DESIGN AND ANALYSIS OF A MULTI-CYLINDER FOUR STROKE SI ENGINE EXHAUST MANIFOLD USING CFD TECHNIQUE

Mohd Sajid Ahmed[1], Kailash B A[2], Gowreesh[3]

[1]PG Student,[2]Assistant Professor, Department of Thermal Power Engineering
VTU PG Centre, Regional office,Kalaburagi,Karnataka, India.
[3]Associate Professor, Department of Mechanical Engineering, JSSATE, Bengaluru, Karnataka, India.

Abstract

Exhaust manifold is one of the important components of internal combustion engine which plays a major role in the improvement of fuel consumption of the engine. Combustion efficiency of the engine can be improved by improving the exhaust manifold design in internal combustion engine. Performance of the engine increases if the exhaust manifold is of a good condition. The designing of exhaust manifold is a complex procedure and is dependent on many parameters. Better the exhaust manifold design better is performance of the engine. The major work is to improve the design to lower the backpressure in the exhaust manifold which increases the performance of the engine.Computational fluid dynamics (CFD) is one of the most popular and current running software which is mostly used in automobile industries in order to reduce the cost which is spent in design and analysis of various models in the fluid flow field. In the present work, analysis is carried out for different shape of exhaust manifolds using CFD software.To achieve the optimal geometry for the low backpressure and high exhaust velocity, five different models were designed and comprehensively analyzed with the help of velocity contours and pressure contours. Using the commercially available software the flow through exhaust manifold is done. Comparison is done for five different models using the velocity contour and pressure contour and best possible model for lower backpressure and high exhaust velocity is suggested.

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

***

1. INTRODUCTION

In an IC engine, one of the most important component is the exhaust manifold. Exhaust manifold designing is a complicated procedure as the designing depends on many parameters viz. exhausts velocity, backpressure, mechanical efficiency etc. The significance of any of these parameters depends upon the designer's needs. Usually backpressure, exit exhaust velocity, power requirement are the some thoughts of the exhaust manifold design. Exhaust manifold is made up of cast iron or stainless steel .It collect gases from different cylinders and supply to the exhaust manifold. Many research and lots of studies have been done in this field. The liquid cooled exhaust manifold using CFD is done by the sir Scheeringa et al. The purpose of the study is to understand the manifold operation and flow property distribution and heat transfer detailed information.The replication of actual condition in model due to involvement of such a huge number of factors is almost impossible. In an exhaust manifold of any multicylinder engine the output gases are collected from more than two cylinders into a single pipe. The inlet of the multicylinder exhaust manifold is connected to the outlet of the multicylinder heads. Exhaust manifold is attached to the engine and major part where the stream from different cylinders is to be collected into a one pipe in multicylinder engines. The collected hot exhaust gases come out from the single outlet of the exhaust manifold from the header of the cylinder.The backpressure of the exhaust manifold at the outlet should be less in order to have a optimum performance of the engine. Higher the backpressure lower is the performance or economy for domestic vehicles. Higher backpressure is required for the race cars to achieve the higher speed in shorter time. The exhaust gases should have high velocity at the outlet of the exhaust manifold to have a good mechanical efficiency and thermal efficiency.

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 one of the common problem associated with the exhaust manifold. The literature review reveals that the lots of work have been done for the improvement of the exhaust manifold in order to improve the working of the engine. CFD method reduces the cost of manufacturing and production time. Literature review shows that lots of exhaust manifold study have been done using the CFD technique. Some of the literature review are as follows.

PL. S. Muthaiah [1], He has analyzed the exhaust manifold in order to reduce the backpressure and also to increase the particulate matter filtration. He has modified the different exhaust manifold by varying the size of the conical area of the exhaust manifold and varying the size of the grid wire mesh packed throughout the exhaust manifold. When size of the grid mesh packed decreased the backpressure increases which leads to lower the performance of the engine due to more fuel consumption and hence low volumetric efficiency. When size of the grid mesh packed increased the backpressure decreases the filtration of the particulate matter also reduces which will not satisfy the standards of the pollution control. Computational fluid dynamics is used for the study of the exhaust manifold and best possible design of the exhaust manifold with minimum backpressure and maximum particulate matter filtration efficiency is suggested.

K.S. Umesh, V.K. Pravin and K. Rajagopal [2] In this work eight different models of exhaust manifold were designed and analyzed to improve the fuel efficiency by lowering the backpressure and also by changing the position of the outlet of the exhaust manifold and varying the bend length. The eight different modified models are short bend centre exit (SBCE), short bend side exit (SBSE), long bend centre exit (LBCE), long bend side exit (LBSE), short bend centre exit with reducer (SBCER), short bend side exit with reducer (SBSER), long bend centre exit with reducer (LBCER), long bend side exit with reducer(LBSER).After analysis they included that the exhaust manifold with long bend centre exit with reducer (LBCER), gives the highest overall performance.

Kulal et al.(2013) [3]work comprehensively analyzes eight different models of exhaust manifold and concluded the best possible design for least fuel consumption. CFD is the current trend on automotive field in reducing the cost effect for analysis of various models on the basis of fluid flow. A multi-cylinder Maruti - Suzuki Wagon-R engine with maximum speed of 1500 rpm is taken for the analysis. The load and performance test is conducted. From the experiment backpressure and exhaust temperatures are measured. The mass flow rate and velocities are calculated. Flow through the exhaust manifold is analyzed using commercially available software with mass flow rate and pressure as boundary conditions.

Vivekananda Navadagi and SiddaveerSangamad [4]they analyzed the flow of exhaust gas from two different modified exhaust manifold with the help of Computational fluid dynamics. To achieve the optimal geometry for the low backpressure they have analyzed two different exhaust manifold, base geometry exhaust manifold and the modified geometry exhaust manifold. In the base model of the exhaust manifold the outlet is at side of the first inlet where as in the modified model of the exhaust manifold the outlet is at the centre of the exhaust manifold. Analysis has been done for the two different exhaust manifolds. The results were compared for the two models and it is found that the modified model gives low backpressure in comparison with other base model which ensures the improvement in the efficiency of the engine.

2. METHODOLOGY

2.1 Design of Exhaust Manifold

The five models are designed using the computer aided drawing software. The five models are analyzed using commercial CFD tool. The models are prepared and discretized using CFD-Grid generation tool. Basic CAD models used are shown with dimensions.

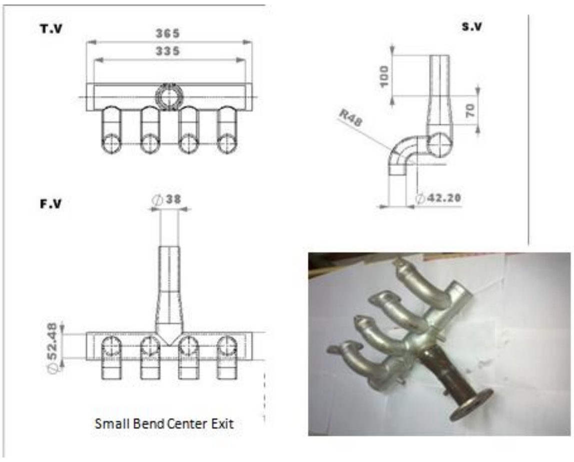

Fig-1: Dimension of the exhaust manifold

2.2 Design of Models 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.

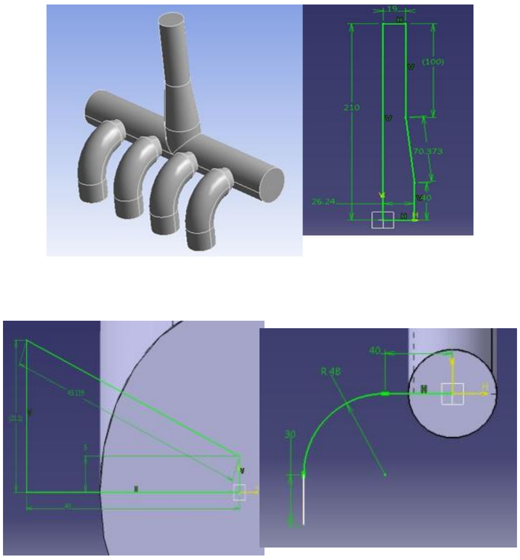

Fig-2: MODEL 1

Model 2

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

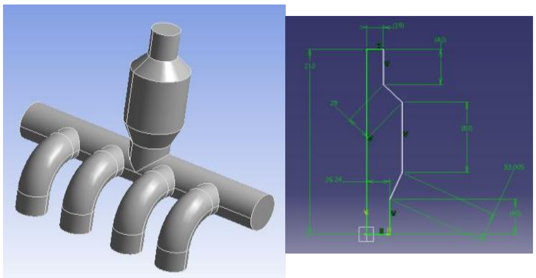

Fig-3: MODEL 2

Model 3

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

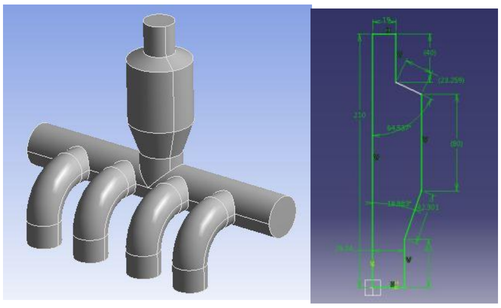

Fig-4: MODEL 3

Model 4

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

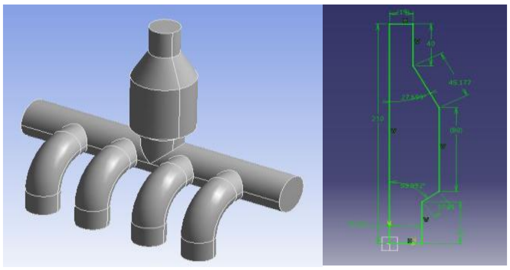

Fig-5: MODEL 4

Model 5

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

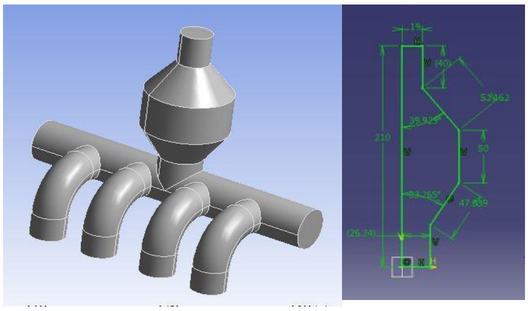

Fig-6: MODEL 5

2.3 Boundary conditions used

The boundary conditions used are Mass flow boundary conditions

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

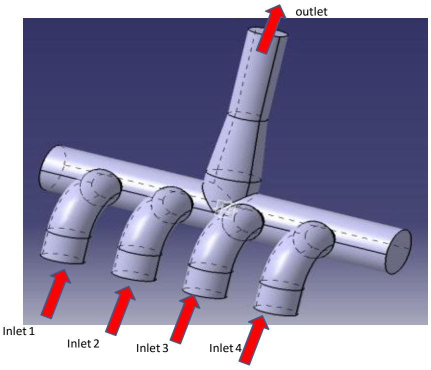

Fig-7: Boundary condition

Table -2: Boundary condition

Entity	**Zone**	**Zone Type**
Mass flow inlet	Boundary	Inlet
Pressure outlet	Boundary	Outlet
Boundary 1	Boundary	Wall-reduction
Boundary 2	Boundary	Wall-oxidation

2.4 Meshing

Figure 8 to 12 shows five meshed models

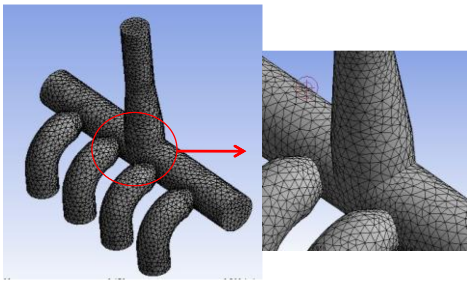

Fig-8: Meshed model 1

Total Number of Nodes = 2560
Total Number of Elements = 1540
Total Number of
Hexahedrons = 1540
Total Number of Faces = 1139
Aspect Ratio = 7
Orthogonal angle = 48.4

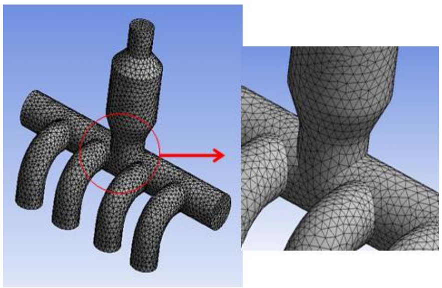

Fig-9: Meshed model 2

Total Number of Nodes = 2958
Total Number of Elements = 1789
Total Number of
Hexahedrons = 1789
Total Number of Faces = 1305
Aspect Ratio = 9
Orthogonal angle = 49.56

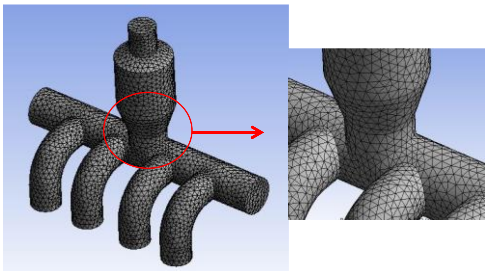

Fig-10: Meshed model 3

Total Number of Nodes = 2723
Total Number of Elements = 1347
Total Number of
Hexahedrons = 1347
Total Number of Faces = 1015
Aspect Ratio = 4
Orthogonalangle=39.56

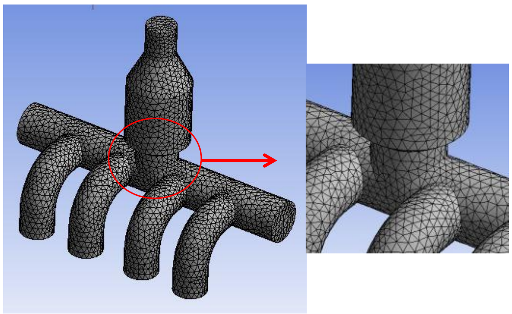

Fig-11: Meshed model 4

Total Number of Nodes = 2275
Total Number of Elements = 1066
Total Number of

Hexahedrons = 1066
Total Number of Faces = 1385
Aspect Ratio = 3
Orthogonal angle = 36.95

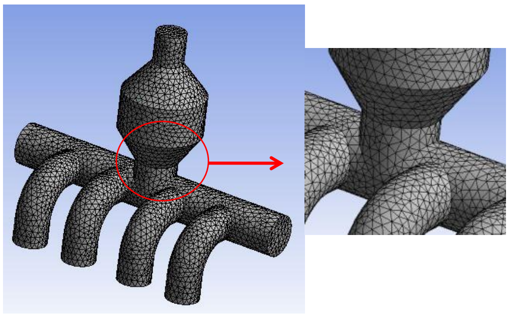

Fig-12: Meshed model 5

Total Number of Nodes = 3125
Total Number of Elements = 1685
Total Number of Hexahedrons = 1685
Total Number of Faces = 1535
Aspect Ratio = 8
Orthogonal angle = 47.58

2.5 Planes created for Postprocessing

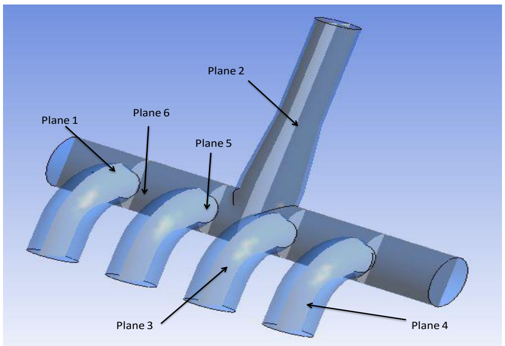

Fig-13: Planes for postprocessing

Totally six planes are created for the analysis of the exhaust manifold as shown in the above figure 13. For better understanding of the results and all results are discussed with reference to these planes.

3. RESULTS AND DISCUSSIONS

3.1 Results of the model 1

(a) Along plane 1

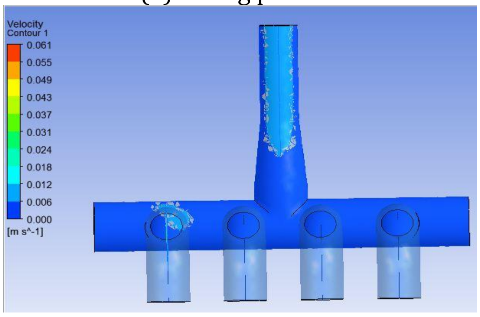

Fig-14: Velocity contour

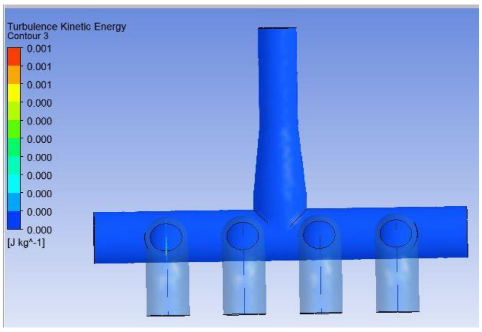

Fig-15: Turbulence Kinetic energy contour

From the fig. 14 it is evident that due to the convergent shape of the inlet, velocities are found to be lower at the connecting area and outlet. The low velocity results in high backpressure. It is observed that exhaust velocities are considerably decreases by designing the manifold using the convergent inlet. Low turbulence kinetic energy is observed throughout the flow is shown the fig 15.

(b) Along plane 2,3,4,5&6

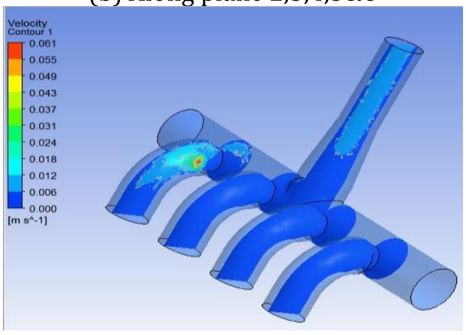

Fig-16: Velocity contour

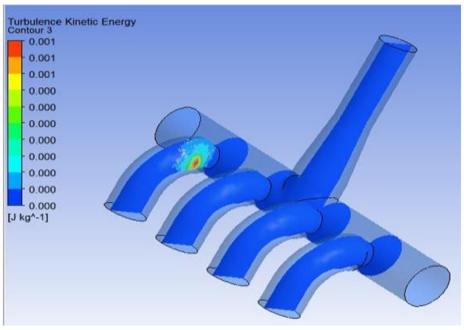

Fig-17: Turbulence Kinetic energy contour

Figure 16 represents the variation of velocity through the exhaust manifold along the plane 2, 3, 4, 5 and 6. This ascertains that exhaust velocities are considerably decreases by designing the manifold using the convergent inlet. Figure 17 gives the turbulence along the planes 2, 3, 4, 5 and 6. It is seen that turbulence is low in these design. At inlet 1 some bursts are seen.

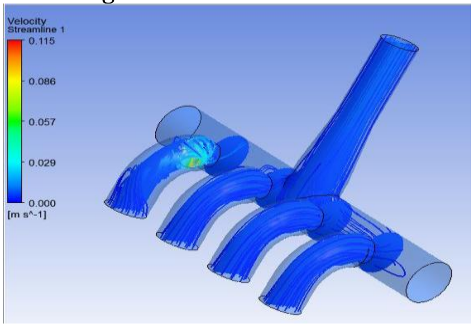

Fig-18: Velocity streamlines

Figure 18 gives the variation of flow of the exhaust gases. It is observed that exhaust gases circulates at the inlets 3 and 4 in this case.

(a) Along plane 1

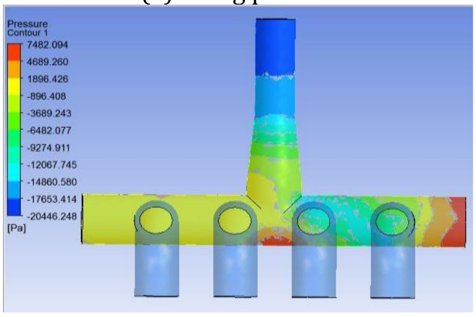

Fig-19: Pressure contour

(b) Along plane 2, 3, 4, 5&6

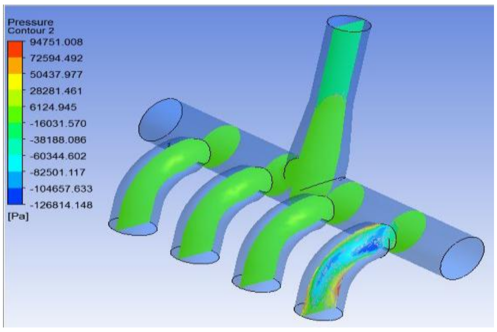

Fig-20: Pressure contour

Fig. 19 and fig. 20 gives the variation of pressure in the exhaust manifold for model 1 along plane 1 and along the planes 2, 3, 4, 5 and 6 respectively. It is seen that pressure at the outlet is less which leads to more backpressure.

3.2 Results of the model 2

(a) Along plane 1

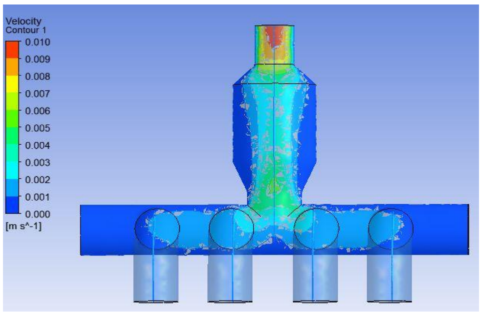

Fig-21: Velocity contour

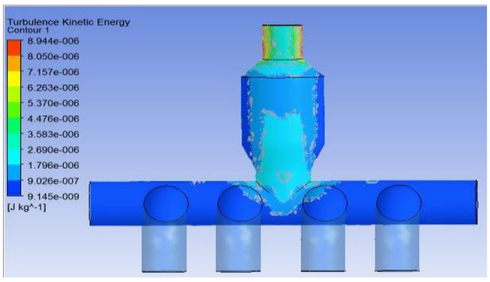

Fig-22: Turbulence kinetic energy contour

From the figure 21 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 increased by designing the exhaust manifold using this outlet in comparison with model 1. From the figure 22 it is evident that higher turbulence kinetic energy is observed at the outlet. At the inlets the turbulence energy is minimum.

(b) Along plane 2, 3, 4, 5 &6

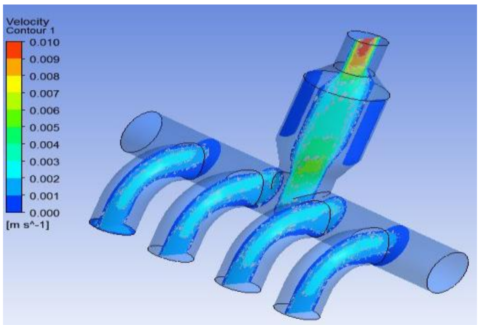

Fig-23: Velocity contour

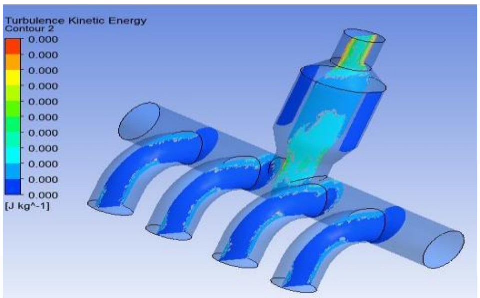

Fig-24: Turbulence Kinetic energy contour

Figure 23 gives the velocity contour along the plane 2, 3, 4, 5 and 6. It is observed that the velocity at the inlets is less and it increases at the outlet. Figure 24 gives the turbulence kinetic energy contour along the plane 2, 3, 4, 5 and 6. Turbulence energy increases from inlet to the outlet and the outlet turbulence energy is more.

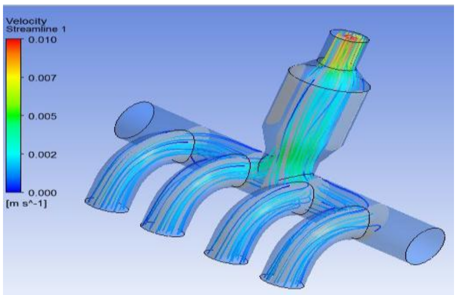

Fig-25: Velocity streamlines

Figure 25 gives the flow velocity of the model 2. It is observed that the velocity is slightly high at the outlet in comparison with the model 1.

(a) Along plane

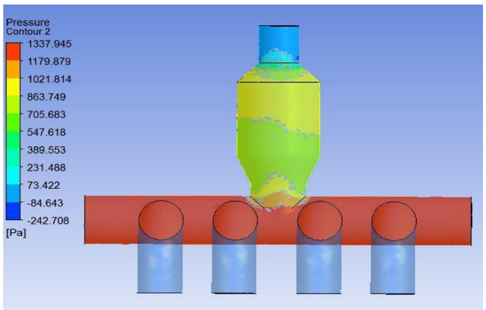

Fig-26: Pressure contour

(b) Along plane 2, 3, 4, 5 & 6

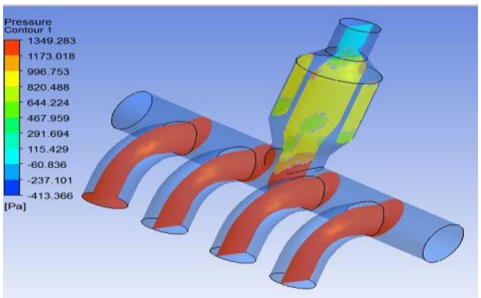

Fig-27: Pressure contour

Figure 26 gives the pressure contour along plane 1.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. Figure 27 gives the pressure distribution in the exhaust manifold along the plane 2, 3, 4, 5 and 6.

3.3 Results of the model 3

(a) Along plane 1

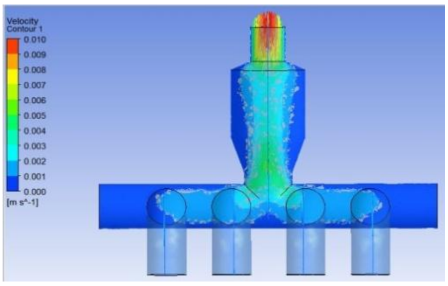

Fig-28: Velocity contour

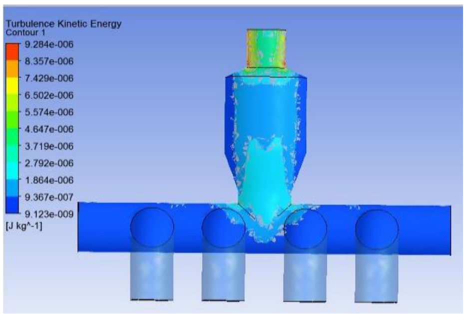

Fig-29: Turbulence kinetic energy contour

Figure 28 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 are considerably increases by designing the manifold using the divergent-convergent outlet. Figure 29 gives the turbulence kinetic energy contour. It is observed that the turbulence increases by decreasing the convergent length. Higher turbulence kinetic energy is observed at the connecting area and the outlet.

(b) Along plane 2,3,4,5&6

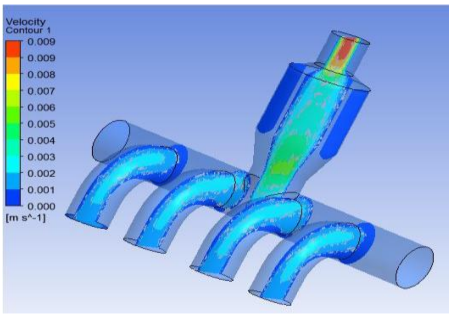

Fig-30: Velocity contour

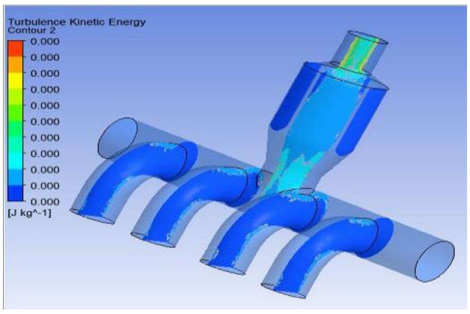

Fig-31: Turbulence kinetic energy contour

Figure 30 gives velocity contour along the plane 2, 3, 4, 5 and 6. It is observed that the velocity is higher at the outlet. Figure 31 gives the turbulence kinetic energy along the plane 2, 3, 4, 5 and 6.

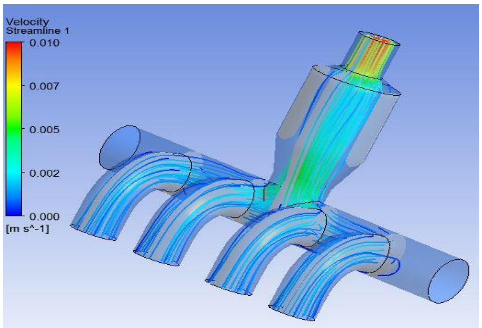

Fig-32: Velocity streamline

Figure 32 gives the velocity streamlines for the model 3. It is observed that the flow takes place without any recirculation. Velocities are found to be higher at the outlet and the connecting area.

(a) Along plane 1

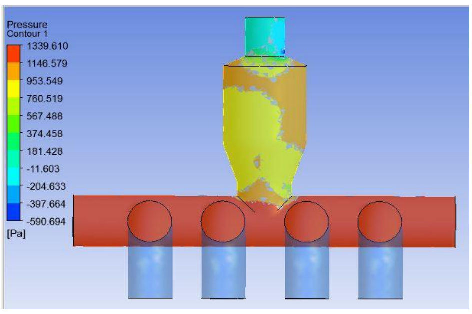

Fig-33: Pressure contour

(b) Along plane 2, 3, 4, 5&6

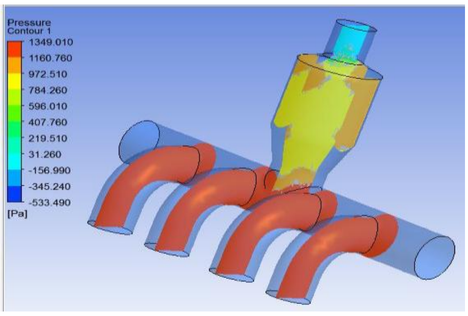

Fig-34: Pressure contour

Figure 33 gives the pressure distribution for model 3. It is found that the pressure is higher at the inlets. Pressure is higher at the outlet in comparison with the model 1 and 2. It is observed that exhaust pressure are considerably increases by designing the manifold by decreasing the convergent length of the outlet. Figure 34 gives the pressure contour along the plane 2, 3, 4, 5 and 6. It is seen that the pressure is higher in comparison with the model 1 and 2

3.4 Results of the model 4

(a) Along plane 1

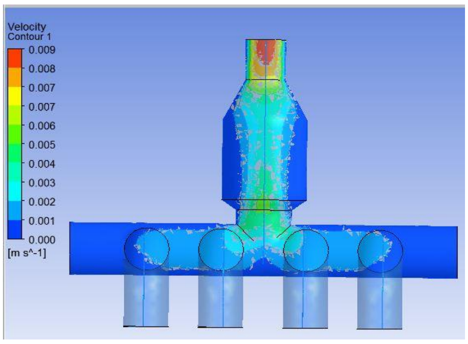

Fig-35: Velocity contour

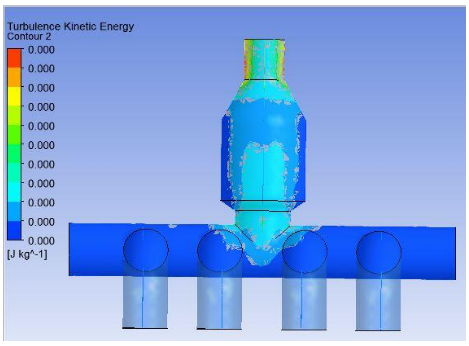

Fig-36: Turbulence kinetic energy contour

Figure 35 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 be higher at the outlet. Velocity at the outlet is higher in comparison with the model 1, 2 and 3. It is observed that the exhaust velocities are considerably increased by designing the exhaust manifold by reducing the divergent length of the outlet in comparison with model 1, 2 and 3. Figure 36 gives the turbulence kinetic energy contour of model 4. It is evident that higher turbulence kinetic energy is observed at the outlet in comparisons with the models 1, 2 and 3.

(b) Along plane 2, 3, 4, 5&6

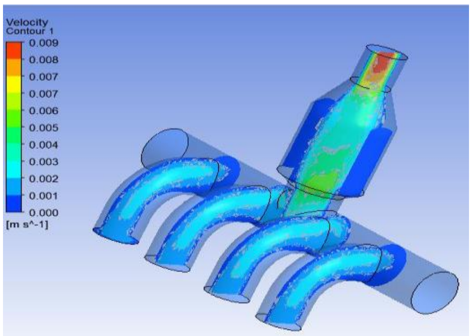

Fig-37: Velocity contour

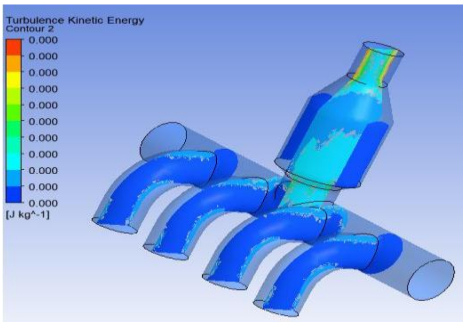

Fig-38: Turbulence kinetic energy contour

Figure 37 gives the velocity contour along the plane 2, 3, 4, 5 and 6. It is observed that the velocity is higher at the outlet in comparison with model 1, 2 and 3. Figure 38 gives the turbulence kinetic energy contour along the plane 2, 3, 4, 5 and 6. Turbulence kinetic energy is high at the outlet.

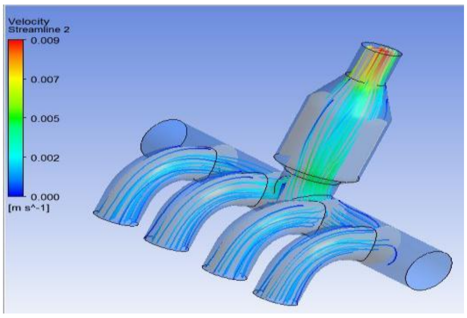

Fig-39: Velocity streamline

Figure 39 gives the velocity streamlines of the model 4. It is observed that the flow takes place without any recirculation. Uniform flows are observed in the model. Velocities are found to be higher at the outlet.

(a) Along plane 1

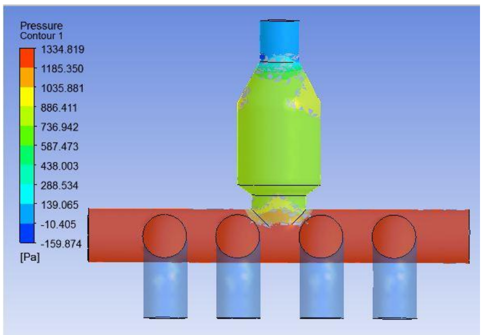

Fig-40: Pressure contour

(b) Along plane 2, 3, 4, 5 and 6

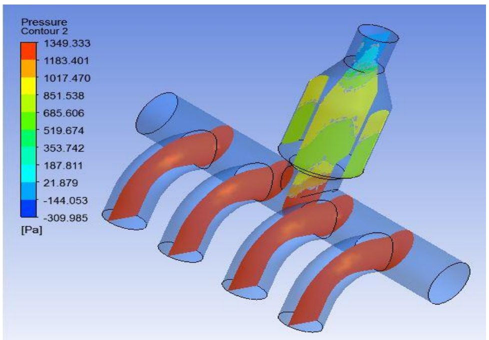

Fig-41: Pressure contour

Figure 40 gives the pressure distribution for model 4. It is found that the pressure is higher in comparison with the model 1, 2 and 3. Higher pressures are observed at the inlets. It is observed that by reducing the divergent length of the outlet the pressure increases considerably. Figure 41 gives the pressure contour of the model 4. Higher pressure is observed at the outlet in comparison with the model 1, 2 and 3.

3.5 Results of the model 5

(a) Along plane 1

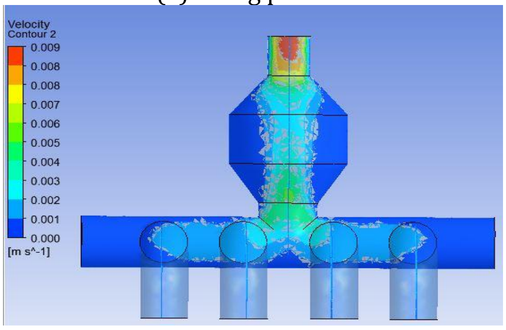

Fig-42: Velocity contour

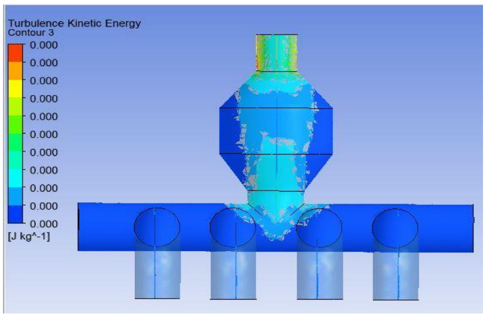

Fig-43: Turbulence kinetic energy contour

Figure 42 gives the velocity contour of the model 5. It is observed that due to the divergent convergent shape of the outlet velocity is higher at the outlet in comparison with the other models. It is observed that the exhaust velocities are considerably increased by designing the exhaust manifold by reducing the straight length of the outlet in comparison with other models. Figure 43 gives the turbulence kinetic energy contour of the model 5. It is observed that the turbulence is almost same in comparison with the other models.

(b)Along plane 2,3 ,4 ,5&6

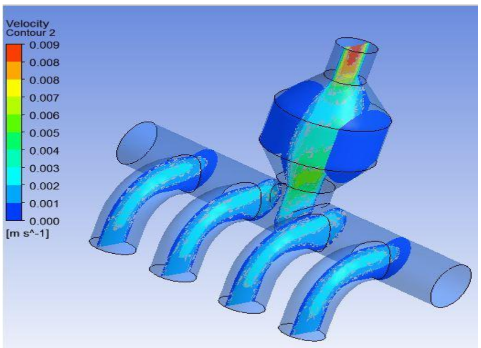

Fig-44: Velocity contour

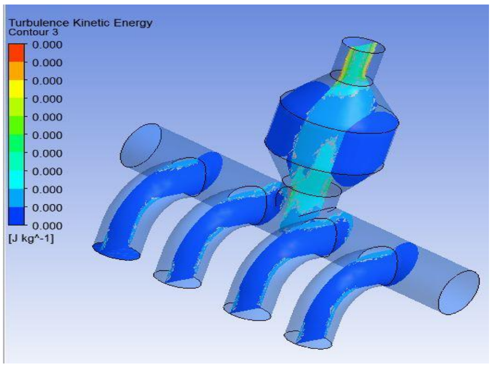

Fig-45: Turbulence kinetic energy contour

Figure 44 gives the velocity contour of the model 5 along the plane 2, 3, 4, 5 and 6. Higher velocity is observed at the outlet in comparison with the other models. Figure 45 gives the turbulence kinetic energy contour of the model 5 along the plane 2, 3, 4, 5 and 6. Higher turbulence kinetic energy is observed at the outlet.

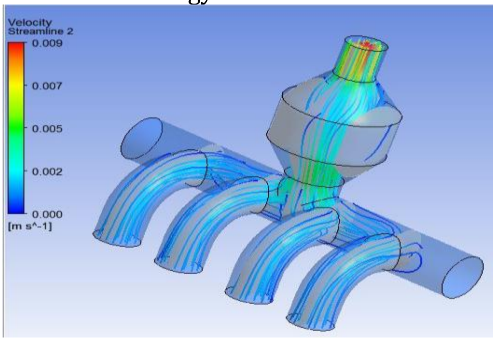

Fig-46: Velocity streamline

Figure 46 gives the velocity streamline of the model 5. It is observed that the flow is uniform in comparison with the other models. No recirculations are observed in the model. It is observed that by reducing the straight length of the outlet the velocity is higher.

(b) Along plane 1

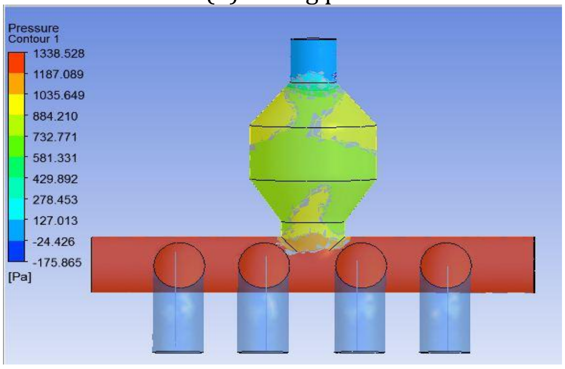

Fig-47: Velocity contour

(b) Along plane 2,3,4,5&6

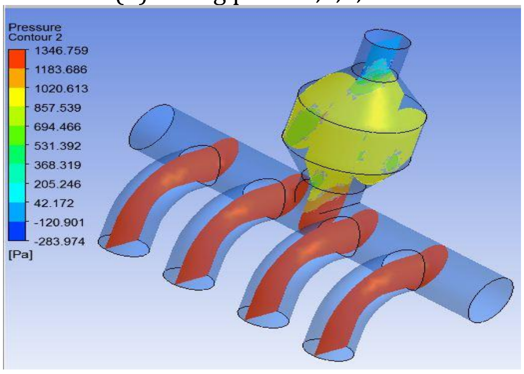

Fig-48: Pressure contour

Figure 47 gives the pressure distribution for model 5. It is found that the pressure is higher in comparison with the other model. Higher pressure is observed at the inlets. It is observed that by reducing the straight length of the outlet the pressure increases considerably. Figure 48 gives the pressure contour of the model 5 along plane 2, 3, 4, 5 and 6. Higher pressure is observed at the outlet in comparison with the other model.

4. CONCLUSIONS

In this work, different exhaust manifolds are analyzed with the help of commercial CAE software and flow of exhaust is observed and velocity and pressure distribution along the length of exhaust manifold is obtained through simulation. Five different models designed and results were analyzed through CFD Post processing. The use of different shapes of exhaust manifold helps in easy flow of exhaust.

- Model 5 facilitates easy flow of exhaust without recirculation and low backpressure at the exhaust outlet in comparisons with all other models.
- Turbulence kinetic energy is almost zero in the model 5 and hence the exhaust flows easily.
- Velocity at the outlet of model 5 is more and hence the backpressure reduces considerably.
- The optimum design for a exhaust manifold is Model 5 with 0.845 bar back pressure and outlet velocity 12.5m/s.
- The minimum backpressure and higher exhaust velocities are achieved by using exhaust manifolds with reducers, thus also reducing emissions.

REFERENCES

[1]. PL. S. Muthaiah, “CFD Analysis of catalytic converter to reduce particulate matter and achieve limited back pressure in diesel engine”, Global induced during suction and compression stroke l journal of researches in engineering A: Classification (FOR) 091304,091399, Vol.10 Issue 5 (Ver1.0) October 2010

[2]. K.S. Umesh, V.K. Pravin and K. Rajagopal “CFD Analysis and Experimental Verification of Effect of Manifold Geometry on Volumetric efficiency and Back Pressure for Multi-cylinder SI Engine” International Journal of Engineering & Science Research IJESR/July 2013/ Vol-3/Issue-7/342.

[3]. *Kulal et al.*(2013) “Experimental Analysis of Optimal Geometry for Exhaust Manifold of Multi-cylinder SI Engine for Optimum Performance” Global journal of researches in engineering A: Classification (FOR) 091304,091399, Vol.10 Issue 5 (Ver1.0) October 2010.

[4]. Vivekananda Navadagi and SiddaveerSangamadDevelopment of a Partial Filter Technology for Hdd Retrofit, Sae Technical Paper 2006-01-0213.Jacobs, T., Chatterjee, S., Conway, R., Walker, Development of a Partial Filter Technology for Hdd Retrofit, Sae Technical Paper 2006-01-0213.

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

[6] Chiavola*et al*, “Possibilities of the application of catalyst technologies for the control of particulate

emission for diesel vehicles", SAE Transaction 2001, paper no. 2001280044.

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

[8] 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.