Design Optimization And CFD Analysis Of An Air Inlet System For SAE-Student Formula Racing Car

M.Dinesh Kumar, C.Yogesh, S.T.Nayab Hussain, R.Krishna, M.Sreenivasan.

Abstract: As per the rulebook of Formula SAE committee, India, it is must to have a restrictor for 20mm diameter fitted in the air inlet system of Student Formula SAE car's engine. The ultimate goal of installing a restrictor is to regulate the engine's power output and performance increased by regulating the engine's mass flow. This research paper is ultimately aimed on improving the layout of the air inlet system, which functions as a venturi pipe with a 20 mm diameter of the throat as the restrictor. Through increasing the pressure drop across the venturi, this allows for a total allowable mass flow rate into the tube. Through considering the converging-diverging angles of venturi, analytical calculations based on standard findings were conducted using Computational Fluid Dynamics (CFD) as a tool in ANSYS to achieve total mass flow speed and pressure fall throughout the venturi. Using the CFD method, it is observed from the analysis results that minimum pressure fall and peak output can be obtained through the venturi design by using the converging angle of 12 degrees and the diverging angle of 6 degrees.

Index Terms: Air-inlet system, ANSYS, CATIA, Computational Fluid Dynamics (CFD), Mach number, mass-flow restrictor, pressure, SAE, Venturi.

1 INTRODUCTION

Formula Student Car Racing is an event organized once a year by the Indian chapter of the Society of Automotive Engineers, where students form as a group or a team from various engineering colleges across the country, contesting to design, fabricate and run a model of an open formula racing car under multiple guidelines on vehicle design, dynamics and safety aspects of the vehicle. The event is judged by a jury of different automobile experts weighing a minimum of 1000 points which is distributed evenly across numerous static and dynamic cases. In their rulebook, the Indian SAE Formula Racing Committee enforced a rule that a restrictor having the diameter of 20 mm must be added to the air intake mechanism and that air flow into the engine must be achieved through this same restrictor, either for a single or multi-cylinder engine. The engines used in FSAE were limited to the size of610 cc in volume with a power output of 120 horse-power having a maximum of 15000 revolutions [1].The engine used for KTM Duke 390 is taken into consideration in this paper. It is a single cylinder engine having a volume of 373.2 cc fitted with 4 valves delivering a maximum power of 43 BHP @ 9000 rpm and providing a maximum torque of 37 NM @ 7000 RPM. Choosing the particular engine is because good power production is comparatively inexpensive. Nevertheless, the introduction of a 20 mm restrictor will decrease the air flow to the engine, it compensates for the drop in the pressure rate by increasing the air velocity. This paper therefore designs the intake manifold as in the form of a venturi meter for the engine, which the venturi throat acts as a restrictor [2].Therefore, mass flow at the restrictor exceeds its maximum speed of Mach number 1 with potential reduction of pressure drop across the venturi, enabling the engine to consume air with less drag.

- *The corresponding author M.Dinesh Kumar is currently working as Assistant Professor and as Department Research coordinator in Automobile engineering at PACE Institute of Technology &Sciences, Ongole, Andhra Pradesh, India.*
- *PH: +91 9965074145.*
- *E-mail: mdineshautomobile@gmail.com*

2 RESEARCH PROCEDURE

Research procedure employs the selection of appropriate restrictor that gives the best result output, usage CFD techniques, for calculating different angles of convergence and divergence of the venturi, which the best optimized angles that give up the best result were used. The working output efficiency is high by raising the order of restrictor, flow path and the runner.

2.1 Restrictor

A restrictor is a device that is fitted on the flow path that reduces the mass flow into the engine. In the engine intake manifold, it is possible to use two forms of restrictors, namely an orifice plate and a venturi duct. An orifice plate is a basic disk which incorporates a concentric hole in the middle, which the flow converges through the hole, which reduces the upstream pressure of the system that expands the flow by increasing velocity. Following Bernoulli's theory, the orifice effect can be measured by measuring the pressure and velocity difference.

2.2 Venturi

A venturi tube is a small pipe which incorporates a converging and diverging segment with a constricted narrow throat in the middle, which passes through a venturi drain, through a gap and at low pressure drawn into the engine. The pressure drop results in a rise in the speed of motion due to the increase of kinetic energy, which fulfils the theory of mechanical energy conservation [3]. The venturi effect can be accomplished by measuring the difference between pressure and velocity by integrating the Continuity formula using Bernoulli's theory.
The choice of restrictor considering few assorted parameters is given below in Table 1.

2.3 Choice of Restrictor

Taking into account the parameters specified in Table 1, it could be observed that it was comparatively better to fit a venturi as a restrictor than to fit an orifice, considering the discharge coefficient primarily [4]. As according to the SAE rulebook, venturi's throat is the 20 mm restrictor. The throttle body diameter of the engine considered in this paper is 46

TABLE 1

DIFFERENCE BETWEEN ORIFICE PLATE & VENTURI NOZZLE

Parameters	Orifice plate	Venturi nozzle
Co-efficient of discharge (C_d)	0.58 to 0.65	0.85 to 0.975
Pressure loss	Medium on a scale of high to low.	Low on a scale of high to low.
Accuracy	3 %	1 %
Viscosity effect	High	High
Manufacturing cost	Low cost	High cost
Manufacturing method	Compact	Complex

mm, which the throttle body is fitted into the inlet face of designed venturi, as the throttle body diameter would be served as inlet diameter and outlet diameter of the venturi nozzle.

2.4 Mass flow and Mach number

The constants and variables must be defined when considering venturi's model as a restrictor. In order to apply boundary conditions for CFD analysis, such parameters are required. By the information gathered from above discussion, we have diameters of the inlet, throat and outlet. Furthermore, the additional requirements are the venturi's converging angle and diverging angle. This paper describes three parameter that are known and two that are uncertain. The application of boundary conditions of variables such as pressure and velocity is a dynamic approximation method which is a complex calculation. This paper therefore deals with the estimation of the outlet mass flow rate of the venturi using the chocked mass flow rate equation [5]. Choked flow is a state in fluid dynamics consistent with the Venturi effect. Choked flow is a restricting phenomenon where the mass flow for a given upstream pressure and temperature will not increase with a further decline in the downstream pressure environment.

Mass-flow chocking equation is given by:

$$\dot{m} = r \times V \times A \quad (1)$$

For an ideal compressible gas, maximum mass flow rate at M = 1 (when flow is chocked) is given by:

$$\dot{m} = [(A \times P_t)/\sqrt{T_t}] \times [\sqrt{\gamma/R}] \times [(\gamma+1)/2]^{-[(\gamma-1)/(2\gamma-2)]} \quad (2)$$

Where $\dot{m}$ is mass flow rate,
A is the area which is 0.001256 m2. (20 mm restrictor),
P_t is the total pressure which is 1.01325 Bar at ambient condition,
T_t is the total temperature which is 300 K.
γ is the specific heat ratio which is 1.4.
R is the gas constant which is 0.287 KJ / Kg-K.
Using the above equation, calculating using the known parameters we get, Mass flow rate as

$$\dot{m} = 0.0703 \text{ Kg/s}$$

Where M is the Mach number, which is considered as significant parameter in a compressible flow stream. Mach number is a dimensionless quantity defined as the ratio of velocity of flow that past a boundary to the speed of sound in the medium. It is important to use the Mach number to categorize compressible flows into various Mach number regimes. It is given by: M = V/C

Where V is the flow velocity and C is the speed of sound in the medium. Categorization of Mach number is given in figure 1.

TABLE 2

DIFFERENT ITERATIONS FOR DESIGNED VENTURI

Iteration number	Converging angle (degree)	Diverging angle (degree)	Pressure difference (Bar)
1	10	6	3.05
2	12	6	1.62
3	14	6	2.98
4	16	6	2.92
5	18	6	2.87

Figure 1 describes that during initial subsonic upstream conditions the conservation of the mass theorem involves a raise in the velocity of the flow as it flows through the constricted narrower cross-sectional area. The Venturi effect causes a reduction in the residual stress and hence in the intensity at the constriction. It is evident from the figure above that the application of converging-diverging nozzle decreases the air density going through the restrictor. This results in pressure reduction at downstream side of the designed venturi.

3 DESIGN & ANALYSIS

3.1 Design of Venturi

The venturi design is performed using CATIA. The designed venturi is structurally meshed utilizing quad elements consisting of 2560 and 1806 component nodes in numbers [7]. The mesh geometry file is therefore imported using ANSYS to determine the inlet and outlet wall boundary conditions for CFD analysis. Many number of iterations considering several converging-diverging angles are carried out to find the optimal result that provide the minimum pressure drop [10].Among number of iterations considered, table 2 provides the optimal five iterations that is chosen based on various studies. Among the five iteration, which give the minimum pressure difference is resulted the optimal venturi design.

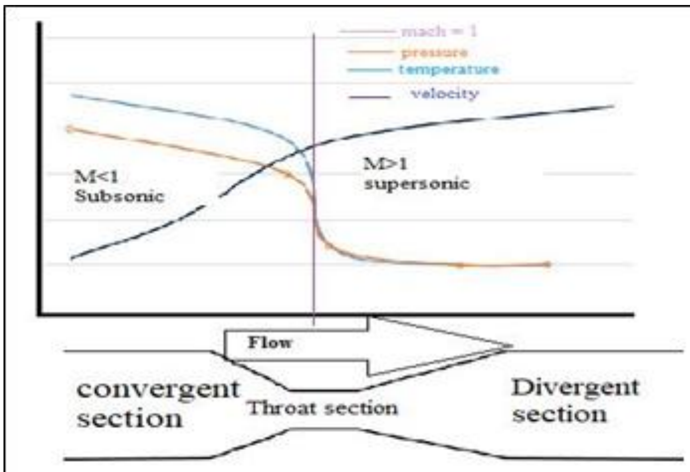

Fig.1. Categorization of Mach number

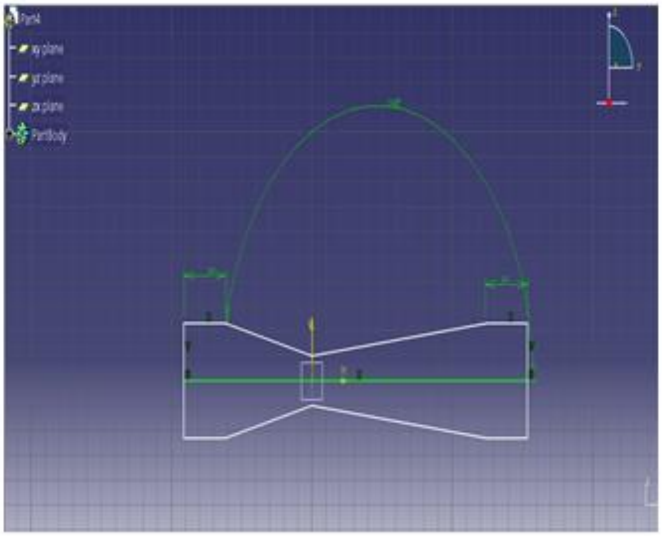

Fig.2. 2D model of the designed venturi

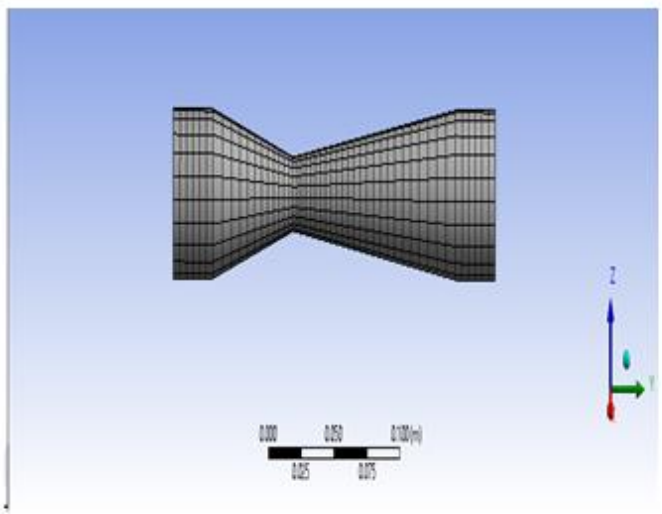

Fig.3. Mesh geometry

3.2 Analysis

The CAD model of designed venturi was imported to ANSYS for analysis. Mesh geometry was of he designed venturi was created. Finite numbers of nodes and elements have been defined in order to achieve précised resultant values of analysis. Many number of iterations considering several converging-diverging angles are carried out to find the optimal result that provide the minimum pressure drop [10].Among number of iterations considered, table 2 provides the optimal five iterations that is chosen based on various studies. Among the five iteration, which give the minimum pressure difference is resulted the optimal venturi design.

4 RESULTS AND DISCUSSIONS

For the simulation of calculated mass flow of 0.073 Kg/s, the iterations given in table 2 are performed by specifying the mesh geometry of fine quality using ANSYS which gives different results for the same iteration respectively. Pressure contour for iteration 1, 2, 3, 4 and 5 are given in the figure 4, 5, 6, 7 and 8 respectively.

TABLE 3

PRESSURE DIFFERENCE FOR DIFFERENT ITERATIONS

Iteration number	Converging angle (degree)	Diverging angle (degree)	Throat Diameter (mm)
1	10	6	20
2	12	6	20
3	14	6	20
4	16	6	20
5	18	6	20

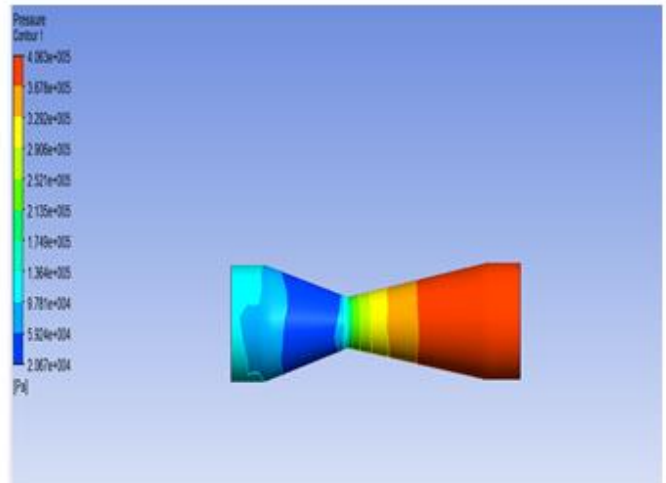

Fig.4. Pressure plot for iteration 1

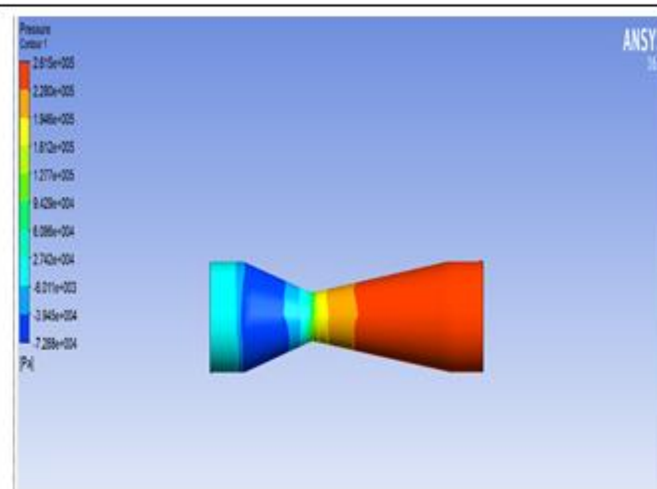

Fig.5. Pressure plot for iteration 2

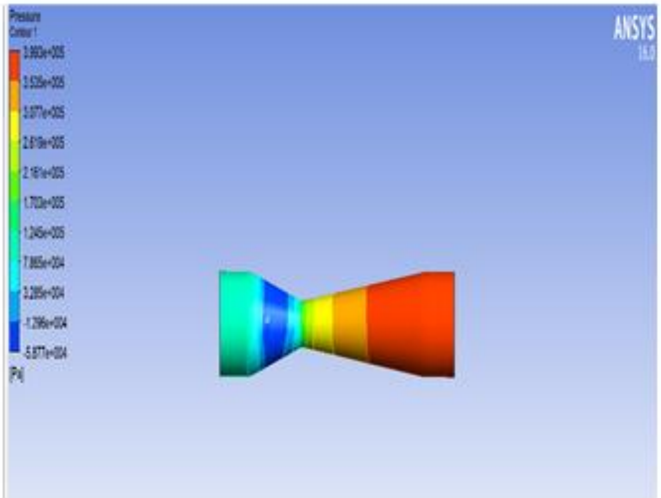

Fig.6. Pressure plot for iteration 3

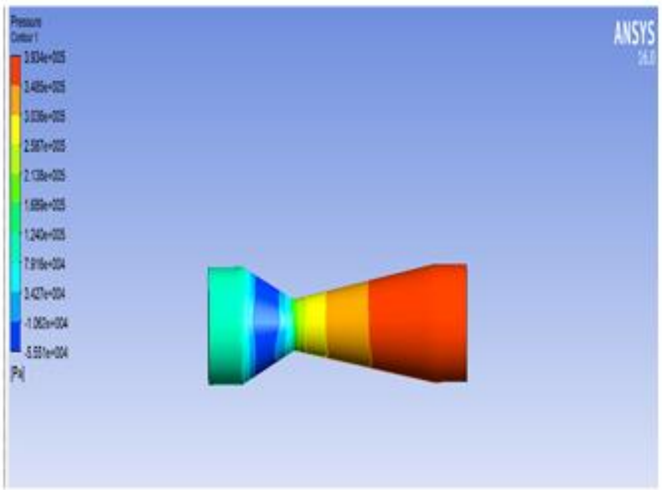

Fig.7. Pressure plot for iteration 4

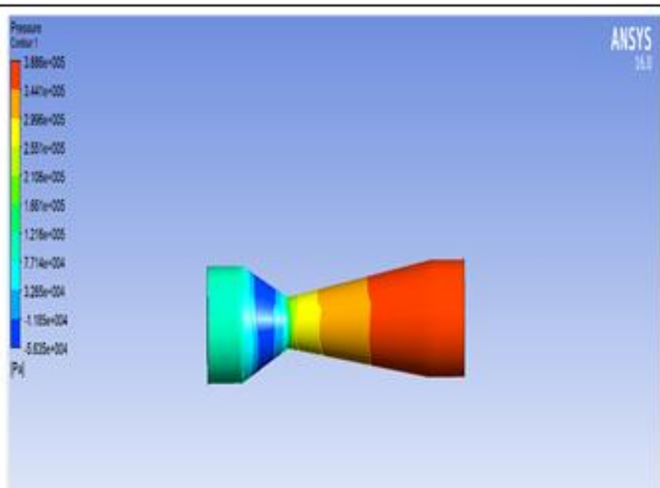

Fig.8. Pressure plot for iteration 5

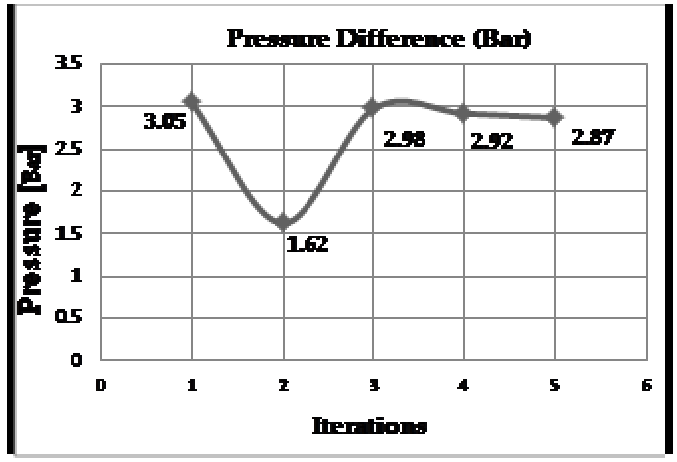

Fig.9 Pressure difference for respective iteration

The resulted pressure difference for different iterations is listed in the table 3 below. Table 3 reveals that second iteration that has converging-diverging angles of 12 degrees and 6 degrees respectively has the minimum pressure difference of 1.62 Bar, through the venturi. Figure 9 gives the tabulation for pressure difference for the respective iterations, which shows that iteration 2 has drastic difference in pressure drop in contrast with rest of the other iteration.

5 CONCLUSION

This paper examined the pressure performance for a venturi model with a 20 mm diameter throat as the restrictor. Design of venturi is done using CATIA. CFD analysis was performed using ANSYS 16.0. The analysis concludes that for a 20 mm restrictor in the flow path, compared to an orifice plate, a venturi nozzle having converging angle and diverging angle of 12 degree and 6 degree respectively serves as an optimal design that provide a better mass flow rate to the engine with maximum 0.0703 Kg/s and minimum pressure loss of 1.62 Bar through the venturi nozzle.

6 REFERENCES

[1]. https://www.formulaasaeindia.org/pdf/supra-rule-book-2019.pdf (accessed as on 20.11.2019)

[2]. Venugopal, T. and Anubhav, R., 2019. A New Approach for Development of a High-Performance Intake Manifold for a Single-Cylinder Engine Used in Formula SAE Application. SAE International Journal of Engines, 12. (03-12-04-0027).

[3]. Ranganathan, S., Thiyagarajan, A.S., Kuppuraj, S. and Chandrasakaran, P., 2019. Design and Analysis of De-Lavel Restrictor with Throttle Body for Formula Vehicle (No. 2019-28-0009). SAE Technical Paper.

[4]. Rosetti, A., Iotti, C. and Cantore, G., 2019. Intake Manifold Primary Trumpet Tuning Options for Fuel Flow Limited High Performance ICE (No. 2019-24-0005). SAE Technical Paper.

[5]. Jain, S. and Patil, R.S., 2019, August. Numerical Simulation of Novel Nozzle Intake Manifold. In Journal of Physics: Conference Series (Vol. 1276, No. 1, p. 012037). IOP Publishing.

[6]. Mohamad, B., Karoly, J. and Zelentsov, A., 2019. CFD Modelling of Formula Student Car Intake System. Facta Universitatis, Series: Mechanical Engineering.

[7]. Gao, P., Zhu, H., Shan, H. and Lin, L., 2019. Numerical Simulation and Optimization Design of Intake Manifold. Journal of Changshu Institute of Technology, (5), p.22.
[8]. Singh, J., Sankaram, M.V.N., Naranje, V. and Salunkhe, S., 2019. Formula SAE Power Increment. In Innovative Design, Analysis and Development Practices in Aerospace and Automotive Engineering (I-DAD 2018) (pp. 249-256). Springer, Singapore.
[9]. Ahuja, M., Saini, I.S. and Singh, A., Design optimization for air intake system of formula student race car. International Journal of Aerospace and Mechanical Engineering Volume 5 – No.2, March 2018.
[10]. Kumar, M.V. and Murugesan, S., 2018. Analysis and Development of Restrictor for Parallel Twin Engine in FSAE Cars. International Journal of Engineering & Technology, 7(3.6), pp.311-314.
[11]. Prasetyo, B.D., Ubaidillah, Maharani, E.T., Setyohandoko, G. and Idris, M.I., 2018, February. Computational studies of an intake manifold for restricted engine application. In AIP Conference Proceedings (Vol. 1931, No. 1, p. 030035). AIP Publishing.
[12]. Bhatnagar, A., 2018. Air Intake System for SAE SUPRA, Formula Bharat. International Research Journal of Engineering & Technology, Vol 5, issue 9, (pp. 834-836).
[13]. Kumar, M.V. and Murugesan, S., 2018. Analysis and Development of Restrictor for Parallel Twin Engine in FSAE Cars. International Journal of Engineering & Technology, 7(3.6), pp.311-314.
[14]. Sayyed, A., 2017. Air Flow Optimization through an Intake system for a Single Cylinder Formula Student (FSAE) Race Car. International Journal of Engineering Research & Technology, 6, pp.183-188.
[15]. Melaika, M., Rimkus, A. and Vipartas, T., 2017. Air restrictor and turbocharger influence for the formula student engine performance. Procedia Engineering, 187, pp.402-407.
[16]. Vaz, J., Machado, A.R., Martinuzzi, R.K. and Martins, M.E., 2017. Design and Manufacture of a Formula SAE Variable Intake Manifold (No. 2017-36-0181). SAE Technical Paper.
[17]. Singhal, A., 2016. Designing & Validating a New Intake Manifold for a Formula SAE Car. Int J Eng. Res& Technol (IJERT), 5(07).