International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 02 | Feb 2021 www.irjet.net p-ISSN: 2395-0072

Design Development and Manufacturing of Air Intake Flow System In FSAE Car

Shubham Shinde[1], Saurabh Walekar[2]

[1-2]*Student, Department of Mechanical Engineering, Trinity College of Engineering and Research, Pune-048*

Abstract - *Design and Manufacturing venturi type restrictor which is to be fitted in the intake manifold of a Formula SAE car engine. The main purpose of 20mm restrictor in intake manifold is to restrict mass flow passing to the engine thus reducing its maximum power. Objectives of this research is to optimize a venturi type design to allow maximum possible mass flow rate to the engine from 20 mm restrictor buy reducing the difference in pressure across venturi at all speeds. Analytical calculations are done based on standard results to get maximum mass flow rate and CFD tool is used to calculate minimum pressure drop across the restrictor buy varying converging and diverging angles of venturi. It can be observed from CFD results that for converging and diverging angle of 12 degrees and 6 degrees respectively minimum pressure drop can be achieved.*

Key Words: **Formula SAE, Intake Restrictor, Flow optimization, CFD.**

1.INTRODUCTION

SAE SUPRA is a competition organized by SAE INDIA (formerly Society of Automotive Engineers). The SAE SUPRA rules committee imposed a rule that power of any 4-stroke petrol engine used in the competition must not exceed 610cc per cycle and all engine airflow must pass through a 20mm restrictor to limit its power compatibility.

This rule reduces the air flow to the engine intake thereby arising a need for a device that can allow maximum air flow through the restrictor. Thus, the aim of this project is to design and create an air flow restrictor which is to be fitted in between the throttle body and engine that can allow maximum airflow to the engine. The SAE SUPRA car is being built by team Aviators of TCOER.

An IC engine takes in air from the surroundings and the air fuel mixture is combusted inside the engine cylinders. In an IC engine, the pressure is low during the intake stroke, causing the air from the atmosphere to enter the cylinders. The higher the rpm, greater the pull and lower the pressure. Due to the mass flow rate being a fixed parameter, the aim is to achieve maximum mass flow with minimum pull. This can be done by analyzing the restrictor device for minimum pressure difference between the atmosphere and cylinder, resulting in maximum air flow to the engine. To begin this project investigation into the Formula SAE competition rules and regulations was required. The competition has strict guidelines on the safety features of the cars and other limitations to restrict the power and cost in order to challenge the imagination and ingenuity of competitors. Following this a great deal of research was done into the many different systems available to be incorporated into the drive train of this style of race car. This research will not only lead to my design decisions for this year's car, but should also become a foundation for future students involved in developing a race car for Team's Motorsport in future competitions. The task of designing and manufacturing a competitive vehicle requires a team of students each responsible for specific areas of the car. As a result of this the project requires great teamwork and communication to be completed successfully, particularly in the time permitted.

The performance of an internal combustion engine depends directly on the amount of air elaborated in every cycle during its functioning. On this basis, a way to limit the maximum power of an engine is to introduce a pressure loss along the intake manifold. According to this principle, Formula SAE rules impose the adoption of a restrictor, with a diameter of 20mm, along the intake line between the throttle valve and the engine inlet. More in detail, the restrictor has the aim to cause a pressure drop that is proportional to the second power of the instantaneous velocity of the airflow. In order to reduce the influence of the restrictor, it is necessary to minimize the maximum instantaneous flow velocity through it [1].

A way to obtain this objective is to place a plenum chamber between the restrictor and the engine. The plenum allows one to minimize the pulsating flow through the restrictor. As consequence the pressure drop at the restrictor decreases and the mass flow rate elaborated by the engine rises. This allows a growth of the engine performance. The greater is the plenum volume the higher is the power that the engine can provide [1]. Air travels through a throttle body and intake plenum before reaching individual runners that feed each cylinder. An optimized intake system will net the engine more airflow and more power; however, designing, machining, and testing a number of intake systems can be costly. Design improvements are incremental and the design approach is essentially a process of trial and error. Computational fluid dynamics (CFD) flow modeling software offers an alternative to the experimental method of design. CFD is based on computer simulation, where the governing equations of fluid motion (e.g., Navier-Stokes equation) are solved numerically. CFD allows a designer to simulate a range of intake shapes and flow conditions without having to machine multiple prototypes for physical flow testing [2].

According to the stoichiometric air-fuel ratio, 14.7 grams of air is required to burn 1gram of fuel. Due to the restrictor rule the actual intake of the engine KTM Duke 390 which has intake diameter of 54 mm reduces to 20 mm which creates a high change in pressure. The Supra engines have to run at around 7000 rpm to 11000 rpm resulting in requirement of more air for combustion which can be achieved by increase in mass flow rate, but the 20mm restrictor area being less, air has to pass with very high velocity to fill the engine with required amount of air.

The main purpose of 20mm restrictor in intake manifold is to restrict mass flow passing to the engine thus reducing its maximum power. Objectives of this research is to optimize a venturi type design to allow maximum possible mass flow rate to the engine from 20 mm restrictor buy reducing the difference in pressure across venturi at all speeds. Analytical calculations are done based on standard results to get maximum mass flow rate and CFD tool is used to calculate minimum pressure drop across the restrictor buy varying converging and diverging angles of venturi [4]. Another problem with intake system is that in conventional engines it is optimized for a specific speed at which it gives maximum torque. Studies showed that longer intake manifolds give peak torque at low engine rpm & shorter engine manifolds gives peak torque at higher engine rpm. Conventional engines have intake manifolds which are compromised to get benefit of both [5].

1.1 PROBLEM STATEMENT

The output and torque of an engine have the greatest effect on the engine's character. These, in turn, are greatly affected by the degree to which the cylinder is filled and the geometric form of the intake tract. High torque requires an intake manifold with geometry different to one for high power output. A medium intake manifold length with a medium diameter represents a compromise but it results in the lower torque & power at very low & high speeds which ultimately results into degraded performance & less fuel economy. Variable intake manifold poses an ideal alternative to solve this problem.

1.2 OBJECTIVE

To Develop an Air Intake Flow System for FSAE Car to Increase Mass Flow Rate and Reduce Pressure Drop

1.3 SCOPE

The FSAE being a very widely spread international engineering event where students from various institutions participate, it forms a diverse environment for students which help exchange the practice of interchanging the ideas with other students from various cultures and background creating a fit working experience in engineering industry.

1.4 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 [6].

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 is very important in the case of solving boundary value problems, otherwise leads to numerical instability of the system. Considering the above factors in mind, the mass flow rate for choking condition is calculated [6].

Considering any models in CFD analysis, there can be 3 types of errors which have to be considered seriously while solving the problem: -

(a)Due to the improper meshing, there can be deviation from the actual model which leads to error in the system. This problem is solved by applying structured mesh and applying convergence criterion.

(b)Numerical instability due to improper boundary conditions or truncation errors while conversion of partial differential equations into algebraic equations. The small error will grow rapidly diverging from the actual error. This problem is solved by giving appropriate and suitable space step.

(c)The mesh has been imported into ansys and solved. The numerical error is monitored till the value of error gets reduced to 0.001% so that numerical stability is achieved [6].

Inlet manifold generally experiences two kind of pressure waves one being compression and other is suction wave. Intake manifold can be tuned in such a way that the volumetric efficiency of more than 100% can be achieved. Thus, improvement in the torque output and power of the engine is possible. Air flowing through the intake manifold runner, past the intake valve and into the cylinder flows alike until the intake valve closes. After the closure of valve air strikes on the closed valve & high-pressure wave is created. This is the compression wave & travels back and forth along the closed intake runner length. Tuning the intake manifold is nothing but making the compression wave come back at the inlet valve exactly when it opens. This compression wave then rushes into the cylinder to the feel it at values more than that of normal one & maximum volumetric efficiency is achieved. The effect is termed as Ram effect charging & length of intake manifold to achieve that Ram effect can be predicted by Chryslers Ram Theory [5].

Another wave is the rarefaction or suction wave, it generates when the inlet valve opens and the vacuum of the cylinder is exposed to the inlet manifold. This low pressure when travels from inlet valve opening to the other end of intake manifold i.e. to the atmosphere in case of single cylinder engine & gets reflected as pressure wave. The high-pressure wave then generated travels back towards inlet valve. The timing of arrival of this reflected high pressure wave can be tuned with opening of the intake valve by appropriately designing the inlet manifold. Tuning of the intake manifold in this way increases the local density of the air at the inlet. In cylinder pressure get increased at ivc causing maximum possible volumetric efficiency [5].

2. LITERATURE REVIEW

M. A. Ceviz conducted experiments to study effects of intake plenum volume variation on engine performance and emission. Brake and indicated engine performance characteristics, coefficient of variation in indicated mean effective pressure were taken into account. He concluded that the engine performance can be increased by using intake plenum volume that is continuously variable. M.A. Ceviz and M. Akin investigated the effects of intake plenum volume on the performance of a spark-ignited engine with electronic fuel injector. Si engines with multipoint fuel injection system showed better characteristics than carbureted one. The results showed that the variation in the plenum length causes an improvement in fuel consumption at high load and low engine speeds. Research work done by Jensen Samuel et. al. in their research work entitled Effect of Variable Length Intake Manifold on a Turbocharged Multi-Cylinder Diesel Engine they have validated major engine parameters with 1000hp V46-6 turbo diesel engine and the deviations were found to be less than 5% of the experimental data.[1]

Devendra Deshmukh et. al. done experiments on 125cc four stroke SI engine & simulated the same with the help of 1-D thermodynamic analytical model to study gas exchange process by intake & exhaust tuning. They analyzed the effect of intake & exhaust tuning on performance of the engine to find out that optimization results into the improved engine performance & improvement into the maximum vehicle speed. Experimental & simulated results were into good agreement which proves the usefulness of commercially available 1-D thermodynamic model [2].

L. J. Hamilton et. al. studied the effect of intake geometry on the SI engine performance in their research paper (SAE 2009-01-0302). They experimented on HONDA CBR600F4i motorcycle engine by fitting intake of different lengths in steps of 0.5m. The engine was controlled with a MOTEC programmable engine controller and performance parameters were measured using a Land and Seawater brake dynamometer coupled to the output shaft. They have calculated the engine parameters with the help of water brake dynamometer & other sophisticated measuring instruments. 1-D engine simulation code Ricardo Wave was used for computational study. The intake pressure was also measured with pressure transducer experimentally. The pressure at intake valve closure was observed to study its effect on engine performance. The key findings from study were Volumetric efficiency was seen to vary in the range of 50% to 110% and torque was seen to vary from 20 to 55 Nm over the speed range of 3,000 to 12,500 RPM. Intake pressure at IVC position showed dominant effect on the volumetric efficiency. Intake tuning resulted into the resonance charging which produce maximum volumetric efficiency. Experimental & computational investigation showed that Intake runner length has a significant effect on volumetric efficiency and torque. Bending intake runners to conform to packaging requirements appears to have no significant effect on volumetric efficiency or torque [3].

Ronak I. Sayani, Swapnil S. Ghodake, Sagar D. Mohite, Vaibhav R. Shelar, Prof. Irshad Shaikh studied the effect of intake geometry air restrictor venturi for a SAE SUPRA vehicle to achieve maximum air flow with minimum pressure drop to optimize the performance of the vehicle. SAE SUPRA competition has imposed a rule regarding air intake that all the air should pass through a single restrictor of 20mm diameter to be placed in between the throttle and engine. This rule is imposed to limit the power capability of the engine. From data obtained by several analysis gathered from ANSYS – Fluid Flow (Fluent), it can be stated that the most optimized values for the venturi are 14 and 6 degrees for converging and diverging angles respectively. [4]

Pruthviraj Vitthal Wable, Sahil Sanjog Shah, 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 20 mm 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 [5].

Anshul Singhal, Mallika Parveen, Studied that flow restriction device to be fitted in the FSAE (Formula Society of

Automotive Engineers) car being built by Team Gear Shifters of BITS-Pilani, Dubai Campus. The car is an open-wheeled race vehicle, designed to go from 0-100 kmph in under 4 seconds and have a top speed of about 140kmph. In order to comply with the rules of the competition imposed by FSAE, a single circular restrictor of 20mm diameter must be placed in the intake system between the throttle and the engine and all engine airflow must pass through the restrictor. This is done primarily to limit the power capability from the engine. An Internal Combustion Engine takes in air from the environment and the air-fuel mixture is combusted inside the engine cylinders to generate the power required to run creates a low pressure during the intake stroke, causing the air from the atmosphere to enter the cylinders. The higher the rpm, the greater the pull, and the lower the pressure created inside the cylinder. According to the stoichiometric air-fuel ratio, to burn 1 gram of gasoline 14.7grams of air is required. By reducing the diameter of the flow path from 50mm to 20mm, the flow cross-section area has reduced drastically. At low rpm of the engine when the engine requires less air, the reduction in area is compensated by the accelerated flow of air through the throat (20mm section) [6].

Vikas sharma, Studied That things are pretty simple for an engine equipped with a carburetor, because here will be the air mixed with the fuel without too many adjustments. The most advanced part of the system is an Air Temperature Sensor in the air intake, used to measure the air temperature, in order to command a flap which allows cool air to enter through a heated pipe, so as to prevent carburetor icing. On the other hand, we have now fuel-injected cars, which is a more advanced system. Air Intake Systems are designed to supply particle-free and dust-free air to a vehicle's engine. At the same time, the Air Intake System plays an important role in reducing and modifying the ambient noise coming from the engine. Air is drawn in through the air intake; a long plastic tube which has the main purpose to guide the air into a fairly steady stream until the air filter housing. After the filter, the air will go, in order, through an Air Flow Meter, Throttle Body with an Air Valve, then Air Intake Chamber, Intake Manifold, and finally will get to the Cylinders [7].

3. DESCRIPTION

To flow a maximum amount of air into each cylinder during an intake stroke, as more air means more fuel can be burned to produce useful work. Improving flow through an intake system boosts overall engine efficiency. Restrictions to airflow include an air filter, throttle body, intake plenum, and intake valves. When well designed, each device is optimized for flow. Formula SAE rules mandate that an intake system restrictor be placed in the intake system between the throttle body and engine.

Device is to have a maximum throat diameter of no greater than 0.787 inches (20 mm) for gasoline fueled engines. The restrictor serves as a design constraint for competition. The device curbs power output by impeding airflow at high RPM when airflow would typically be greatest. There are several design parameters that affect the total efficiency of the restrictor. A few of the more important include the length of the diffuser, the diffuser angle, the inlet length, and the inlet shape. Formula SAE promotes experimentation.

Thus, finding the correct combination of lengths and angles for the restrictor is a necessary task. CFD flow modeling software offers a useful tool towards this end. he maximum flow through an intake system under steady-flow conditions is useful as a means to gauge flow capacity. The intake system is considered to be choked when the flow rate reaches max. When choked a reduction in pressure downstream of the throat will result in no additional flow through the system.

A turbocharger or supercharger (a device used to boost intake manifold pressure and allowed than 2015 Formula SAE rules with the restriction that it must be placed downstream of the intake system restrictor) may improve low- and mid-range performance but will not improve flow once the intake system is choked because of its location downstream of the throat.

4. DESIGN OF VENTURI

4.1 Principal of venturi

The working of venturi meter is based on the principle of Bernoulli's equation. **Bernoulli's Statement: -**It states that in a steady, ideal flow of an incompressible fluid, the total energy at any point of the fluid is constant. The total energy consists of pressure energy, kinetic energy and potential energy or datum energy.

Mathematically,

$$\frac{p_1}{\rho g} + \frac{v_1^2}{2g} + Z_1 = \frac{p_2}{\rho g} + \frac{v_2^2}{2g} + Z_2$$

Here all the energies are taken per unit weight of the fluid.

The Bernoulli's equation for the fluid passing through the section 1 and 2 are given by

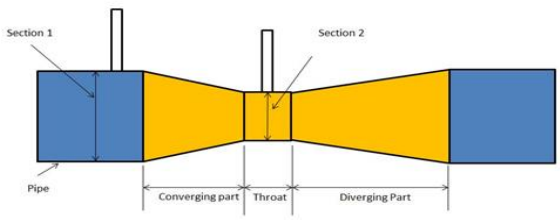

Fig-1: Bernoulli's Equation Setup

The venturi meter is used to measure the rate of flow of a fluid flowing through the pipes.

Let's understand how it does this measurement step by step.

- Here we have considered two cross section, first at the inlet and the second one is at the throat. The difference in the pressure heads of these two sections is used to calculate the rate of flow through venturi meter.
- As the water enters at the inlet section i.e. in the converging part it converges and reaches to the throat.
- The throat has the uniform cross section area and least cross section area in the venturi meter. As the water enters in the throat its velocity gets increases and due to increase in the velocity the pressure drops to the minimum.
- Now there is a pressure difference of the fluid at the two sections. At the section 1(i.e. at the inlet) the pressure of the fluid is maximum and the velocity is minimum. And at the section 2 (at the throat) the velocity of the fluid is maximum and the pressure is minimum.
- The pressure difference at the two sections can be seen in the manometer attached at both the section.
- This pressure difference is used to calculate the rate flow of a fluid flowing through a pipe.

Expression for the rate of flow through Venturi meter: -

Considered a venturi meter is fitted to a horizontal pipe through which fluid (water) is flowing as shown in the figure given below.

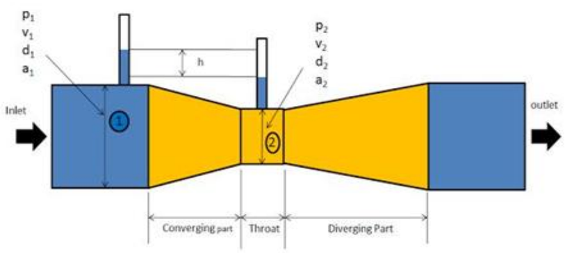

Fig-2: Venturi Meter

Let, d_1, p_1, v_1 & a_1, are the diameter at the inlet, pressure at the inlet, velocity at the inlet and area at the cross section 1. And d_2, p_2, v_2 and a_2 are the corresponding values at section 2.

Applying Bernoulli's equation at sections 1 and 2

$$\frac{p_1}{\rho g}+\frac{v_1^2}{2g}+Z_1=\frac{p_2}{\rho g}+\frac{v_2^2}{2g}+Z_2 \quad \text{.................................} (1)$$

As the pipe is horizontal, so $z_1 = z_2$

Therefore,

$P_1 - P_2)/\rho g$ is the difference of pressure heads at section 1 and 2.

$$H=(P1-P2)/\ \rho g \ \text{...} (2)$$

Substituting this value of h in equation (1), we get

$$H=(V_2^2-V_1^2)/\ \rho g \ \text{...} (3)$$

Now applying continuity equation at section 1 and 2

$$A_1V_1=A_2V_2$$

$$V_1=A_2V_2/A_1$$

Substituting this value of v_1 in equation (2) and solving, we get discharge

$$v_2=\frac{a_1}{\sqrt{a_1^2-a_2^2}}\sqrt{2gh}$$

Substituting value of v_2 in above equation

$$Q=\frac{a_1a_2}{\sqrt{a_1^2-a_2^2}}\sqrt{2gh}$$

Q is the theoretical discharge under ideal conditions. Actual discharge will be less than the theoretical discharge. The actual discharge is given by the formula,

Where C_d is the coefficient of venturi meter and its value is less than 1.

$$Q_{act}=C_d\frac{a_1a_2}{\sqrt{a_1^2-a_2^2}}\sqrt{2gh}$$

4.2 Identifying What Are Constants and Variables in The Design of Venturi:

To know the parameters which are known and unknown is important for calculating actual flow conditions inside and outside the restrictor. These parameters provide rigid base for boundary conditions to be used in CFD analysis. As from above step we have two dimensions which are fixed, so we have two dimensions on which the venturi will perform and these are converging diverging angles and length of venturi. Thus, we have defined two known and two unknown physical parameters for design of venturi. We also know that temperature at inlet is ambient and pressure at inlet is atmospheric. For boundary conditions at outlet of venturi we can have pressure, velocity or mass flow rate. Calculating pressure and velocity at outlet of venturi involves complex procedures and thus gives rise to some errors. Mass flow rate at outlet can be easily calculated by using choked flow equation [4].

Calculating maximum mass flow rate

M = 1 (choked flow)

A = 0.001256 m^2 (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

4.3 Convergence and Divergence Angle:

While designing the venturi meter different convergent and divergent angles were considered which gave the best performance was taken into consideration.

Deciding parameters - mass flow rate, pressure drop, velocity at throat, flow separation, power obtained at different RPM, Volumetric efficiency, compression ratio.

After performing different iterations and analyzing these results on CFD the final design was selected this gave the maximum performance.

The design of venturi gave the maximum mass flow rate so that plenum can be filled up in minimum time and the required amount of air is available at various engine speeds.

4.4 CAD Model of Venturi

Fig.3 shows the cad model of venturi, after performing different iterations for different values of converging and diverging angle and analyzing those results on CFD the design which gave minimum pressure drop, maximum velocity at throat, continuous flow along the walls of converging and diverging section, minimum losses, maximum mass flow rate, reduced back flow of air was considered as the optimum design for venture

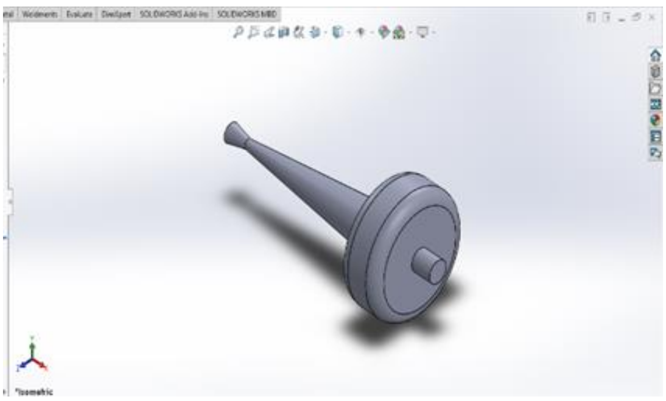

Fig-3: CAD model of venturi

5. CONCLUSIONS

This project is dedicated to designing of intake system for a restricted single cylinder engine. Intake system design includes designing of three components viz. Restrictor, Plenum and Runner. Restrictor design is based on the geometric constraints and results of steady state CFD analysis. Plenum volume is based on literature study and its shape is decided through transient CFD analysis. Runner design is based on induction wave tuning principle and its dimensions are based on empirical relations available in the literature. There is wide scope of future work related to this project. Following points illustrate areas where future work is going to be done and overall performance of Intake.

Future scope for deciding optimum plenum volume: This will be done through observing effect of different plenum volumes on engine performance through engine dynamometer testing. 3-4 different plenum volumes, mostly 1.5 times, 1.75 times, 2 times and 2.5 times engine displacement will be manufactured. One by one, these plenums will be installed on engine and power- torque curves will be obtained. The one which gives best performance amongst all will be selected as final plenum volume. This volume will be incorporated with the shape selected through procedure discussed in earlier sections.

Final dimensions of runner will be decided through runner tuning which is part of planned Engine Dynamometer testing in future.

With availability of higher processing capacity computer lab, ANSYS Fluent transient simulations will be done for more than one & half engine cycle operation and it will yield more accurate results.

6. REFERENCE

1. Ceviz, M. A. & Akin, M., 2010." Design of a new SI engine intake manifold with variable length plenum". Energy Conversion and Management.
2. Deshmukh, D., Kumar, R., Garg, M., Nayeem, M. J., and Lakshminarasimhan, V, "Optimization of Gas Exchange Process on a Single Cylinder Small 4-Stroke Engine by Intake and Exhaust Tuning: Experimentation and Simulation", SAE Paper 2004-32-0007.
3. L. J. Hamilton, J. S. Cowart and J. J. Rozich, "The Effects of Intake Geometry on SI Engine Performance" SAE Paper 2009-01-0302
4. Ronak I. Sayani, Swapnil S. Ghodake, Sagar D. Mohite, Vaibhav R. Shelar, Prof. Irshad Shaikh, "Design And Analysis Of Air Restrictor Venturi In Sae Supra", BE, Automobile Dept, TCOE, Boisar, Prof. Automobile Dept, TCOE, Boisar
5. Pruthviraj Vitthal Wable, Sahil Sanjog Shah, "Air flow optimization of venturi type intake restrictor", 2017 IJEDR, Volume 5, Issue 4, ISSN: 2321-9939

6. Anshul Singhal, Mallika Parveen, "Air Flow Optimization via a Venturi Type Air Restrictor", Proceedings of the World Congress Engineering 2013 Vol III, WCE 2013, July 3 - 5, 2013, London, U.K
7. Vikas sharma, Dronacharya College of Engineering, Gurgaon, Haryana, "Research Paper On Study Of Air Intake System", December 2015, IJIRT, Volume 2 Issue 7,