International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

Design and Analysis of Air Intake System of SAE Supra Student Formula Car

Vinayak Chauhan
ME Student, Department of Mechanical Engineering, Ajay Kumar Garg Engineering College, Ghaziabad, India

***Abstract*—This study takes a look at the design process of the air intake system of SAE Supra Student formula Car. Over the years, much of the design of this system had been carried out through an iterative trial and error process, so the study attempts to identify the scientific and engineering principles pertaining to the design of this system. The intake system is being subdivided into various components, and the relevant principles will be discussed. Following that, data is collected from the engine cylinders, cam-profile, intake valves etc. and a simulation model of the air intake system will be developed. This model is then being applied, sequentially, to the various components. Flow analysis for individual components are carried out, and verified against performance simulations of the entire engine system, followed by physical testing of several of the components using simulation softwares like acusolve and field view.**

***Index Terms*—Venturi, Restrictor, Plenum, Runner, Flow Analysis**

I. Introduction

The basic function of the intake manifold is to get the air from the carburetor or throttle body directed into the intake ports. A great intake manifold design can provide substantial performance advantages than a less optimal one. Design goals to be met by the intake manifold

- Low resistance to airflow.
- High air velocity for a given flow rate.
- Excellent fuel and air distribution throughout.

To achieve desired power and torque, air flow characteristics matter in respect to naturally aspirated engines. Getting air into an engine is the key to making power. As per the rules of SUPRA SAEINDIA and FORMULA STUDENT competition, there is a 20mm restrictor present between throttle body and engine, to limit engine power capability. To achieve stagnation of air, plenum is used. Runner connects the plenum with engine and is tuned at certain rpm to optimize engine performance.

II. Scope

- To optimize design of convergent- divergent type restrictor.
- To optimize plenum shape for having least flow resistance and maximum air flow velocity
- To obtain optimum plenum volume.
- To obtain optimum runner diameter.

Mass Flow Rate Constant throughout the intake system.

III. Research Methodology

The basic design parameters to be considered for the air flow and Computational Fluid Dynamic (CFD) analysis, is the design constraints due to conglomerate of the rear part of the chassis and other various rules to be followed. This air intake manifold is placed between the air filter and the throttle body of the engine and hence the inner diameter at the inflow and outflow of the nozzle is pre-determined and cannot be changed. The length of this body cannot be extended after a certain magnitude, since it will provide hindrance to the adjacent parts and must be within the height of the main roll hoop of the chassis. Thus the CFD analysis is carried out for the location of the neck, length and angle pertained in order for it to not have air resistive vortex regions and to reduce the drag flow. Any CFD problem is solved by putting the appropriate boundary conditions to the Navier's stokes equation. Using Finite volume methods, the equations are recast in conservative form, and then solved on every discrete control volume. As there is no turbulence involved in the system, there is no involvement of Reynolds stresses and eddy currents. Thus the Navier stoke equation can be solved by treating the system as boundary value problem. Selection of the appropriate boundary conditions are very important in the case of solving boundary value problems, otherwise leads to numerical instability of the system.

IV. Research Procedure

Research procedure consisted of appropriately calculated steps which at every instance provide proper connectivity between the last and next step involved in research of venturi. Following are the steps to ensure efficient working of entire research.

A. Restrictor

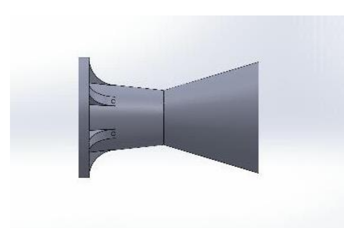

Fig. 1. Restrictor

For convergent section, both the end diameters are constrained (48mm of throttle body and 20mm of the restrictor).

For divergent section outlet diameter is 42mm. Considering parameters such as Pressure Difference at inlet and outlet of restrictor.

V. MATH

The 2 major things were required while designing the restrictor,

- High velocity at the throat.
- Minimum pressure between the inlet and outlet.

For this a number of designs have been tested with various convergent and divergent angle.

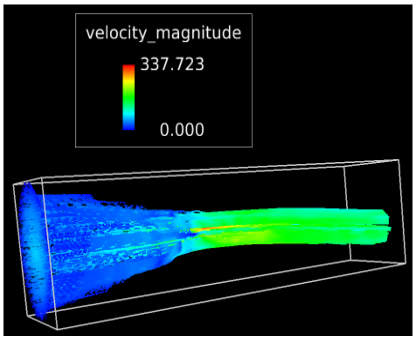

Fig. 2. Converging angle- 14º Diverging angle- 8º

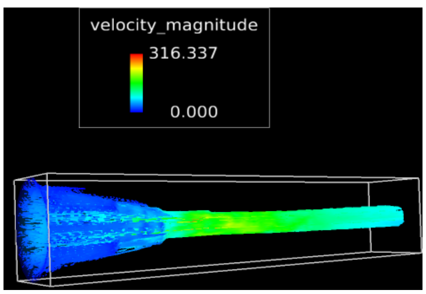

Fig. 3. Converging angle- 14ºDiverging angle- 6º

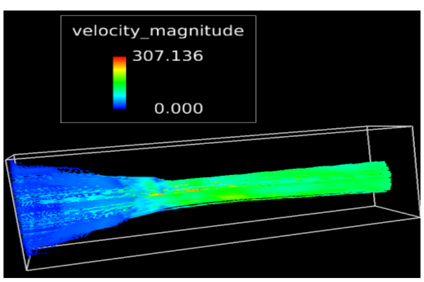

Fig. 4. Converging angle- 16º Diverging angle- 8º

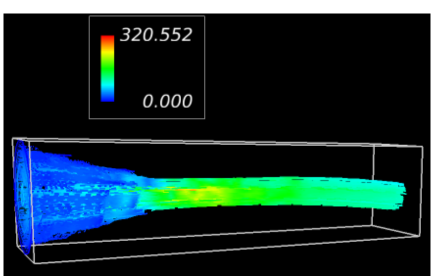

Fig. 5. Converging angle- 16ºDiverging angle- 6º

So the final restrictor angles which were selected in the end are 14º converging and 6º diverging.

The boundary conditions for CFD simulation were taken theoretically by formula given by NASA (mass flow for a compressible fluid).

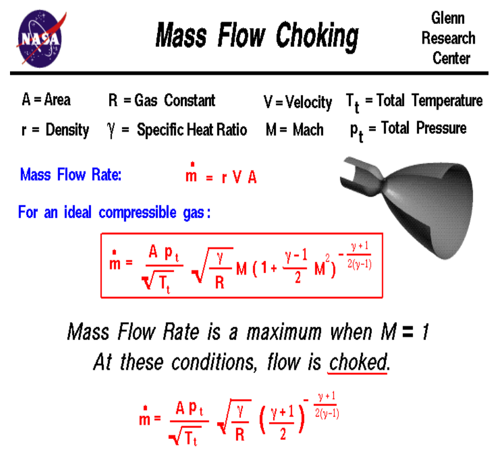

Fig. 6. Mass flow choking

By placing the values in this formula we got the boundary conditions

Inlet = (mass flux) =0.0703 kg\sec

Outlet= (pressure) =0.8 atm

A. Plenum

As rpm goes up you need a larger plenum, but a larger plenum will reduce throttle response and low-end power. A plenum also minimizes peak air velocity through the carburetor (or throttle body).The induction pulses in an intake cause velocity to rise and fall with each pulse. In conical shape, central high velocity flow has wider flow area, as less vortices are created. Also, velocity values are higher compared with other plenum shape, so conical shape was chosen. The other fact which we have to consider while designing the plenum is that it should have twice the volume of engine i.e. if the engine is of 390 cc then the minimum plenum volume should be 390*2=780cc.

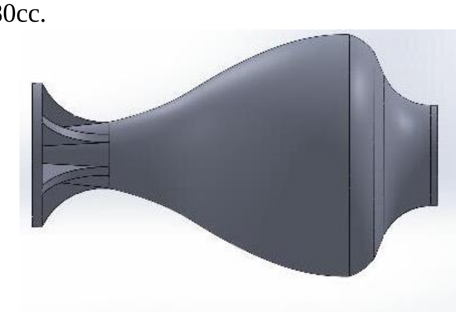

Fig. 7. Plenum

B. Runner

Intake runner length is the key factor to decide. The

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

performance of whole intake system is depends on tuning of runner length. Intake runner length is designed for 5000 rpm so as to get low end torque and power based on track & driver's experience. According to Induction wave theory
The formula for optimum intake runner length (L) is:

L = (EVCD*0.25*V*2/RPM*RF)–(0.5*RUNNER DIAMETER)
= (464º*0.25*1300*2/5000*4) – (0.5*1.8897)
L =359.032mm
RV = Reflective Value
V = Pressure Wave Speed =1300 ft/s
D = Runner Diameter = 1.8897 inch
EVCD = 720-(ECD)
EVCD = 720-226-30=464º
30º is subtracted such that pressure waves to arrive before the valve closes and after it opens EVCD = 464º. According to induction wave tuning theory, intake system was tuned at 5000 RPM, resulting in total runner length of 359.032mm. Runner Diameter is selected as 48 mm same as throttle body as both are connected through fasteners. The volume of runner should be 1.5 times the volume of engine i.e. 390*1.5=585cc.As rpm goes up you need a larger plenum, but a larger p.

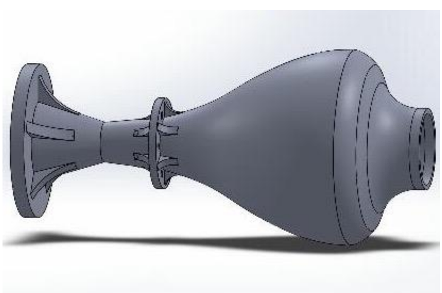

Fig. 8. Restrictor and plenum assembly

VI. Studies and Findings

For analysis and simulation we have used acusolve and field view by Altair hyper works.

1) *Meshing details*
Relative mesh size = 10mm
Curvature angle= 25o
Curvature mesh size factor=0.5
Mesh growth rate =1.0

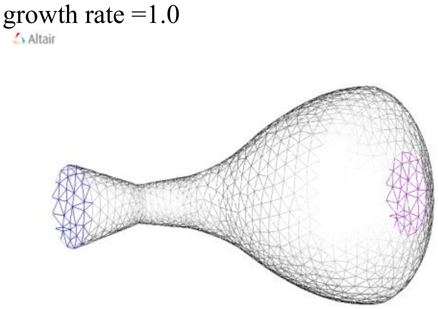

Fig. 9. Meshed assembly

2) *Problem description*
Analysis type = Transient
Flow equation = Navier stokes
Turbulence equation = k omega
3) *Solver details*
Time steps= 1000
Convergence tolerance=0.001

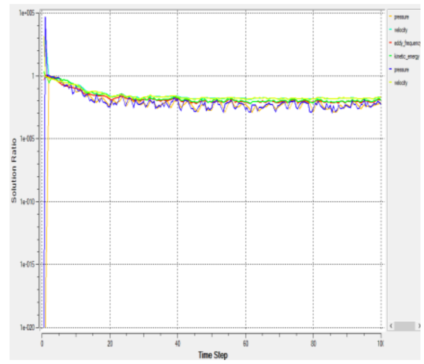

Fig. 10. Iteration curve

4) *Results*

B. Pressure Curve

By studying the pressure curve, which is plotted between the pressure and length of intake system, we find out that the pressure is highest at the inlet and suddenly drops at the throat of the venture as the velocity increases very rapidly. This curve can thus be justified by the choked flow. Also the pressure is constant throughout the plenum which is again the desired result.

C. Velocity Magnitude Curve

The velocity magnitude curve which we obtain is plotted between velocity magnitude and length of intake system, in this we see that the velocity is very much higher at the throat area of the venturi due to choked flow.

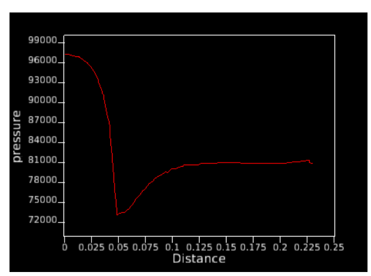

Fig. 11. Pressure curve

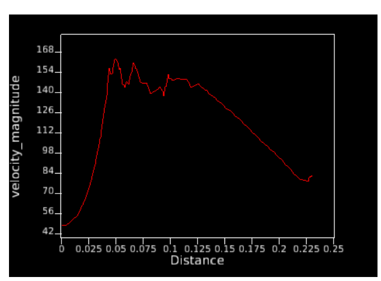

Fig. 12. Velocity magnitude curve

International Journal of Research in Engineering, Science and Management
Volume-1, Issue-10, October-2018
www.ijresm.com | ISSN (Online): 2581-5792

D. Pressure Contour

This contour clearly depicts the pressure variations throughout the intake system, also there is a fall of pressure at the throat which proves the existence of venturi effect and choked flow.

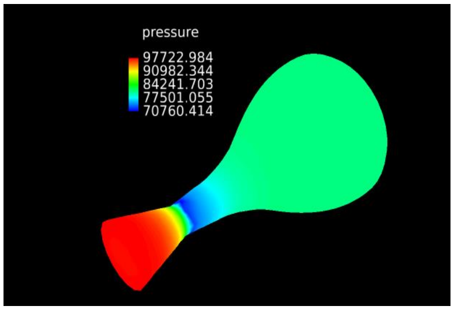

Fig. 13. Pressure contour

E. Velocity Stream line

The velocity streamline figure below shows the various streamlines of air or we can say the actual flow of air through the intake system, which shows that the air is dense at the centre of intake system.

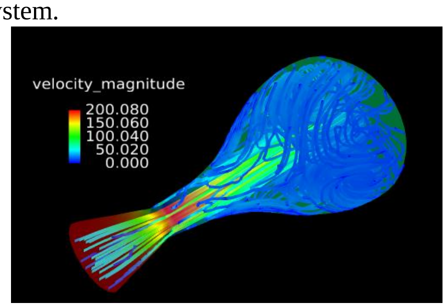

Fig. 14. Velocity stream line

F. Mass Flux Point

After performing the CFD analysis we find that the mass flux is constant throughout the intake system i.e. 0.0703kg/sec. which is the desired result.

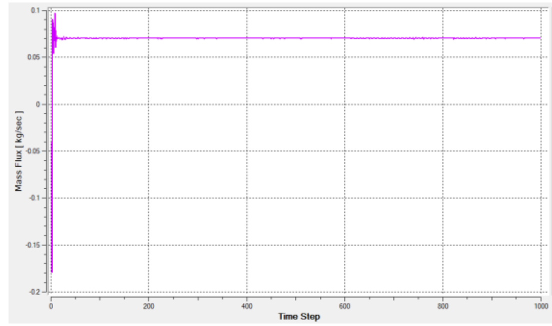

Fig. 15. Mass flux point

VII. CONCLUSION

A venturi, generally is used in diverse purpose for various applications in fluid dynamics involving either liquid or gas. In this project it is brilliantly used in reducing power of engine. As in this competition all the teams are busy trying to squeeze almost all single horse power available even with the restrictor attached, this gives rise to increasing research in optimization and finding out alternative technology for increasing mass flow rate to engine. One such technology used is supercharging of air downstream the venturi to increase the pressure on engine side. A venturi in itself can allow a maximum of 0.0703 kg/s of air flow to engine, considering no losses in friction and turbulence. From all the research done till now it is clear that at converging angle of 14 degree and diverging angle of 6 degree we get maximum recovery of pressure. Computational fluid dynamics played important role in all analysis.

REFERENCES

[1] ASME MFC-7M, Measurement of Gas Flow by Means of Critical Flow Venturi Nozzles. New York: American Society of Mechanical Engineers; 1987
[2] Vizard, David, "Rule for Intake Manifold Runner Length".
[3] http://www.grc.nasa.gov/WWW/k-12/airplane/mflchk.html
[4] H.K.Versteeg and W.Malalasekera, An Introduction to computational fluid dynamics. England: Longman Scientific & Technical, 1995, ch. 9.
[5] Section 8, "Sonic Flow Nozzles and Venturi — Critical Flow, Choked Flow Condition", ASME Publication.