ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 5 – No.2, March 2018

DESIGN OPTIMIZATION FOR AIR INTAKE SYSTEM OF FORMULA STUDENT RACE CAR

Mohnish Ahuja
Student (B.E Mechanical)
Chandigarh University
mohnish.ahuja25@gmail.com

Inderjeet Singh Saini
Student (B.E Mechanical)
Chandigarh University
inderjeetsinghsaini@outlook.com

Amberpreet Singh
Student (B.E mechanical)
Chandigarh University
amberbhangu@yahoo.com

ABSTRACT
Air intake system plays main role in obtaining excellent quality of air to the engine. Restrictor is a major part of air intake system in FORMULA STUDENT competitions and is used to control the efficiency and maximum power. This research paper focuses on optimization of the restrictor that is used to pass the air flow through 20mm diameter to limit the efficiency and maximum power. CFD was used to get the excellent mass flow rate by performing CFD at variable convergent – divergent angles. By examination of all the result in CFD, the convergent-divergent angles of 12 degree and 6 degree respectively were employed where minimum pressure drop can be achieved while getting the maximum efficiency at the same time.

Keywords
Restrictor, nozzle, converging - diverging angles, Mach number, CFD, Air intake system, optimization.

1. INTRODUCTION
Formula Bharat is a car racing national competition organized by Mobility Engineering Consortium, where in participants are supposed to design, manufacture and run a prototype of open formula type racing car. This competition is conducted once in a year where teams of more than 60 universities across the countries put there developed prototype up against each other in this competition. The performance from a different viewpoints is tested and evaluated by the FORMULA BHARAT committee. Formula Bharat rules committee has imposed a rule of adding a 20 mm restrictor to intake manifold and also states that all air flowing to the engine should pass through this single restrictor be it a single cylinder or multi cylinder engine. This limits the maximum amount of power produce by the engine as it deliberately reduces the input mass flow rate of the air flowing to engine. The Engine output must therefore, amount to 610 cc engines having an output of 120 horsepower with 15000 revolutions. For this the IC engine would require proper air-fuel ratio to work as per the desired specifications.

The formula based car designed by Chandigarh university team "iSHELL" used a single-cylinder; water cooled 373.2 cc engine having an output of 44 horsepower at 10000 RPM and a torque of 36 N-m at 8000 RPM. The flow of air takes place from the throttle body and the plenum of intake system before entering the runner which feeds to the engine. The Internal Combustion Engine takes in air from the atmosphere and the appropriate ratio of air-fuel mixture is combusted into the engine cylinder. At such high speed, engine requires large amount of air for combustion. Thus, the mass flow rate should increase and in order to compensate for this, the air has to pass with very high velocity (Mach number =1) to fulfill the engine with required quantity of air. According to studies, mass flow rate is a fixed parameter for 20 mm restrictor used for the calculations in further optimization. Thus, the objective is to minimize pressure drop by allowing maximum possible mass flow through the venturi type restrictor.

However, design, manufacturing and testing a number of intake systems can be costly and for improving, the design approach is needed. Therefore with the help of solid works CAD software which offers computational fluid dynamics (CFD) flow modeling tool, which allows a designer to simulate a range of flow conditions and intake shapes without manufacturing multiple prototypes for physical testing. So, CFD is considerable as it saves a lot of time and money.

2. RESEARCH PROCEDURE
By employing CFD techniques on the different convergent-divergent angles the optimized angle that yielded the best were used. The working efficiency is high in the increasing order of restrictor, Intake Manifold, Runner. Flow of air was restricted by 20 mm diameter by using simple orifice and converging diverging nozzle.

Restrictor: The purpose of the restrictor is to reduce the flow of air to the Engine. There are two types of instruments which can be used as a restrictor i.e. the Orifice plate as well as the Venturi tube. The Orifice plate is a simple rectangular plate with a hole drilled in it. The Venturi tube is a tube having a diverging section with a throat section is circular shape connecting the both.

The choice of restrictor to be used is laid out in the below comparison.

Table 1: Different parameters involved in the experiment

Parameters	Orifice	Nozzle
Coefficient of discharge	0.60	0.975
Pressure loss	Medium	Low
Viscosity effect	High	High
Accuracy (% of full scale)	3	1
Cost	Low	Medium
Manufacturing	Easy	Difficult

Taking into consideration the above elements mention in table 1 it can be observed that Venturi tube is much better as compared to the orifice plate in overall terms. The pressure loss and coefficient of discharge for Venturi tube is much better as compared to orifice plate. According to the restriction on engine, the more efficient design would help for the competition. Manufacturing cost was not

ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 5 – No.2, March 2018

taken into account. Comparing the above, Venturi tube was selected as the restrictor. According to the rules the restriction of the venturi meter must be 20mm in diameter. This will be the throat of the Venturi tube. The diameter of the throttle body for KTM duke 390 engine is 46mm. As the throttle body fits in the inlet face of the Venturi the diameter of the throttle body would serve as the inlet and outlet diameter of the Venturi tube.

As from the above discussion we have three elements which are fixed i.e. Inlet diameter, outlet diameters, and the throat diameter. This experiment will further require the converging and diverging angles of the Venturi. For this 3 known and 2 unknown parameters were defined . The pressure at inlet is atmospheric whereas the temperature at inlet is ambient. The boundary condition at the outlet can either have pressure, velocity and mass flow rate. Calculating the pressure and velocity at the outlet involves a complex procedure and can end velocity and mass flow rate. Mass flow rate at outlet can be easily calculated using the Choked flow equation. According to the equation given in the fig:1 the mass flow rate at outlet was calculated :

M = 1 (choked flow)

A = 0.001256m² (20 mm restriction)

R = 0.286 KJ/Kg-K

$\gamma = 1.4$

Pt = 101325 Pa

T = 300 K

Mass flow rate = 0.0703 kg/s

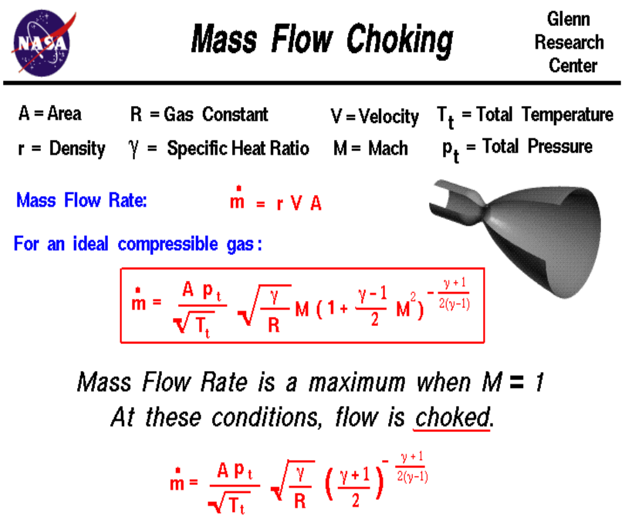


Fig 1: Mass Flow Chocking

The most important parameter in compressible flow is Mach number i.e.; the ratio of the speed of a body to the speed of sound in the surrounding medium.

Mathematically: M=U/C

Where U - flow velocity

C - Speed of sound.

When Mach number is less than 0.3, effects includes compressibility can be neglected because density change occurs here around 3%. When Mach number is from 0.3 to 1 flow it is categorized as subsonic flow and if Ma >1 flow then it comes under supersonic flow, in this specific areas compressibility of fluid increases and its effects are considerable.

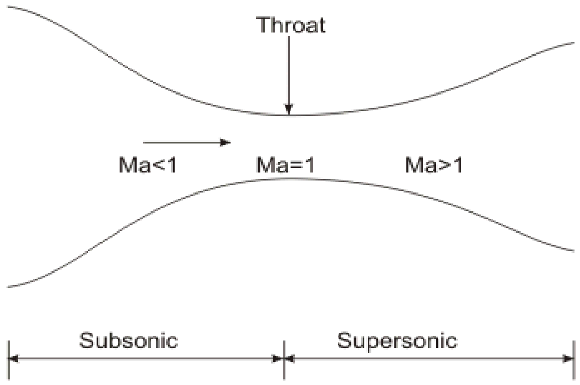
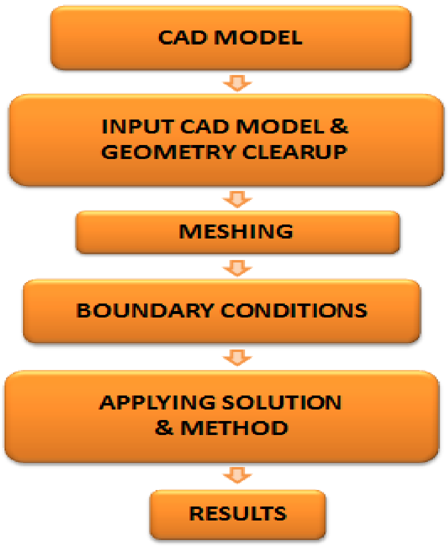


Fig 2: Mach number Variation

Applying above concepts to a diverging converging nozzle it can be understood that density of air is reduced quickly when it is passing through the restriction. This concludes that pressure on downstream side is reduced. As all the air to engine will be passed through this single venturi, pressure available engine inlet will be very less. Thus engine work is increased as it is forced to pull more air from venturi so that there is enough air available inside combustion chamber to burn fuel completely. This scenario usually occurs at higher rpm ~ 6000 and more. Which is not in favorable conditions for engine as it will unable to squeeze out maximum power that is available and the car moves slowly. Therefore the focus to reduce this pull from engine and recover maximum amount of pressure at outlet of venturi.



ISSN (O): 2393-8609
International Journal of Aerospace and Mechanical Engineering
Volume 5 – No.2, March 2018

Fig 3: Flow Chart

3. DESIGN METHODOLOGY:

The whole design formation for the venturi was done in CAD part in CATIA v5 and converted into IGES format .The converted format is imported in Ansys15.0 workbench. After importing in fluent the geometry cleanup is done in design modeler. Structural meshing is done in quads elements having nodes of 2560 and 1806 elements. The meshed geometry is further imported in fluent to perform CFD analysis .The fluid properties are entered in the fluent to define the boundary condition of inlet and outlet wall conditions. Then the variables of the solution method and controls were entered into this setup and were undergone for more than 100 iterations after which the results were plotted in CFD post.

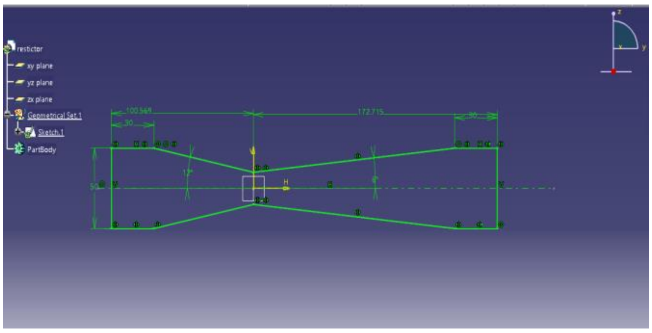


Fig 4: 2-D figure showing Dimensions of prototype

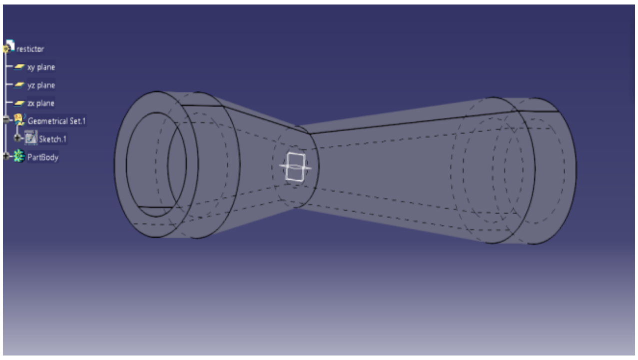


Fig 5: 3-D design of prototype

3.1 CFD Analysis:

Ansys15.0 CAD software is preferred for checking results of CFD simulation. With the help of this software the maximum mass flow rate at minimum pressure from the engine was determined. During CFD analysis no. of iterations were taken at different diverging and converging angles to achieve minimum pressure drop across cross-sectional area.

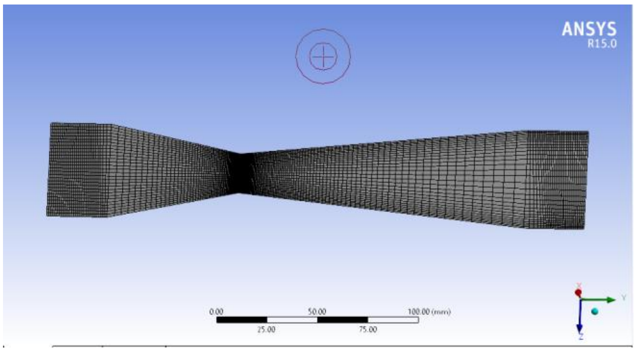


Fig 6: Meshing

4. STUDIES AND FINDINGS

After the collection of data taken from CFD analysis finding all the required data we need for solving the problem next step is to find the required dimensions of venturi at which will get minimum pressure drop. So, to get actual required dimensions some of the dimensions are assumed by applying the basic formulations and knowledge of venturi. CAD modeling was done first using CATIA v5 and was further analyzed in flow simulation for the boundary conditions given below.

Table 2: Iterations carried out on converging and diverging angles

Iteration no.	Converging angle	Diverging angle
1	10	6
2	11	6
3	12	6
4	13	6
5	14	6

The above iterations were carried out in the flow simulation by constructing a volume mesh of fine quality. And the results were obtained from these 5 iterations are as below.

Table 3: Results from 5 iterations

Iteration no.	Converging angle (degree)	Diverging Angle (degree)	Pressure difference (Pa)
1	10	6	2.071962
2	11	6	2.681999
3	12	6	1.977804
4	13	6	3.165509
5	14	6	2.510130

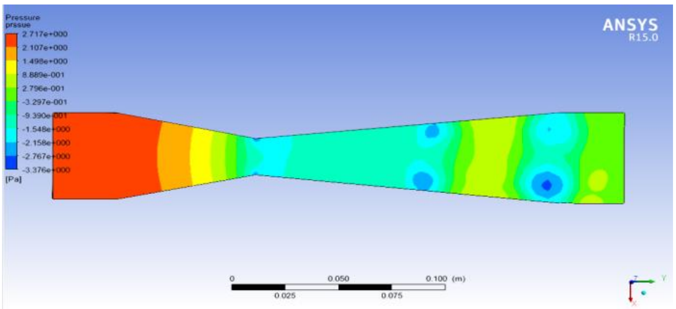


Fig 7: Pressure plot for iteration 1

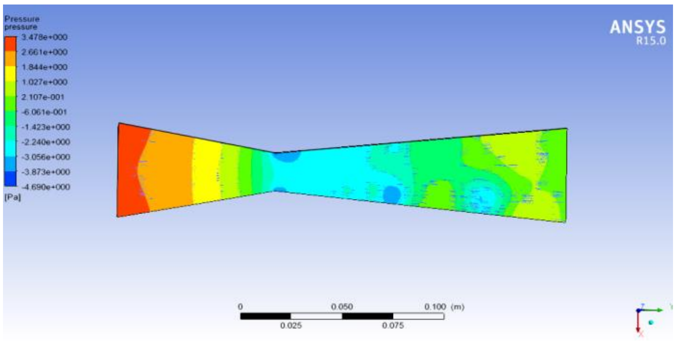


Fig 8: Pressure plot for iteration 2

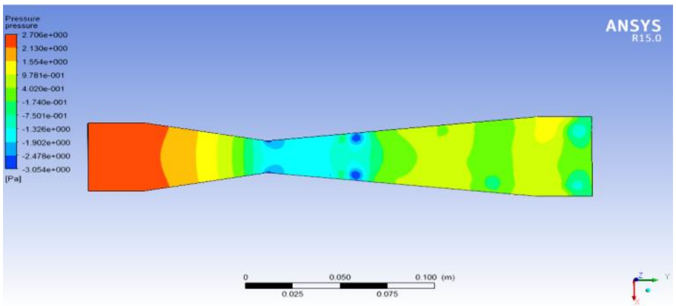


Fig 9: Pressure plot for iteration 3

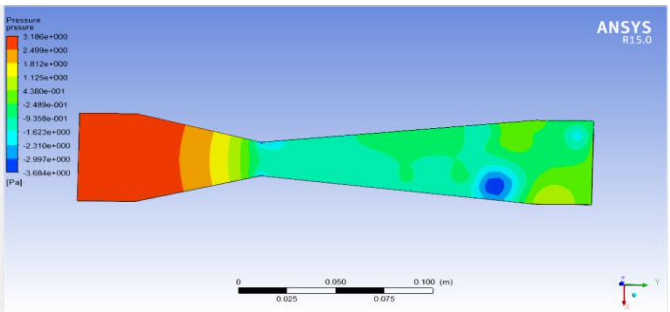


Fig 10: Pressure plot for iteration 4

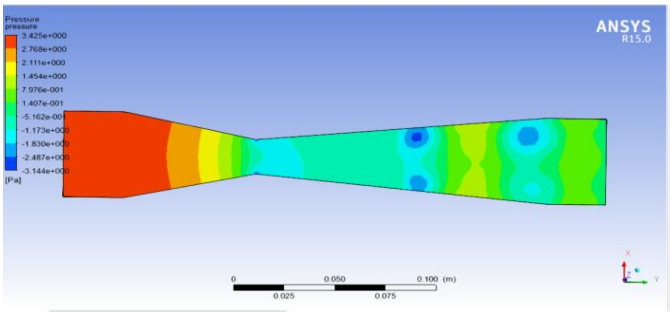


Fig 11: Pressure plot for iteration 5

After the different iterations performed different at different angles. It was found that the diverging angle was 6 degree and converging angle is 12 degree and pressure drop is maximum at these angles.

Following are the images which show other important parameters for final design of venturi.

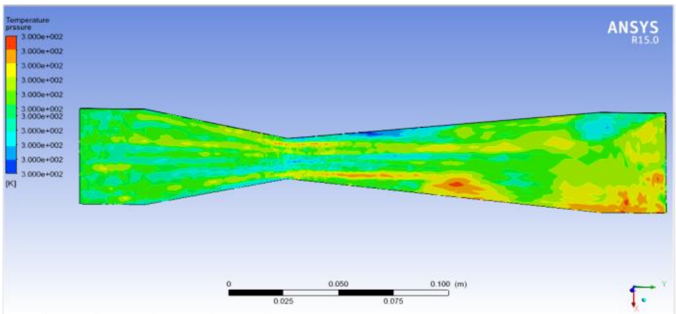


Fig 12: Temperature distribution for final design

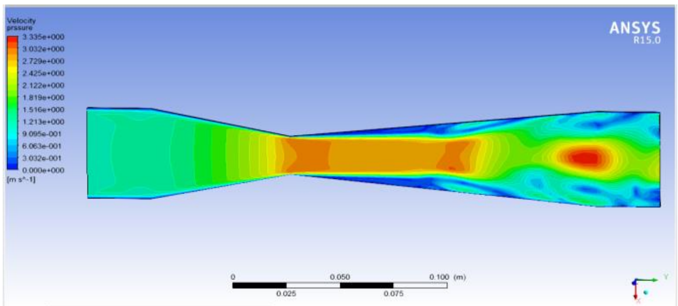


Fig 13: Velocity plot for final design

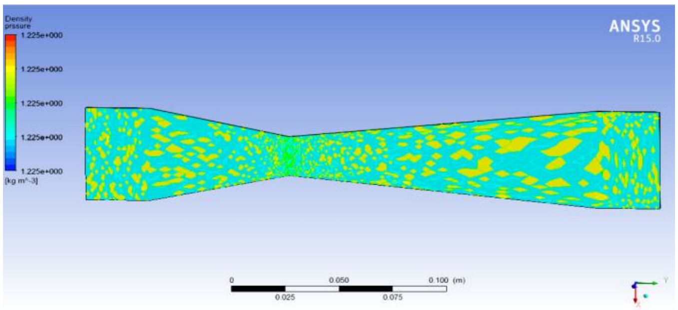


Fig 14: Density plot for final design

ISSN (O): 2393-8609

International Journal of Aerospace and Mechanical Engineering

Volume 5 – No.2, March 2018

5. CONCLUSION

Restrictor is the practical application of fluid dynamics and in this project it is used to reduce the power of engine by passing the air through 20mm converging-diverging nozzle. A venturi itself can allow a maximum of 0.0703 kg/s of air flow to engine. The experimentation results observed in this paper evaluates that converging angle of 12 degree and divergent angle 6 of degree would help us to retain a maximum pressure and best performance of engine. Moreover, computational fluid dynamics plays a vital role for getting all the analysis.

6. ACKNOWLEDGMENT

I would like to express my sincere thanks to my team who helped me to bring up this paper from nowhere to its publication. My sincere thanks to Prof. Vidyadhar of Mechanical department who helped me out in performing computational fluid dynamics accurately, after lots of practice and hours put into it.

7. REFERENCES:

[1] Y. A. C¸engel and M. A. Boles, Thermodynamics: an Engineering Approach, chapter 17, McGraw-Hill, New York, NY, 2011.

[2] R. Stone, Introduction to Internal Combustion Engines ,chapter2 Society of Automotive Engineers, Inc., Warrendale ,PA, 1994.

[3] B. Jawad, A. Lounsbery, and J. Hoste, “Evolution of intake design for a small engine formula vehicle,” in SAE Technical Paper 2001-01-1211, SAE World Congress, Detroit, MI, 2001.

[4] ASME MFC-7M, Measurement of Gas Flow by Means of Critical Flow Venturi Nozzles. New York: American Society of Mechanical Engineers; 1987.

[6] Deepak Ranjan Bhola, “CFS Analysis Of Flow Through Venturi Of A Carburetor,” National Institute of Technology Rourkela, Rourkela – 769008, India, May 2011. Available at: http://ethesis.nitrkl.ac.in/2296/1/final_report.pdf

[7] FSAE Variable Air Intake Consulted on – 22 JAN 2013 http://poisson.me.dal.ca/~dp_06_9/concept.htm

[8] Section 8, “Sonic Flow Nozzles and Venturi — Critical Flow, Choked Flow Condition”, ASME Publication.

[9] https://www.grc.nasa.gov/www/k-12/airplane/mflchk.html