Air flow optimization of venturi type intake restrictor

[1]Pruthviraj Vitthal Wable, [2]Sahil Sanjog Shah
[1,2]UG student
[1,2]Department of Mechanical Engineering,
[1,2]Trinity College of Engineering & Research, Pune, India

***Abstract*—This research paper deals with the optimization of venturi type air intake restrictor that can be used in FSAE (Formula Student) competitions. As per the rulebook provided by FSAE the air going into the engine from the throttle body or carburetor must pass through a 20mm diameter opening. If this rule is violated by any of the team the car is disqualified from the event. The reason behind this rule is to reduce the power output of the engine which will ensure the safety of the driver. In order to compensate the loss of power there must be an arrangement that will compensate this loss. Here Venturi is used as a restriction device. Creo 2.0 is used for CAD modelling and ANSYS 18.1 (Fluent Solver) is used for CFD (Computational Fluid Dynamics) analysis of the venturi.**

***Index Terms*—FSAE, Restrictor, Venturi, Creo 2.0, CFD, ANSYS 18.1**

I. INTRODUCTION

FSAE is a platform which provides an opportunity to the engineering students to enhance their practical knowledge as well as technical knowledge. The team have to design and manufacture a formula type race car in the college premises without the professional help. As this car is made according to the rulebook provided by the organizing committee. The rulebook clearly mention that a team can use the engine up to 600 cc capacity and that too with a restriction device having diameter 20 mm[1]. As per this rule the air going in to the engine must pass through the 20 mm diameter opening [1]. For this purpose a special type of restriction device can be used. The reason behind using the restriction device for the air going into the engine is to reduce the power output of the engine. This is done for the safety of the driver. But by using this device the performance of the car reduce. The biggest challenge is now is that how to compensate for this power loss. As the air passes through the restricted 20 mm diameter hole which will reduce the mass of air going inside the engine. This problem can be solved by using an optimized restriction device which will compensate the power loss as well as it will satisfy the rule.

Basically there two options available for the restriction device that can be used in the air intake system of the car.

1. Venturi
2. Orifice

The following table shows the comparison between the both-

Table no. 1 shows the comparisons between venturi and orifice [2]

Parameters	Venturi	Orifice
Pressure loss	Low	Medium
Coefficient of discharge	0.975	0.60
Manufacturing	Hard	Easy
Cost	High	Low

From the above table it can be concluded that Venturi is more efficient as the restriction device that is used in the intake system of the car as the pressure losses are less as well of coefficient of discharge is high in the venturi [2]. This both characters of the venturi are important to compensate the loss power. As we need less pressure loss and high mass flow in the engine. But the manufacturing is quite hard as it's difficult to manufacture the throat section of the venturi.
Hence Venturi is selected as the restriction device for the intake system.

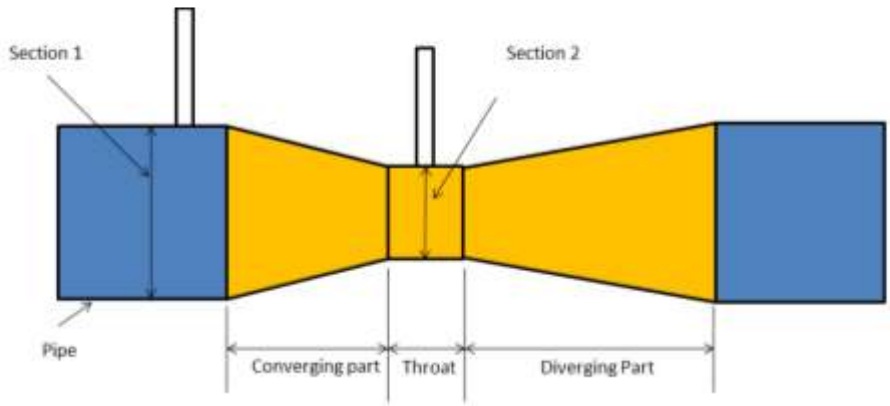

Fig.1 Shows Venturimeter and its components [4]

II. Design

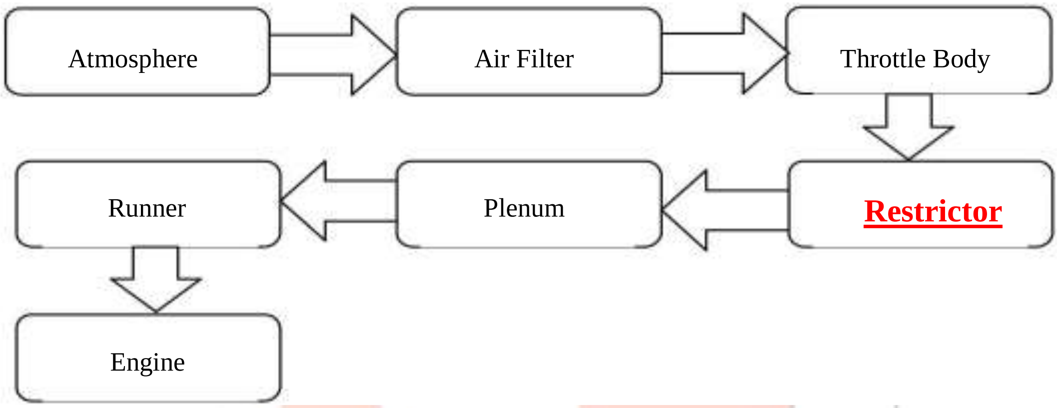

Fig.2 Shows sequence of flow of air in intake system

In India mostly the teams use KTM Duke 390 engine which is a single cylinder engine having displacement of 373.2 cc which gives the maximum power output of 43.5 PS @9000 RPM[4]. The normal inlet opening diameter of the engine near about 40 mm, as restrictor is having the diameter of 20 mm it will reduce the power output of the engine. Without the proper supply of air-fuel ratio the engine will not give a desired performance. The design goal for the restrictor is maximum flow of fluid (air) in the engine and less pressure drop in the intake system.

Now considering a working condition of the engine at 2000 or 3000 RPM with an air restrictor. The loss of the power due to the lack of air in the engine can be compensated by the increase in the velocity of the air at the throat section of the venturi. In the throat section of the venturi the pressure reduce and velocity of the air increases.

Now considering the same engine at higher RPM ie.7000-8000 RPM. At such high RPM engine requires more amount of air for the complete combustion of the fuel, in order to increase the flow rate of air the air in the restrictor flows with high velocity. The velocity of the air goes on increasing as the RPM increases and reaches to a condition where the MACH no. becomes 1 in the restrictor it is also known as the chocked flow. So now the main objective for the design of venturi is lesser pressure drop in the venturi and high flow rate of the air in the engine.

Inlet diameter of the venturi= 40mm

Outlet diameter of the venturi= 46mm

Designing of venturi is the iterative process, where the exact size of i.e. Converging angle and diverging angle of the venturi can be decided after several iterations. Model for the CFD analysis can be made in planer form (2D). It gives the same results as that of 3D geometry as well as the solving time also reduces. The diverging angle plays an important role in the pressure recovery with in the nozzle (venturi). Here the divergent angle is kept constant (6 degrees) and the change in the convergent angle is done. Iterating with the variable convergent angle and constant divergent angle can give desirable design of the venturi. Table no. 2 shows the iterations to be done to get an appropriate result. Figure 3 shows the CAD model of the venturi.

Table no. 2 shows the iterations to be done

ITERATION NO.	CONVERGANT ANGLE	DIVERGANT ANGLE
1	14 DEGREE	6 DEGREE
2	16 DEGREE	6 DEGREE
3	18 DEGREE	6 DEGREE

Fig.3 CAD model of venturi

III. CALCULATIONS

The best method to validate the design is by using the CFD (Computational Fluid Dynamics), this will help to analyze the flow in the venturi. For the analysis there is need of some known parameters which will be given as the input in the CFD analysis. Now the assumption that can be made for the inlet air is that it is 1 atmospheric pressure (101325 Pa) and the temperature of the air is ambient temperature. The outlet parameters are unknown till now. Without the outlet parameter the analysis cannot be done. So now the assumption is that the mass of air entering in the venturi inlet is equal to the mass of air coming out of the venturi outlet. As discussed earlier the condition where the MACH no. becomes 1 (chocked air condition). By using the chocked flow equation given by NASA the flow rate through the venturi can be found out.

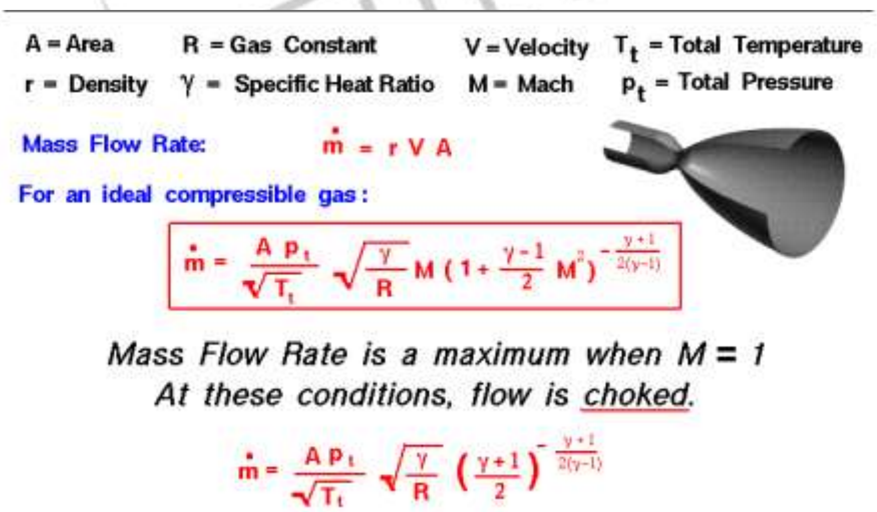

Fig.4 Flow equation for mass flow rate[3]

The values to be substituted in the final chock flow equation are given below:-

Pt= 101325 Pa (Initial pressure)
T = 300K (Ambient temperature)
γ = 1.4 (Adiabatic index)
R (air) =0.286 kJ/Kg-K (Universal gas constant)
A = 0.001256 m^2 (for 20 mm diameter of venturi
M = 1 (Choking Conditions)

Mass Flow Rate at Choking = 0.0703 kg/s

As discussed earlier the mass flow rate will be constant in the venturi. Now the value of mass flow outlet can be give as 0.0703 kg/s.

Known parameters- Inlet pressure and mass flow rate of air

Unknown parameters- Velocity and Pressure at outlet.

In order to find the unknown parameters CFD analysis can be carried out for 3 different venturi which will differ in the converging angle.

IV. CFD ANALYSIS

As discussed in the above section the CFD analysis is carried out to validate whether the venturi is giving the desirable output or not. The planer 2D geometry is made for this analysis. The planer geometry will reduce the complexity of the problem as well as it will also reduce the solving time taken by the solver. CAE software i.e. ANSYS 18.1 in which FLUENT SOLVER is used for the analysis. Figure 5 shows the planer geometry made for CFD analysis

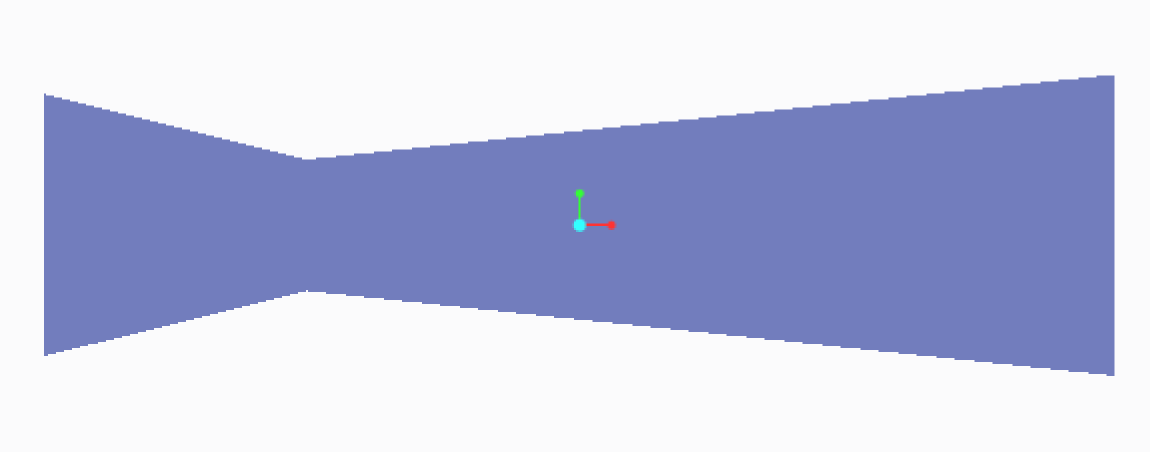

Fig.5.Planer geometry of venturi

In the pre-processing the meshing is done. The mesh is selected as the face mesh, inflation tool for the meshing with 5 layers at a growth rate of 1.2 is used. The total grid size (mesh size) is kept as 3mm. Figure 6 shows the meshed planer geometry.

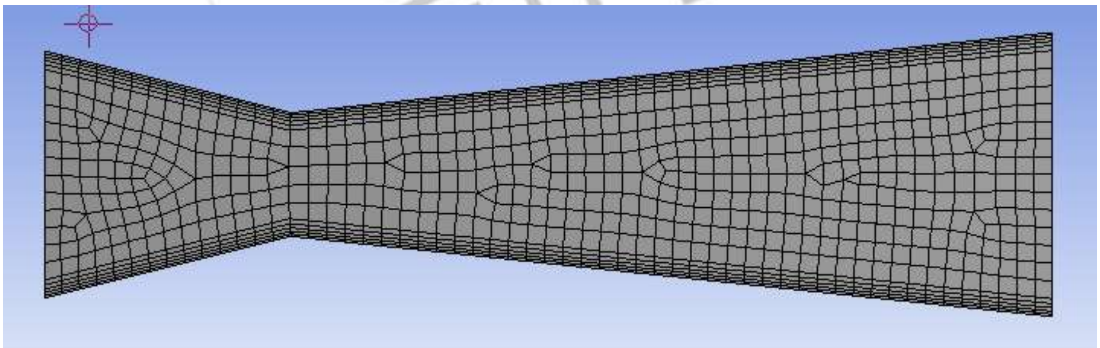

Fig.6 Mesh model of venturi

Boundary conditions for the analysis
INLET – Pressure 1 atm (101325 Pa) [3]
OUTLET- Mass flow rate 0.0703 kg/s
WALLS- Remaining boundaries (stationary)
No. of iterations- 500
Convergence limit set- 0.001x10^-6

V. RESULTS

ITERATION 1

The figure 7 and figure 8 shows the pressure contour and velocity contour which are obtained after the CFD analysis. The solution got converged after 289 iterations.

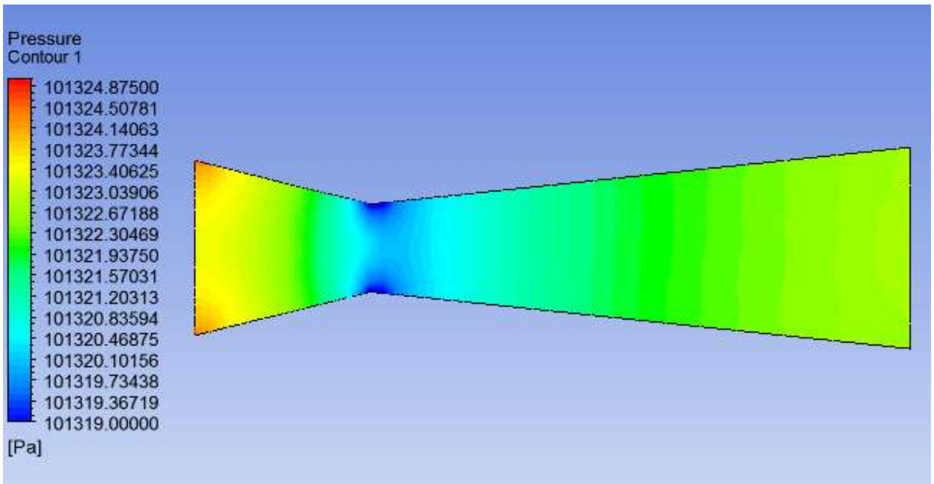

Fig.7 Pressure Contour for Iteration 1

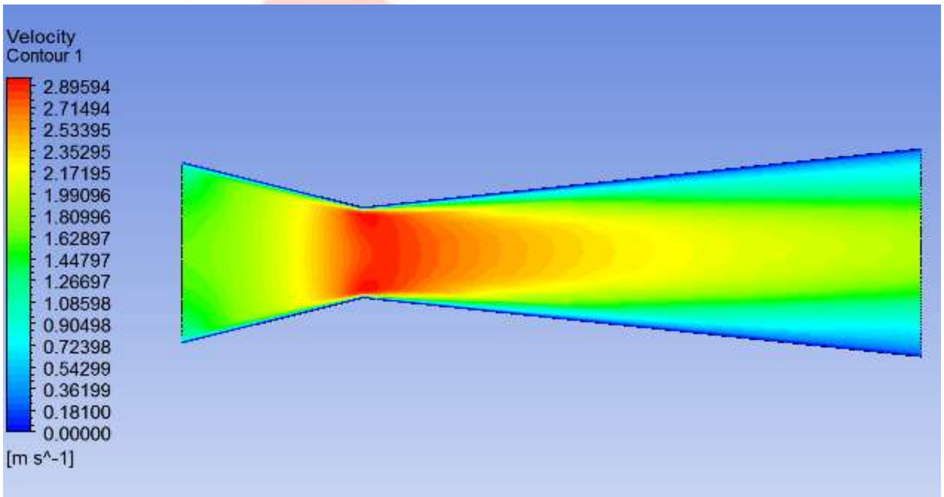

Fig.8 Velocity contour for Iteration 1

ITERATION 2

The figure 9 and figure 10 shows the pressure contour and velocity contour which are obtained after the CFD analysis. The solution got converged after 313 iterations.

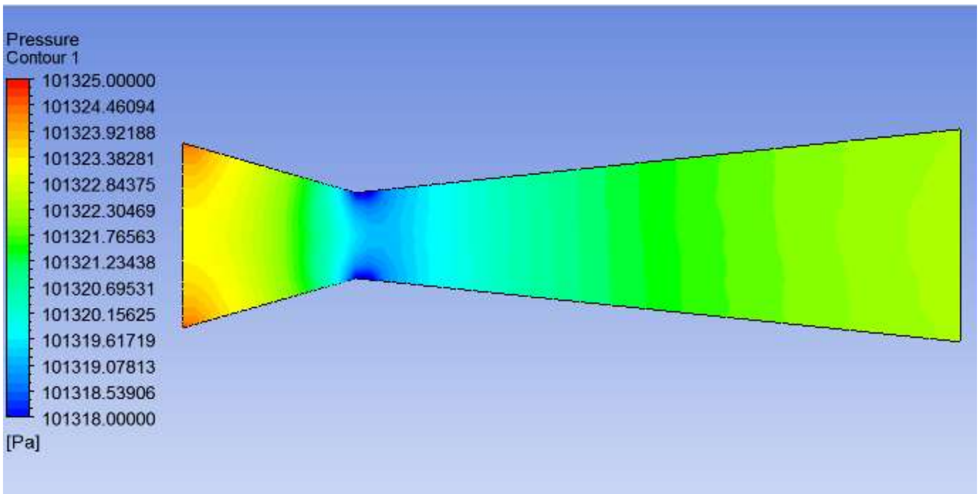

Fig.9 Pressure Contour for Iteration 2

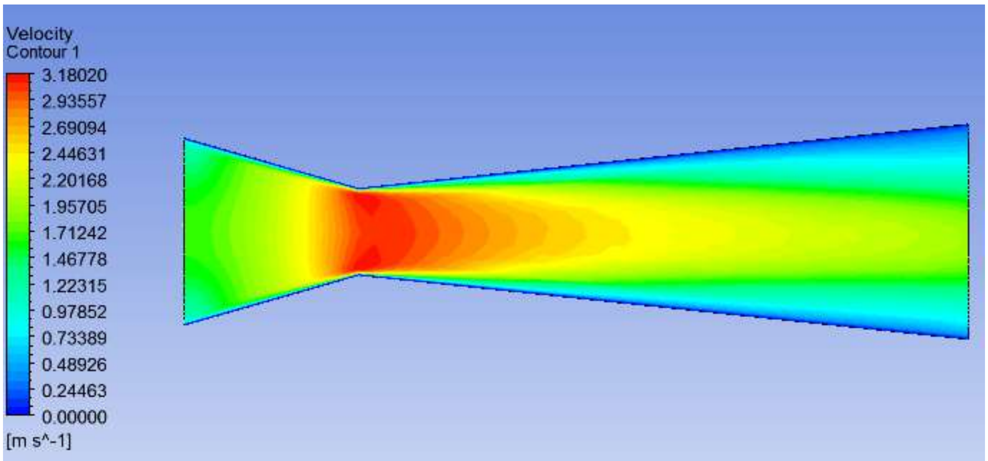

Fig.10 Velocity contour for Iteration 2

ITERATION 3

The figure 11 and figure 12 shows the pressure contour and velocity contour which are obtained after the CFD analysis. The solution got converged after 375 iterations.

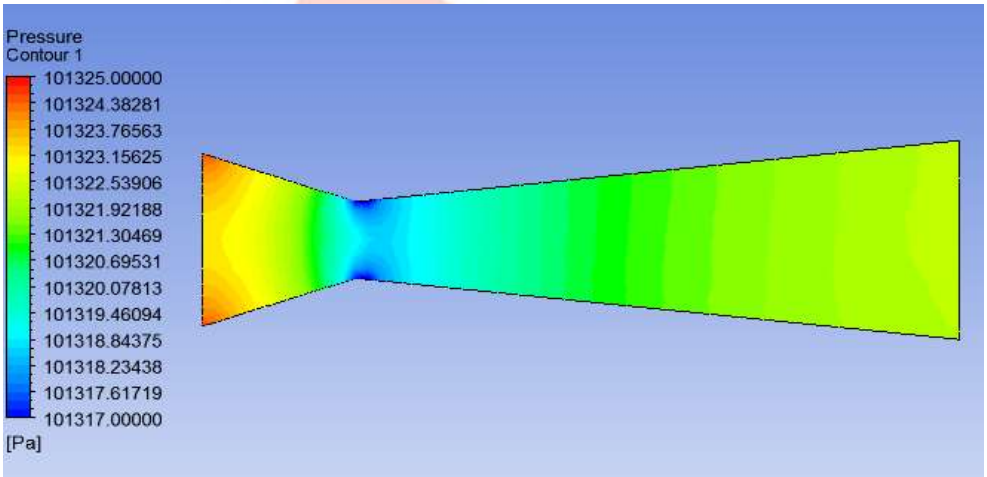

Fig.11Pressure Contour for Iteration 3

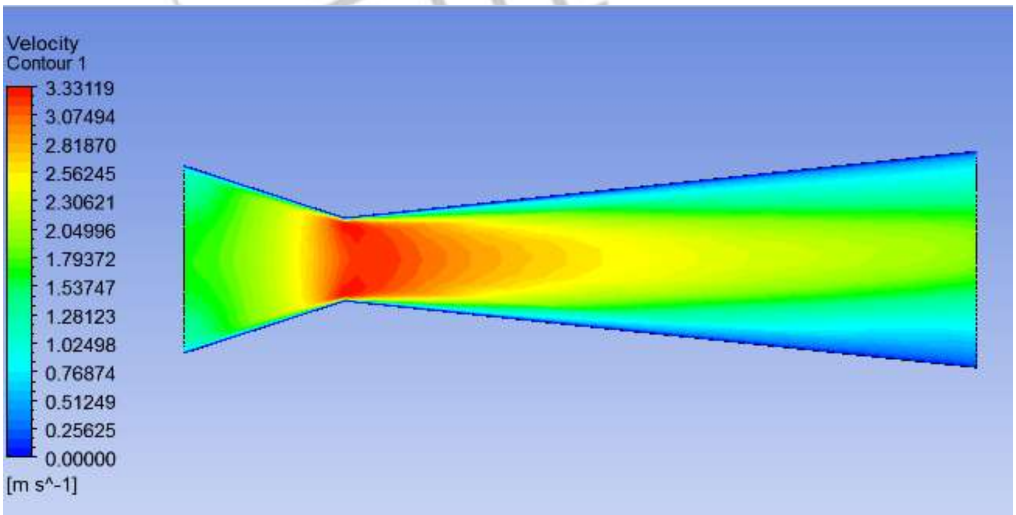

Fig.12 Velocity contour for Iteration 3

RESULT TABLE

ITERATION NO.	MAX PRESSURE (Pa)	MIN PRESSURE (Pa)	PRESSURE DIFFERENCE (Pa)	VELOCITY A6T THROAT (m/s)
1	101324.8750	101319.0000	5.875	2.895
2	101325.0000	101318.0000	7	3.180
3	101325.0000	101317.0000	8	3.33

From the above table it can be seen that the maximum pressure at the start of the venturi is same but there is a difference between the minimum pressure which occurs at the throat section of the venturi. As decided earlier the objective for the venturi design is lesser pressure drop in the venturi. Here in the iteration 1 the maximum pressure drop is 5.875 Pa where as in iteration 2 and iteration 3 the pressure drop is 7 Pa and 8 Pa respectively. So now we can conclude that as the divergent section in all iterations is same (6 degrees) so here the conclusion can be made that the venturi with 14 degree convergent angle and 6 degree divergent angle can be used as the air intake restrictor in the intake system of the FSAE car, as the pressure drop at the throat section is less so it will take less time to recover the pressure at the end of the venturi.

VI. CONCLUSION

From the above results it can be concluded that the venturi with 14 degree convergent angle and 6 degree divergent angle is efficient for the venturi as

The pressure drop is less in this venturi hence maximum pressure recovery can be done.

The mass of air that will go into the engine will compensate the power loss occurred due to restriction.

The length of the venturi is moderate that can satisfy the envelope rule given in the rule book.

The CFD analysis done in FLUENT solver gave quite good results after doing 3 iterations in all.

Hence the venturi can be used in the intake system of the car with other components of intake system like air filter, throttle body, plenum (air box) and runner.

Further the whole intake system with all components included can be analyzed by using the CFD tool

VII. ACKNOWLEDGMENT

The authors would like to thank their teachers who helped them to gain the knowledge about the subject as well as their parents who gave them an inspiration to write this paper, all the friends who directly or indirectly helped to write this paper.

REFERENCES

[1] SupraSAEIndiaRulebook2017,http://suprasaeindia.org/images/downloads/SUPRA_SAEINDIA_2017_RULE_BOOK.pdf page 14-33.

[2] F.M.White, *Fluid Mechanics*. New York: McGraw-Hill, 2003, ch. 9

[3] Anshul Singhal, Mallika Parveen, Proceedings of the World Congress on Engineering 2013 Vol III,
WCE 2013, July 3 - 5, 2013, London, U.K.

[4] https://www.wikipedia.org/

[5]http://www.ansys.com/student/support