OPEN ACCESS

International Journal of Innovative Technology and Exploring Engineering (IJITEE)
ISSN: 2278-3075 (Online), Volume-8 Issue-4, February 2019

Design and Air Flow Analysis in Intake Manifold with Different Cross Section Using CFD

K. Sathishkumar, R. Soundararajan, G. Dinesh, S. Surjith

***Abstract:** The air-fuel mixture is the important source in engine combustion, the transfer of air-fuel mixture is done by intake manifold in the intake system of the automobile. So due to improving the efficiency of the engine most of the automobile manufacturing industry is mainly focused on the intake manifold design for better performance. This project investigates about flow analysis taken in the different cross-section of an intake manifold. The main goal of our work is to change the cross-section of the intake manifold and analysis the model. In this paper, the manifold is designed by considering two different cross-section one is the circular outlet and the other is the convergent-divergent outlet and the created model is analyzed. The flow analysis is carried by using CFD and the values are compared and the best cross-section is concluded. The manifold with a convergent-divergent outlet has high velocity when compared with the circular outlet. The intake manifold can be used in automobile industries.*

***Keywords:** Air intake manifold, Convergent-divergent outlet manifold, Airflow, Fluent, Circular manifold, CFD.*

I. INTRODUCTION

In a vehicle, the transfer of air-fuel is taken by intake manifold to the combustion chamber, so hence the manifold is one of the critical parts in an automobile. The main function of the intake manifold is to distribute the air equally to each cylinder head at the stage of combustion. In most of the research papers to improve the engine performance, the dimension of the plenum is varied and the analysed is done using CFD. While modifying the design of the manifold the formation of turbulence should be strictly avoided. In this project, the outlet of the manifold is designed in different cross section and the air flow is analysed and the velocities are compared with the circular cross-section. In order to avoid the formation of turbulence the edges are filleted and the flow is carried out to check the turbulence formation.

The CFD test is carried in both the cross section manifold by considering three different cases one is by opening of all outlet of the manifold, the second condition is by taking 1&3 cylinders are open and 2&4 cylinders are in closed condition, and the final condition is by taking 1&3 cylinder is in closed Condition and 2&4 cylinders are in open condition. Each condition is tested and the values are discussed and the best cross-section is concluded.

II. LITERATURE REVIEW

The intake manifold is designed with fillet on the corner in order to avoid the formation of turbulence, by using CFD the existing model and the proposed model is tested to check the flow of air inside the manifold, the author had concluded that the proposed will distribute the pressure evenly and it may also neglect the formation of turbulence in the manifold the velocity vector is compared in both the models[1].CFD analysis is done on Ford two valve production engine, and the author had done the same test in experimentally by using Ricardo MK-3 engine and he concludes the result by comparing both experimental and software results and summarize the manual method is a worthwhile goal [2,]. An experimental test on intake manifold is conducted for taking the engine performance such as brake power, torque, specific fuel consumption, thermal efficiency, brake indicated power and $C0_2$ Emission, as a result, the intake plenum volume increased, intake manifold pressure is also increased, and the mixture became Leaner and pollutant is decreased due to the increase in relative air ratio [4]. In this paper, the author had the design and manufacture of an intake system for the 600cc formula society of automotive engineer's engine. Owing to the inherent geometric constraints imposed by the existing manufacturing process, the manufacturing of inlet manifold is done by FDM by consist of the plenum, plenum elbow, and Engine. The main aim of the work is to enhance the charge distribution, for improving the torque in overall process by using low-density materials such as hose clamps and silicon couples in the production of the aluminum counterpart [6]. The author had done a flow test on the numerical created model of the intake manifold, the CFD analysis is carried by taking standard kee turbulence model, by solving the Navier stroke equation in the CFD flow work. And the author has also done experimental measurements to test the brake power, brake torque, and brake thermal Efficiency. As a result of the proposed model, there is an increase in 16% in brake power, 13.9% improvement in brake torque and finally 12.5% increase in their brake thermal efficiency. Due to the improvement in their parameters, there is a decrease in 28% of brake specific fuel consumption (BSFC). So the author had concluded that the proposed design will give better performance [7].

Manuscript published on 28 February 2019.
*Correspondence Author(s)
K.Sathishkumar, PG Student, Department of Mechanical Engg, Sri Krishna College of Engineering and Technology, Coimbatore, TN, India.
R.Soundararajan*, Associate Professor, Department of Mechanical Engg, Sri Krishna College of Engineering and Technology, Coimbatore, TN, India.
G.Dinesh, UG Student, Department of Mechanical Engg, Sri Krishna College of Engineering and Technology, Coimbatore, TN, India.
S.Surjith, UG Student, Department of Mechanical Engg, Sri Krishna College of Engineering and Technology, Coimbatore, TN, India.

Design and produce a spiral intake system by additive manufacturing technique and to place the spiral intake manifold in the engine to do necessary experimental techniques by installing the engine on the test rig and compared with the old manifold system, as a result of spiral manufacturing product the volumetric efficiency was increased this is because of the size and shape of the manifold,

This the spiral intake manifold plays a direct impact on the volumetric efficiency [8]. Three different kinds of intake manifold are designed and analysed and compare their results and also done an experiment analysis and conclude the best design series in this paper. The intake manifold with vertical intake will give the better performance and the EGR distribution is also increases by 14% when it is compared with horizontal intake and the experimental results are more nearest to the CFD analysis results [11]. In this paper, the intake manifold is designed using solid works and the model is analysed using Ansys CFD. In most of the cases, the analysis is done only by changing the plenum length and area of the intake manifold, the current project is based on keeping the manifold plenum length as constant and the outlet cross section is changed and the analysis is done by keeping the boundary condition as same in all models.

III. DESIGN OF AIR INTAKE MANIFOLD

The 3D model of the intake manifold is designed using Solid works2016 software. For create a model of the intake manifold, the following things should be considered:

1. Uniform distribution of air pressure to all the engine cylinders in order to overcome the uncompleted combustion.
2. Proper designs of intake manifold profile helps to reduce the sudden raise in pressure waves which improve induction process and also eliminate the unnecessary turbulence and eddies inside the intake manifold.

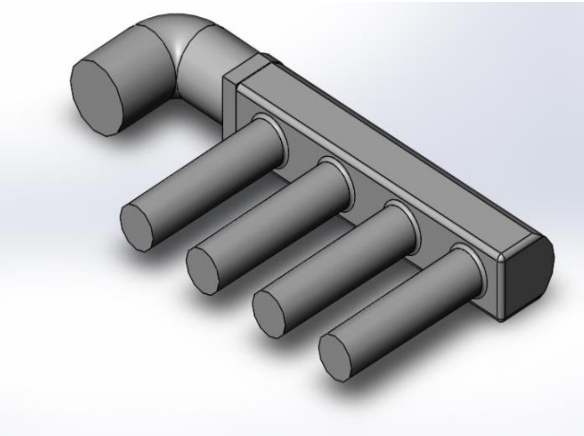

Fig. 1 Manifold with Circular outlets

In this design the inlet valve is taken as 40mm diameter, the outlet valve diameter is taken as 25mm and the plenum length is taken as 100mm, this is for a circular manifold and for the convergent-divergent manifold the inlet and plenum length is taken as same and the outlet valve is taken as a convergent-divergent type. In this the convergent angle is given as 12^o and the divergent angle is taken as 6^o and the throat diameter is given as 20mm, so by this input parameter, the two different types of the manifold are designed using solid works 2016.

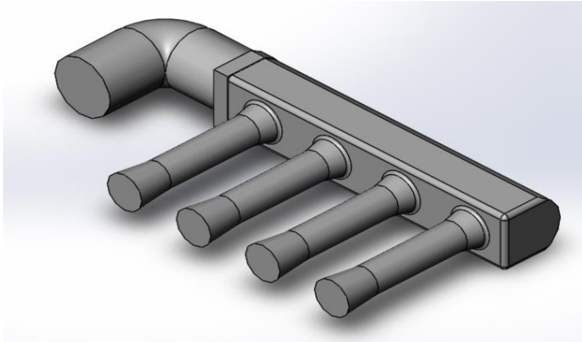

Fig. 2 Manifold with Convergent-Divergent outlets

IV. ANALYSIS USING CFD FLUENT

The air intake manifold is designed and the designed model is imported into Ansys software for doing analysis, in this by using fluent the air flow analysis is done on Ansys workbench 18.2. The three different cases of the condition are taken to do fluent analysis they are,

1. All outlets are open
2. Outlets 1,3 are open and Outlets 2,4 are close
3. Outlets 1,3 are close and Outlets 2,4 are open

A computational fluid element (CFD) is the utilization of mathematical equation, material science, and computational programming to show how a gas or fluid streams in any area. Computational fluid elements based upon the Navier-Stokes conditions. These conditions depict how the speed, weight, temperature, and thickness of a moving fluid are associated. By solving the input parameters using Navier-strokes equations the required outputs are taken. So all the analysis is done and each analysis is explained and the values are tabulated below. To check the flow of the fluid on the intake manifold the following boundary conditions are considered on the CFD fluent, the fluid used for the analysis is taken as air, the density of the fluid is 1.225 kg/m^3, the viscosity of the fluid can be taken as 1.7894x10^-5 kg/m-s, the outlet pressure is taken as 1 atmospheric pressure, and the temperature of the fluid is given as 300k and the solver is selected as pressure based solver , the residual value or the convergent limit is given as 1e-6 and finally the K- ε standard wall function is taken as the Mathematical models for analysis. The boundary condition is given based on the application and the inlet velocity is taken as 30m/s. After all the necessary conditions are given the total number of iteration is mentioned to start the analysis.

V. CFD SIMULATION

A. All Outlets are open

Once the meshing is completed the mesh file is imported in the fluent database, in the fluent database the mesh size, mesh quality, and mesh aspect ratio are checked and the necessary input conditions are given. In this analysis, all the outlets are taken as an open condition so the air flows from the inlet to all four-cylinder outlet equally.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

The flow analysis is done and the results are taken using CFD post-processing, the figure 3-4 gives the pressure contour of the manifold and the figure 5-6 gives the velocity contour of the manifold in open outlet condition.

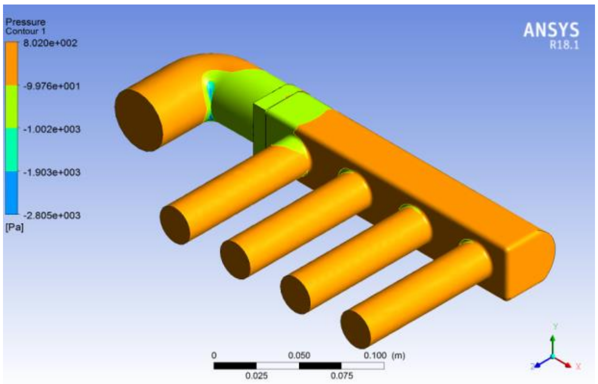

Fig. 3 Pressure Contour of Circular manifold with all open outlets

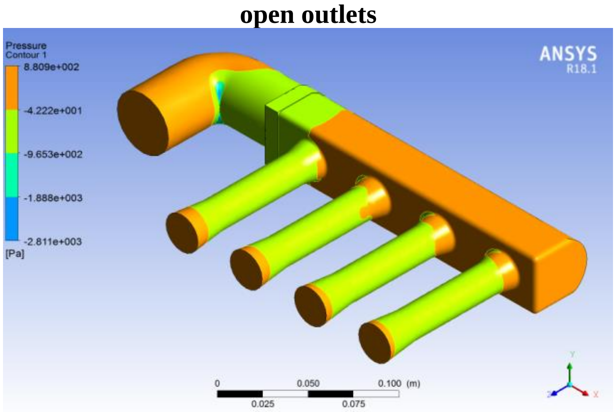

Fig. 4 Pressure Contour of Convergent-Divergent manifold with all open outlets

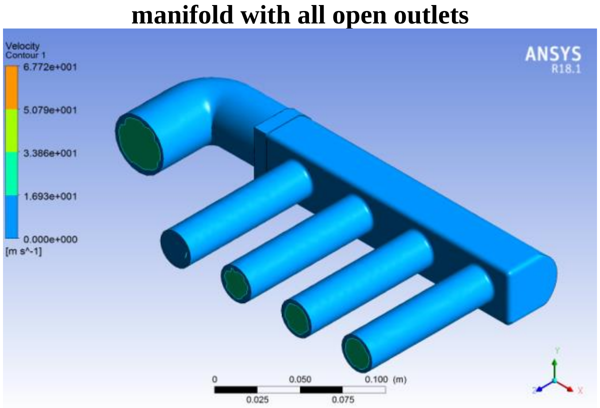

Fig. 5 Velocity Contour of Circular manifold with all open outlets

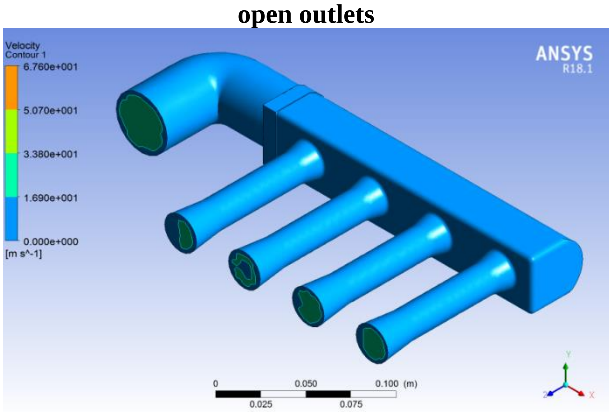

Fig. 6 Velocity Contour of Convergent-Divergent manifold with all open outlets

B. Outlets 1, 3 are open and Outlets 2,4 are close

In this condition the CFD analysis is done based on the sequence, that the outlets 1 & 3 are taken as open condition and the outlets 2 & 4 are taken as close condition in order to close the flow the outlets 2 & 4 are given as wall and the outlets 1& 3 are given as outlet in the named section. By giving all necessary boundary conditions the inlet velocity is given and the number of iteration is run using the fluent database. The flow analysis is done and the results are taken using CFD post-processing, the figure 7-8 gives the pressure contour of the manifold and the figure 9-10 gives the velocity contour of the manifold in the given condition.

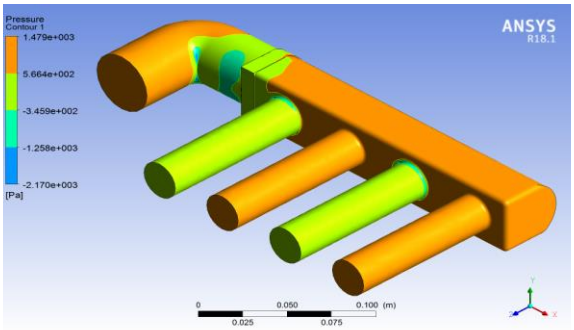

Fig. 7 Pressure Contour of Circular manifold with 1,3 open & 2,4 close

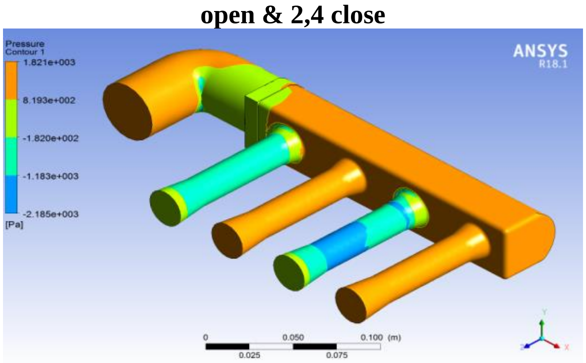

Fig. 8 Pressure Contour of Convergent-Divergent manifold with 1,3 open & 2,4 close

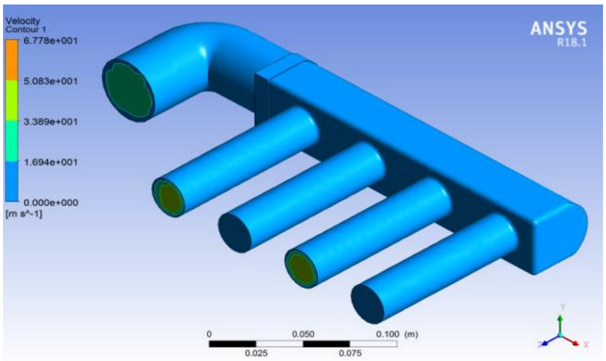

Fig. 9 Velocity Contour of Circular manifold with 1,3 open & 2,4 close

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

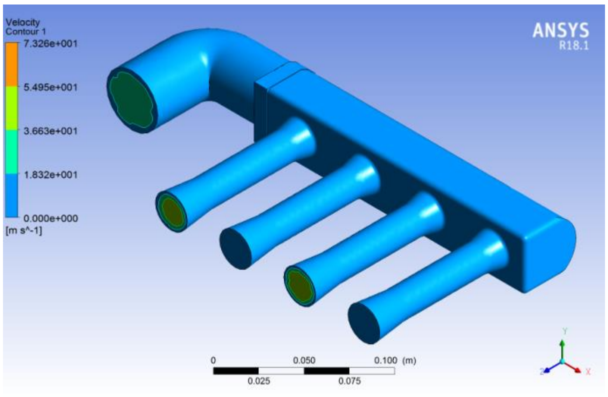

Fig. 10 Velocity Contour of Convergent-Divergent manifold with 1, 3 open & 2, 4 close

C. Outlets 1,3 are close and Outlets 2,4 are open

In this condition the CFD analysis is done based on the sequence, that the outlets 1 & 3 are taken as close condition and the outlets 2 & 4 are taken as open condition in order to stop the flow the outlets 1 & 3 are given as wall and the outlets 2 & 4 are given as outlet in the named section. By giving all necessary boundary conditions the inlet velocity is given and the number of iteration is run using the fluent database. The flow analysis is done and the results are taken using CFD post-processing, the figure 11-12 gives the pressure contour of the manifold and the figure 13-14 gives the velocity contour of the manifold in the given condition.

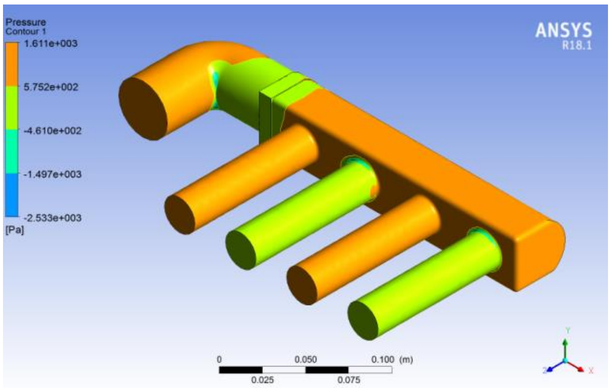

Fig. 11 Pressure Contour of Circular manifold with 1,3 close & 2,4 open

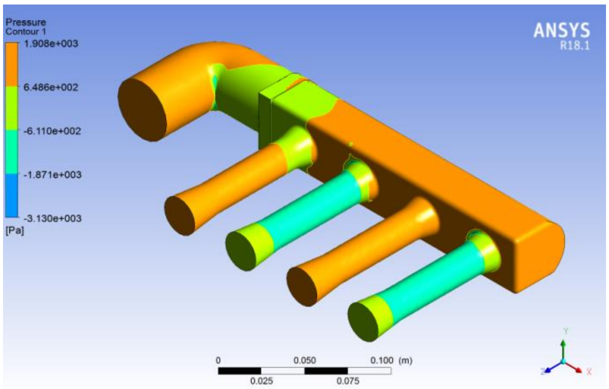

Fig. 12 Pressure Contour of Convergent-Divergent manifold with 1,3 close & 2,4 open

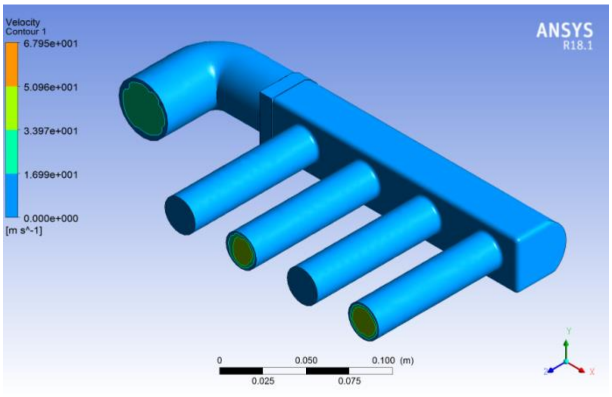

Fig. 13 Velocity Contour of Circular manifold with 1,3 close & 2,4 open

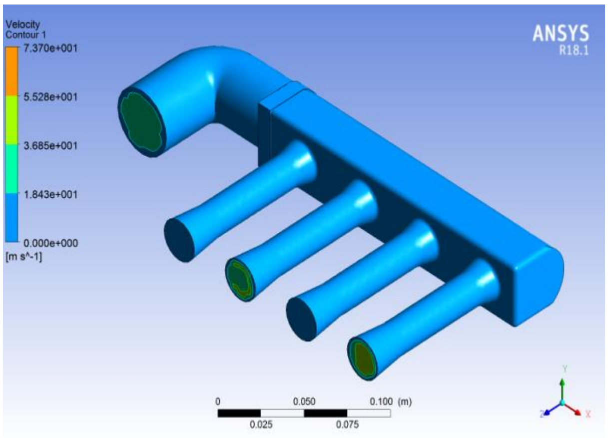

Fig. 14 Velocity Contour of Convergent-Divergent manifold with 1,3 close & 2,4 open

VI. RESULT AND DISCUSSION

Using CFD analysis flow characteristics are studied at the different cross-section of intake manifold the cylinder. The various parameters such as velocity, pressure are analysed. By using CFD post-processing the various results are taken at each domain and the values are tabulated below. The improvements in each manifold cross-section are compared by the results in the CFD post-processing.

Table 1 Pressure Contour of the Manifold

Conditions	Pressure in Pa	
	Existing Model	Proposed Model
All open outlets	802.007	880.9
Outlets 1,3 Open & 2,4 Close	566.4	819.3
Outlets 1,3 Close & 2,4 Open	575.2	648.6

*Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)*

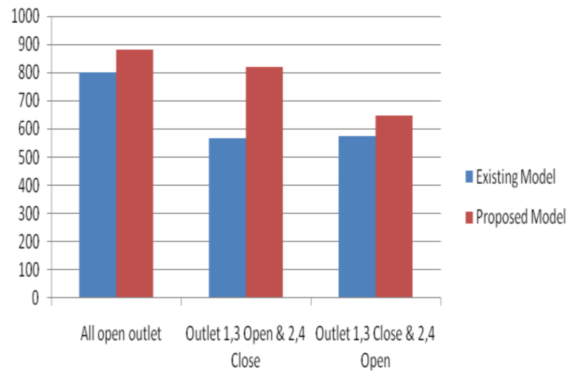

Fig. 15 Pressure Changes at Different Conditions

From the above figure, we can easily conclude that the pressure is changed at each condition based on the geometrical structure of the manifold. On seeing the table 1, we came to conclude that there is a rise in pressure on the proposed model when it compared with the existing model. This pressure difference will be taken due to the change in the area of the outlet in the manifold, in the convergent-divergent manifold the area is reduced in the convergent section and there is a constant area maintained in the throat section and the area is increased in the divergent section. So due to the increase in area on the divergent section when compared to the convergent section the pressure is increased in the divergent section of the outlet. On seeing the existing model of the manifold there is a constant area maintained on the outlet so there is no increase in pressure. By comparing both model the proposed model has a change in pressure difference.

Conditions	Velocity in m/s	
	Existing Model	**Proposed Model**
All open outlets	33.86	33.80
Outlets 1,3 Open & 2,4 Close	50.83	54.95
Outlets 1,3 Close & 2,4 Open	50.96	55.28

Table 2 Velocity Contour of the Manifold

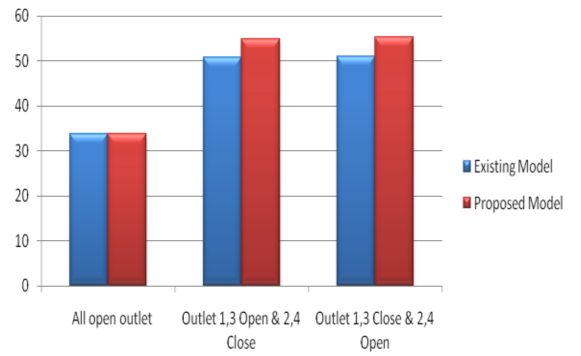

Fig. 16 Velocity Changes at Different Conditions

On seeing the velocity, table 2 will clearly explain the velocity of the manifold in each condition. In this, the velocity of the air is increased due to the increases in pressure, by comparing the result of both models there is an increase in velocity at the outlet of the manifold in the convergent-divergent model. In the existing model, there is a constant pressure is maintained but in the proposed model the pressure of the manifold is changed at each section, due to the changes the flow of air may be varied in the proposed model. So the proposed model gives more performance when compared to the existing model.

VII. CONCLUSION

Subsequent from reviewing work of various research authors as above it can be effectively presumed that intake manifold had an extensive effect on automobile engines. On seeing the results the CFD analysis is done in three different conditions in the second condition based on the firing order, the velocity of the air improves by 8.11% of the flow speed when compared with the existing design. In the third condition, the total velocity is improved by 8.48% from the existing value. We can clearly understand that intake manifold with convergent-divergent outlet has maximum velocity when it compared with the circular outlet. Our proposed design has achieved a better performance compared with the existing model. The small changes in velocity will give better combustion in the engine and it also improves the engine performance so we concluded that the convergent-divergent manifold outlet gives the best result.

REFERENCES

1. Anilkumar. D.B, Dr. Anoop Kumar Elia , Computational analysis of Intake manifold design of a four cylinder diesel engine in Technical research organization,5(4), 2018.
2. Wolf Bauer and John B. Heywood, Oshin Avanessian and Derlon Chu , Flow Characteristics in Intake Port of Spark Ignition Engine Investigated by CFD and Transient Gas Temperature Measurement in SAE technical paper series 961997.
3. V. Bellenger , A. Tcharkhtchi , Ph. Castaing , Thermal and mechanical fatigue of a PA66/glass fibers composite material in ELSEVIER International Journal of Fatigue 28 (2006) pp.1348–1352.
4. M.A. Ceviz , Intake plenum volume and its influence on the engine performance, cyclic variability and emissions in ELSEVIER Energy Conversion and Management 48 (2007) pp.961–966 .
5. M.A. Ceviz , M. Akin , Design of a new SI engine intake manifold with variable length plenum in ELSEVIER Energy Conversion and Management 51 (2010) pp.2239–2244 .
6. Ryan Ilardo , Christopher B. Williams, Design and manufacture of a Formula SAE intake system using fused deposition modeling and fiber-reinforced composite materials in Rapid Prototyping Journal, 16(3), pp. 174 – 179.
7. Mohamed Ali Jemni, Gueorgui Kantchev, Mohamed Salah Abid , Influence of intake manifold design on in-cylinder flow and engine performances in a bus diesel engine converted to LPG gas fuelled, using CFD analyses and experimental investigations in ELSEVIER Energy 36 (2011) pp.2701-2715.
8. A.Manmadhachary, M.Santosh kumar , Y.Ravi kumar , Design&manufacturing of spiral intake manifold to improve Volument efficiency of injection diesel engine byAM process in 5th International Conference of Materials Processing and Characterization (ICMPC 2016) pp. 1084–1090.
9. M. Safari , M. Ghamari and A. Nasiritosi , Intake Manifold Optimization by Using 3-D CFD Analysis in SAE technical paper series 2003-32-0073.
10. Robert M. Siewert, Roger B. Krieger, Mark S. Huebler, Prafulla C. Baruah and Bahram Khalighi and Markus Wesslau , Modifying an Intake Manifold to Improve Cylinder-to-Cylinder EGR Distribution in a DI Diesel Engine Using Combined CFD and Engine Experiments in SAE technical paper series 2001-01-3685.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)

11. Jianmin Xu , Flow analysis of engine intake manifold based on computational fluid dynamics in IOP Conf. Series: Journal of Physics: Conf. Series 916 (2017) 012043.
12. Jordan Lee, Lisa Roessler , Vibration Welded Composite Intake Manifolds Design Considerations and Material Selection Criteria in SAE technical paper series 970076.
13. Case Study On Plastic intake manifold in MATERIALS & DESIGN 13(6), 1992.
14. Ch. Indira Priyadarsini Flow analysis of intake manifold using CFD in International Journal of Engineering and Advanced Research Technology 2(1), 2016.
15. RepairPal Homepage http://repairpal.com/intake-manifold last accessed on 2017/09/18.
16. Yu, J., Vuorinen, V., Kaario, O., Sarjovaara, T., & Larmi, M.: Visualization and analysis of the characteristics of transitional under expanded jets. International Journal of Heat and Fluid Flow 44, 140-154 (2013).

AUTHORS PROFILE

K. Sathishkumar is currently doing his Master of Engineering in Engineering Design. His area of research includes design, analysis, and manufacturing methods. To his credit, he has published various research papers in reputed journals and he is a member in various professional bodies.

Dr. R. Soundararajan is currently working as an Associate Professor in Department of Mechanical Engineering. His area of research includes Design, Manufacturing technology and Optimization. To his credit, he has published various research papers in reputed journals and filed various patent. He is a member in various professional bodies.

G. Dinesh is currently doing his under graduation in Department of Mechanical Engineering. His research area includes Design and Manufacturing.

S. Surjith is currently doing his under graduation in Department of Mechanical Engineering. His research area includes Design and Analysis.

Published By:
Blue Eyes Intelligence Engineering
and Sciences Publication (BEIESP)