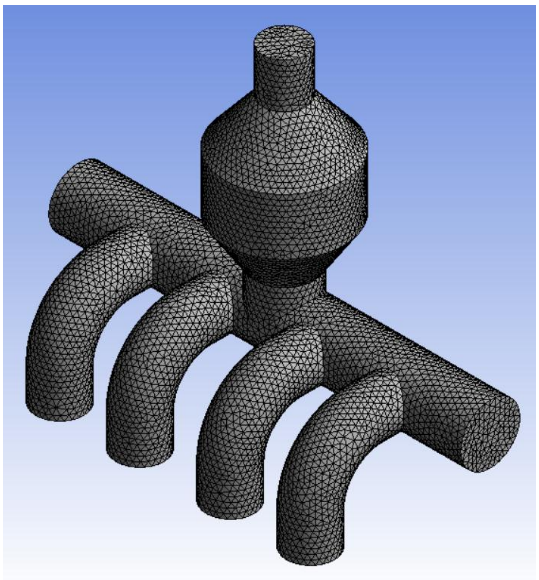

Fig -12: Mesh for Model 5

Total number of nodes - 30317
Total number of nodes - 153864
Maximum Skewness - 0.7
Average Skewness - 0.22

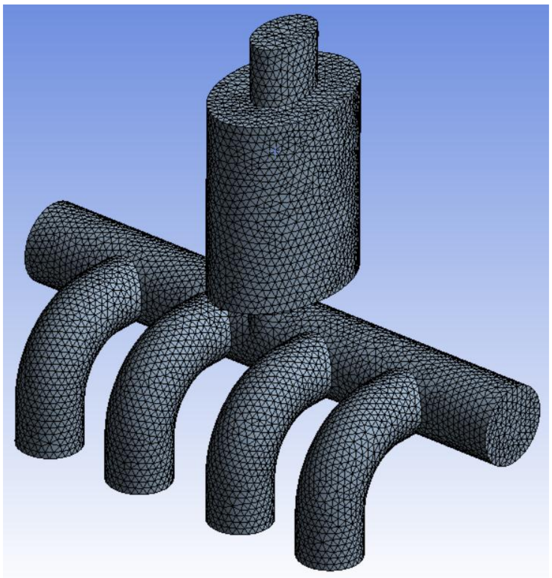

Fig -13: Mesh for Model 6

Total number of nodes - 30751
Total number of nodes - 156290
Maximum Skewness - 0.669
Average Skewness - 0.221

3. RESULTS AND DISCUSSION

3.1 Model 1 Results

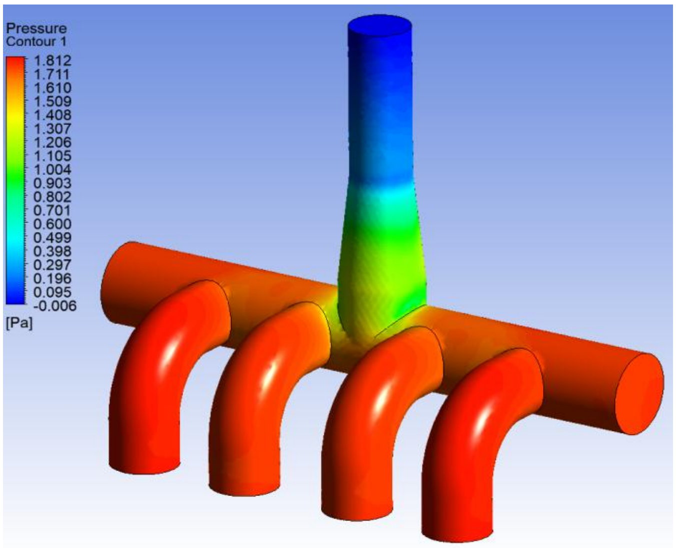

Fig -14: Pressure Contour

Fig. 14 gives the variation of pressure in the exhaust manifold for model 1. It is seen that pressure at outlet is 0 Pa as specified in the boundary condition. However, backpressure is generated and can be seen increasing gradually from outlet towards inlet.

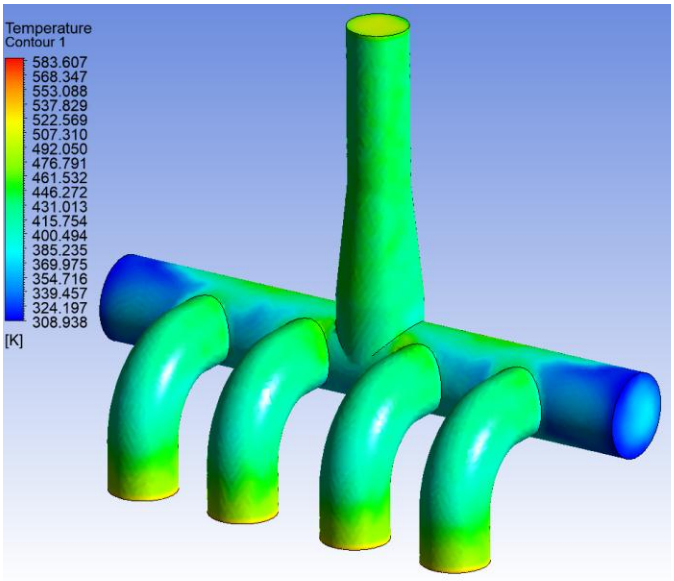

Fig -15: Temperature Contour

Fig. 15 gives the variation of temperature in the exhaust manifold for model 1. It can be seen that hot air is coming though all 4 inlets at same temperature and as the air passes through the manifold it loses heat to the environment due to convection boundary condition through walls. Relatively cooler temperature regions are formed at the side extended region of exhaust manifold as hot air tries to flow out through the outlet directly without flowing to the side extended regions.

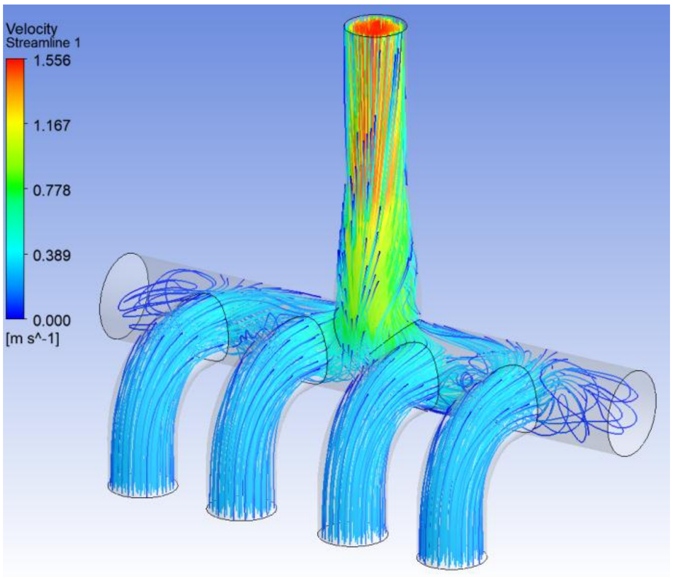

Fig -16: Velocity Streamlines

From the fig. 16 it is evident that due to the convergent shape of the inlet, velocities are found to be lower at the connecting area and inlet. The low velocity results in high backpressure. It is observed that exhaust velocities considerably decrease by designing the manifold using the convergent inlet. It is observed that air circulates in the side extended regions of the exhaust manifold.

3.2 Model 2 Results

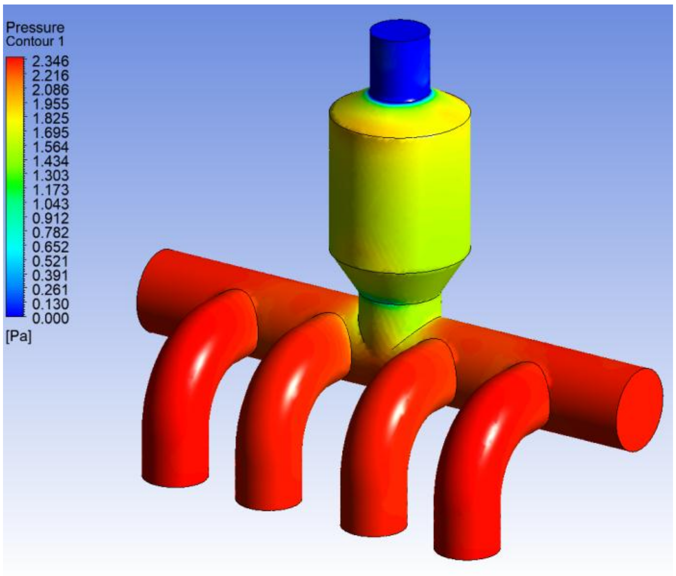

Fig -17: Pressure Contour

Figure 17 gives the pressure distribution over entire domain for model 2. Pressure is higher at the outlet in comparison with model 1. It is evident that the pressure is high at the inlets of model 2. Pressure is high at the middle of the exhaust manifold in comparison with the exit of the outlet.

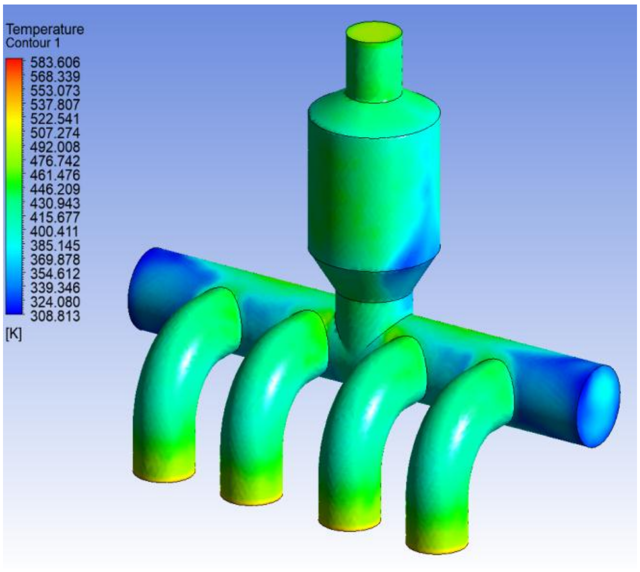

Fig -18: Temperature Contour

Fig. 18 gives the variation of temperature in the exhaust manifold for model 2. Same cold region phenomenon like model 1 on the extended side region is observed for model 2. It can also be seen that cold temperature spot is getting formed on the divergent portion of the exhaust outlet. This occurs due to sudden expansion of air flow cross section.

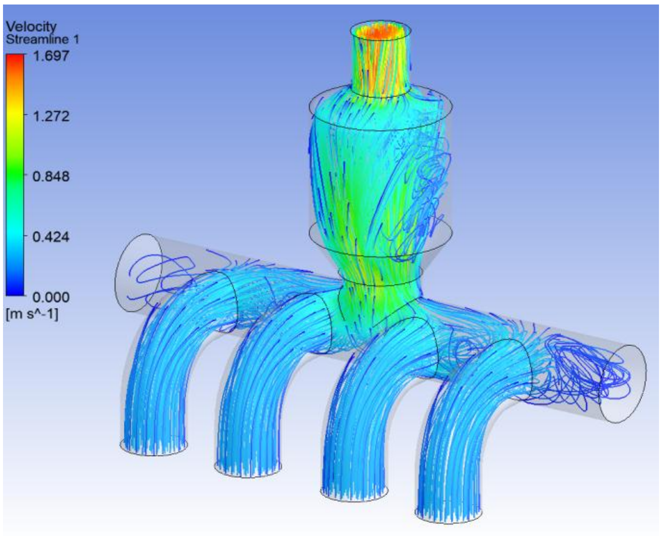

Fig -19: Velocity Streamlines

Figure 19 shows velocity distribution inside exhaust manifold. It is observed that the velocity at the inlets is less and it increases at the outlet. It is seen that the velocities are found to be slightly higher at the outlet in comparisons with model 1. It is observed that the exhaust velocities are considerably increasing by designing the exhaust manifold using this outlet in comparison with model 1.

3.3 Model 3 Results

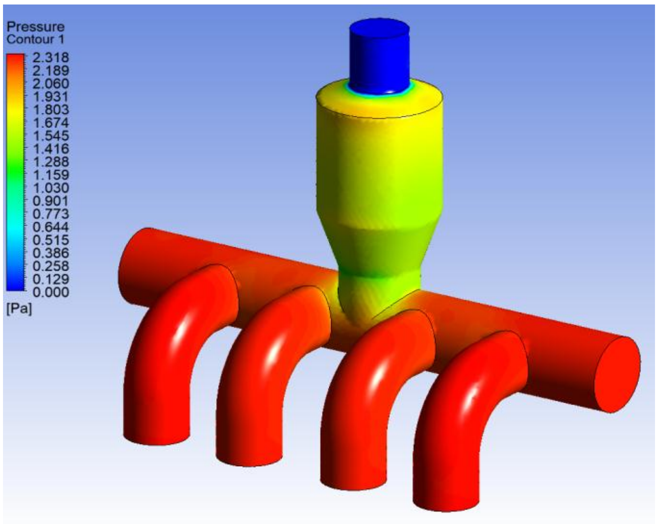

Fig -20: Pressure Contour

Figure 20 gives the pressure distribution for model 3. It is found that the pressure is higher at the inlets. Pressure is higher at the inlet in comparison with the model 1 and almost same in comparison to model 2. It is observed that exhaust pressure is almost same by designing the manifold by decreasing the convergent length of the outlet even further compared to model 2.

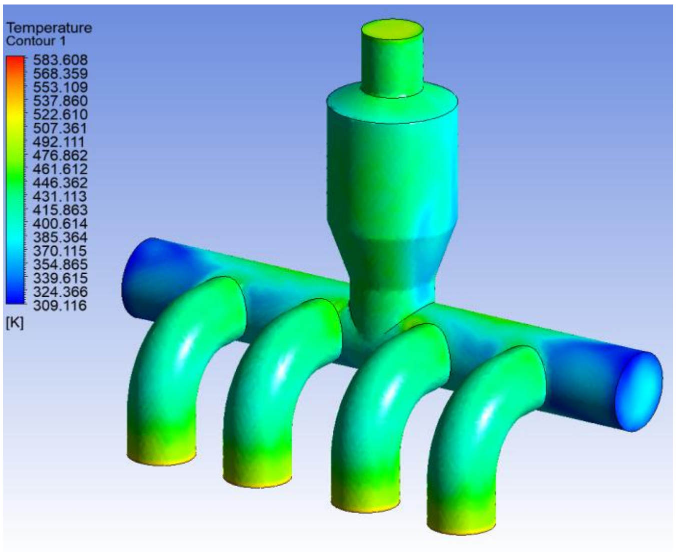

Fig -21: Temperature Contour

Fig. 21 gives the variation of temperature in the exhaust manifold for model 3. Cold region phenomenon like model 1 and 2 is observed on the extended side region of exhaust manifold. It can also be seen that cold temperature spot is getting formed on the divergent portion of the exhaust outlet like model 2. This again occurs due to sudden expansion of air flow cross section area.

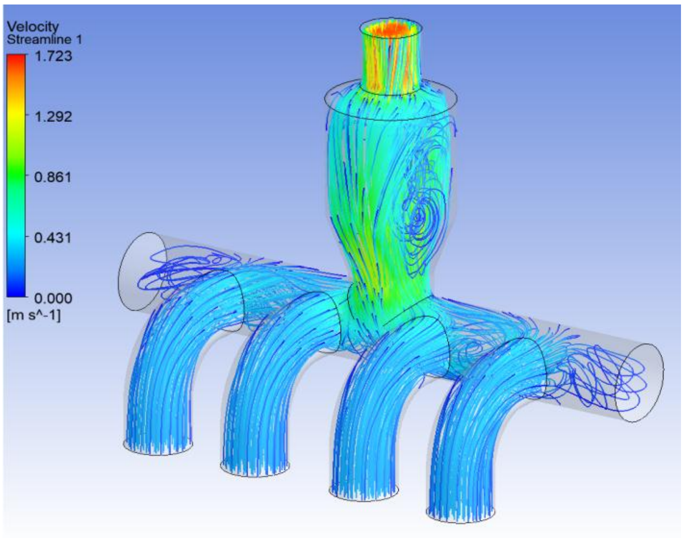

Fig -22: Velocity Streamlines

Figure 22 gives the velocity contour of the model 3. It is seen that the velocity is higher at the outlet in comparison with the model 1 and 2. It is observed that exhaust velocities considerably increase by designing the manifold using the divergent-convergent outlet. It is observed that the velocity is higher at the outlet.

3.4 Model 4 Results

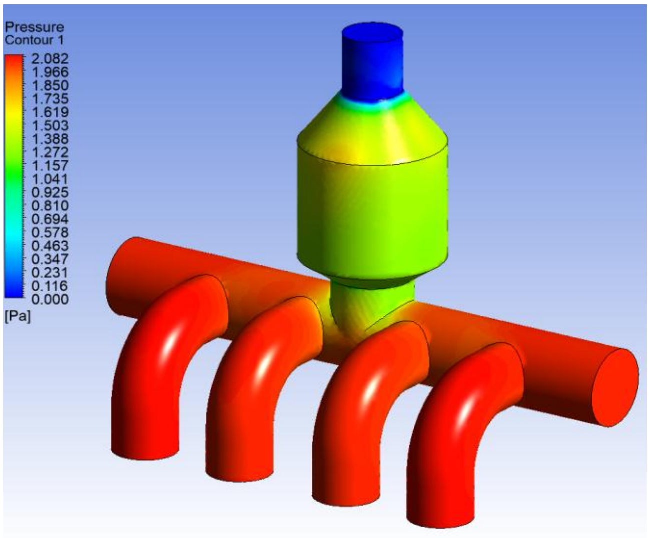

Fig -23: Pressure Contour

Figure 23 gives the pressure distribution for model 4. It is found that the pressure is lower in comparison with the model 2 and 3 but slightly higher than model 1. Higher pressures are observed at the inlets. It is observed that by reducing the divergent length of the outlet the pressure decreases compared to model 2 and 3.

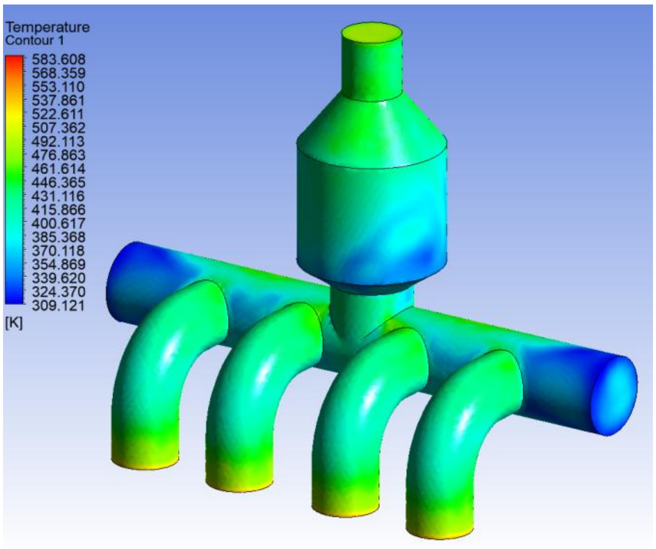

Fig -24: Temperature Contour

Fig. 24 gives the variation of temperature in the exhaust manifold for model 4. Cold region phenomenon like model 1, 2 and 3 is observed on the extended side region of exhaust manifold. It can also be seen that cold temperature spot is getting formed on the divergent portion of the exhaust outlet like model 2 and 3. This phenomenon occurs due to sudden expansion of air flow cross section area.

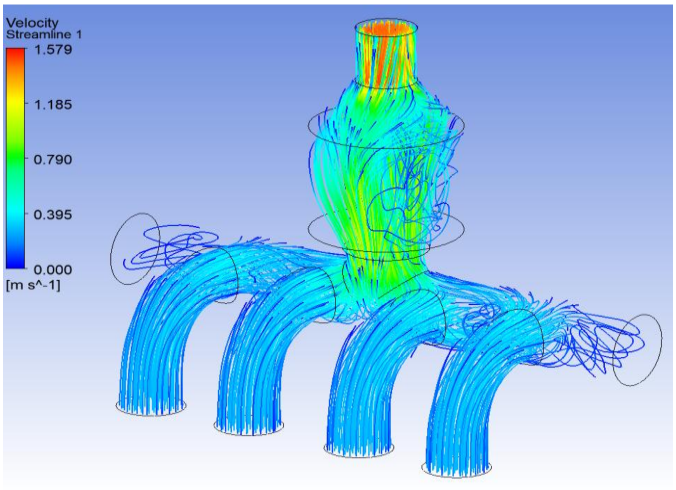

Fig -25: Velocity Streamlines

Figure 25 gives the velocity contour of model 4. It is observed that due to the sudden expansion of the area of outlet velocities are found to slightly increase compared to model 1 at the outlet. It is observed that the exhaust velocities are slightly increasing by designing the exhaust manifold by reducing the divergent length of the outlet in comparison with model 1. Velocities are found to be higher at the outlet due to convergent cross section area near outlet of the exhaust manifold.

3.5 Model 5 Results

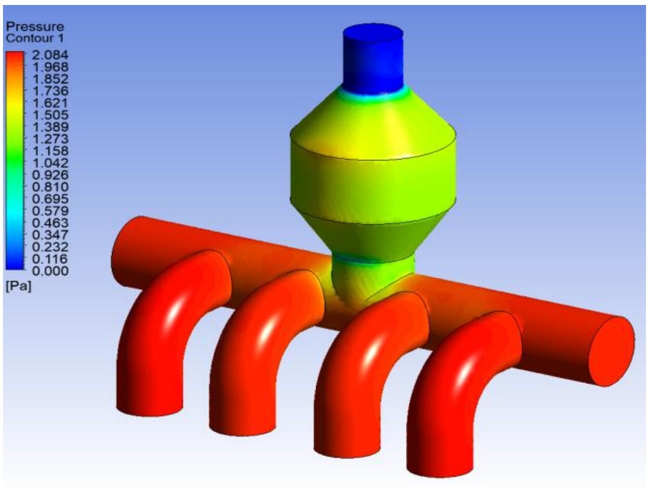

Fig -26: Pressure Contour

Figure 26 gives the pressure distribution for model 5. It is found that the pressure is higher in comparison with the model 1 but it is lower compared to model 2 and 3. Back pressure is slightly higher than model 4. Higher pressure is observed at the inlets. It is observed that by reducing the straight length of the outlet the pressure increases considerably.

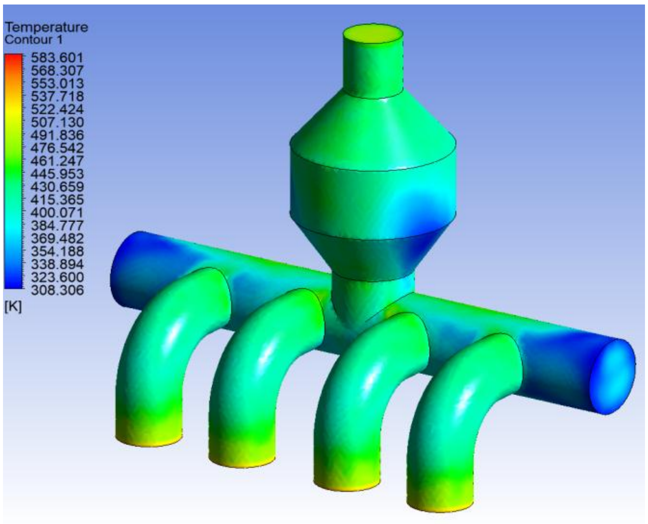

Fig -27: Temperature Contour

Fig. 27 gives the variation of temperature in the exhaust manifold for model 5. Cold region phenomenon is again occurring like model 1, 2, 3 and 4 and is observed on the extended side region of exhaust manifold. It can also be seen that cold temperature spot is getting formed on the divergent portion and straight portion of the exhaust outlet like model 2, 3 and 4. This phenomenon occurs due to sudden expansion of air flow cross section area after the connecting region.

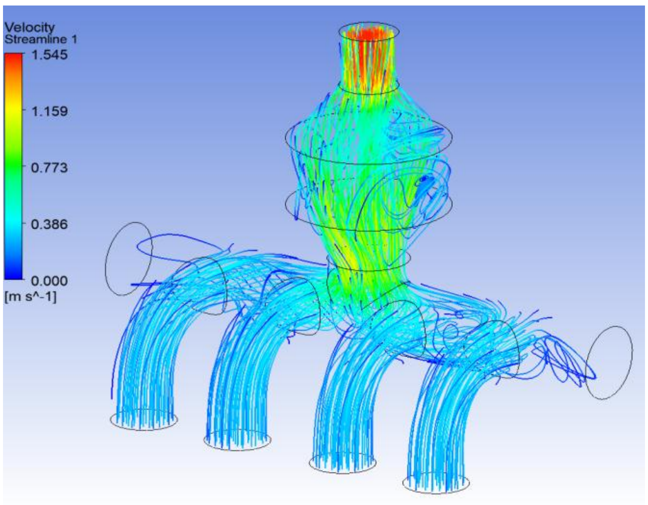

Fig -28: Velocity Streamlines

Figure 28 gives the velocity distribution in model 5. It is observed that due to the divergent convergent shape of the outlet, velocity is almost similar at the outlet in comparison with model 1. It is observed that the exhaust velocities remain almost same by designing the exhaust manifold by reducing the straight length of the outlet in comparison with model 1. The velocity streamlines of the model 5 show that the flow is uniform in comparison with the other models. Very less recirculation is observed in the model compared to all other models.

3.6 Model 6 Results

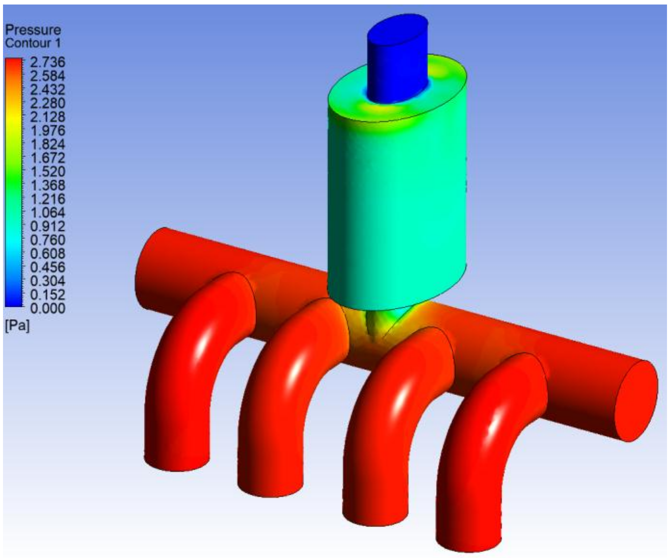

Fig -29: Pressure Contour

Figure 29 gives the pressure distribution for model 6. It is found that the back pressure is higher in comparison with the model 1, 2, 3, 4 and 5. Higher pressure is observed at the inlet region. It is observed that by increasing the straight length of the outlet the pressure increases considerably.

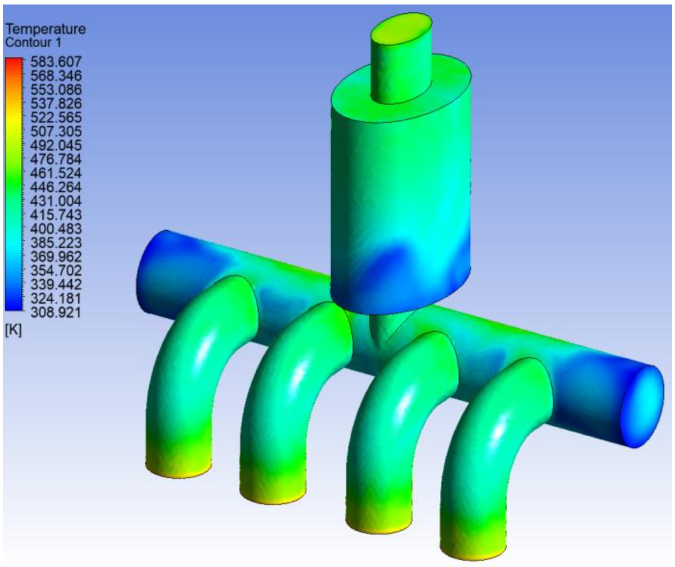

Fig -30: Temperature Contour

Fig. 30 gives the variation of temperature in the exhaust manifold for model 6. Cold region phenomenon is again occurring like model 1, 2, 3, 4, 5 and is observed on the extended side region of exhaust manifold. It can also be seen that cold temperature spot is getting formed on the divergent portion and straight portion of the exhaust outlet. This cold region is bigger and forming over entire circumference of the outlet near the connecting region. This phenomenon occurs due to sharp expansion of air flow cross section area after the connecting region.

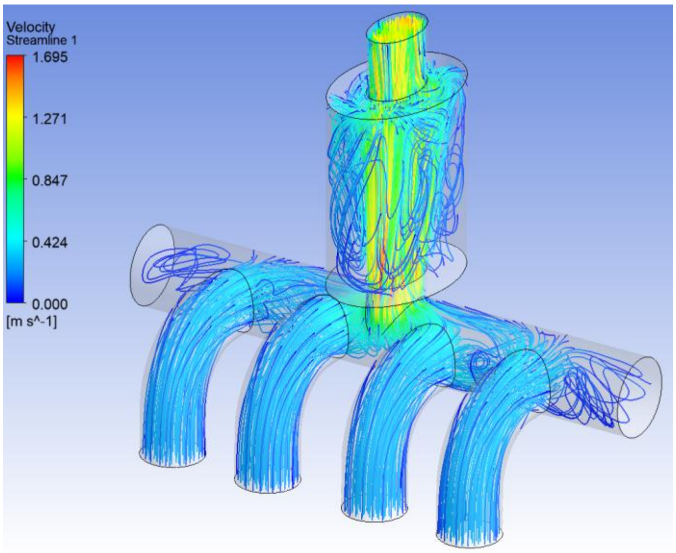

Fig -31: Velocity Streamlines

Figure 31 gives the velocity distribution in model 6. It is observed that due to the sharp divergent convergent shape of the outlet, velocity is lower at the outlet in comparison with model 1, 2, 3, 4 and 5. It is observed that the exhaust velocity decreases by designing the exhaust manifold by increasing the straight length of the outlet in comparison with model 5. The velocity streamlines of the model 5 show that the flow is recirculating in the side extended regions as well as the sharp divergent outlet region.

Comparison of all the results from model 1, 2, 3, 4, 5 and 6 have been listed down in the table below.

Table -3: Results comparison for all models

Design	Pressure (Pa)	Outlet Temperature (°C)	Outlet Velocity (m/s)
1	1.7805	219.63492	1.3709575
2	2.3489877	210.6744	1.3822167
3	2.3197141	212.04085	1.3769754
4	2.073618	212.85724	1.3769688
5	2.0756424	210.60634	1.3799083
6	2.7409584	206.5805	1.174578

4. CONCLUSIONS

Different exhaust manifolds are analyzed in this work using commercial CAE software, and the flow of exhaust is seen, as well as the velocity, pressure and temperature distribution throughout the length of the exhaust manifold is determined through simulation. CFD Post processing was used to analyze the outcomes of six different models. The usage of various shapes of exhaust manifolds aids in the easy flow of exhaust as well as reduction in exhaust outlet temperature.

- Model 6 facilitates low velocity flow of exhaust and high backpressure at the exhaust outlet but the outlet temperature is lowest in comparison with all other models.
- Velocity at the outlet of model 6 is less compared to all other models and hence the backpressure increases considerably.
- The optimum design for an exhaust manifold to achieve lowest outlet temperature is Model 6 with 2.74 Pa back pressure, outlet velocity 1.17m/s and outlet temperature of 206.58°C. Temperature reduction of 5.94% at outlet is observed with this design in comparison to model 1.
- Larger backpressure and lower exhaust velocities are achieved by using exhaust manifolds with sharp convergent divergent shape and longer straight length but it also reduces the outlet temperature.
- The next best design would be model 5 with slightly higher back pressure and slightly higher velocity compared to model 1 along with outlet temperature value of 210.6°C. Temperature reduction of 4.11% at outlet is observed with this design in comparison to model 1.

ACKNOWLEDGEMENT

The authors are grateful to the authorities of NIT Warangal for providing all facilities.

REFERENCES

[1] PL.S. Muthaiah, Dr. M. Senthil kumar, Dr. S. Sendilvelan, "CFD analysis of catalytic converter to reduce particulate matter and achieve limited back pressure in Diesel engine", volume:10 Issue:5 Oct-2010.

[2] Vivekanand Navadagi, Siddaveer Sangamad. "CFD analysis of exhaust manifold of multi-cylinder petrol engine for optimal geometry to reduce back pressure", volume: 3 Issue :3 March-2014.

[3] KS Umesh, VK Pravin, K Rajagopal, "CFD analysis of exhaust manifold of multi-cylinder SI engine to determine optimal geometry for reducing emission", volume:3 Issue:4 Oct- 2013.

[4] Mohd Sajid Ahmed, Kailash B A, Gowreesh, "Design and analysis of a multi-cylinder four stroke SI engine exhaust manifold using CFD technique", volume:02 Issue:09 Dec-2015.

[5] P. Seenikannan, V. M. Periasamy and P. Nagaraj, " A design strategy for volumetric efficiency improvement in a multi-cylinder stationary diesel engine its validity under transient operation",volume:5 issue:3, 2008.

[6] Kim et al, "A non-noble metal based catalytic converter for two strokes, two-wheeler applications", SAE Paper No. 2001011303, 2001.

[7] C. Lahousse, B. Kern, H. Hadrane and L. Faillon Multi-cylinder SI Engine" International Journal of Engineering & Science Research IJESR/July 2013/ Vol-3/Issue-7/342-353.

[8] Binachiet al, "Possibilities of the application of catalyst technologies for the control of particulate emission for diesel vehicles", SAE Transaction 2001, paper no. 2001280044.