DESIGN AND CFD ANALYSIS OF THE INTAKE MANIFOLD FOR THE HONDA CBR250RR ENGINE

by

SUBHASH SESHADRI

Presented to the Faculty of the Graduate School of

The University of Texas at Arlington in Partial Fulfillment

of the Requirements

for the Degree of

MASTER OF SCIENCE IN MECHANICAL ENGINEERING

THE UNIVERSITY OF TEXAS AT ARLINGTON

August 2015

To Racing

ACKNOWLEDGEMENTS

First, I would like to thank my supervising professor Dr. Robert Woods for his continuous guidance and support over the period of my research at University of Texas Arlington. During this tenure under Dr. Woods I learnt a lot on design and simulation, he was always present to help and encourage me throughout my thesis work. I could not have imagined a better guide and advisor for my master's thesis. I convey my regards to Dr. Nomura and Dr. Agonafer as well for taking time to be on my thesis committee. I would like to thank the Maverick Racing team for working on the engine and rebuilding it. I would like to thank Jacob Bell, my project partner and brother-in-arms to have shared this incredible journey of rebuilding the engine and the intake manifold, to have gone through the highs and lows patiently. A project of this scale would not have been finished without his hard work and commitment during the past year. The UTA FSAE Racing team laid the foundations for this project. I would like to thank David Campbell, Wyatt Langston, Dylan Landrum, Girish Bangalore and the entire racing team for answering all my queries on 3D modeling and CFD analysis. I would like to thank my family, friends and for their everlasting support and inspiration.

April 27, 2015

ABSTRACT

DESIGN AND CFD ANALYSIS OF THE INTAKE MANIFOLD FOR THE HONDA CBR250RR ENGINE

Subhash Seshadri, MS

The University of Texas at Arlington, 2015

Supervising Professor: Robert Woods

The scope of this project is to improve reliability issues on a turbocharged Honda CBR250RR engine and boost the performance by redesigning the intake manifold - one of the key components of this engine package. The engine has been used in the past by the UTA Formula SAE team in several open-wheeled race cars providing a light weight platform with a high power to weight ratio. Previous intake manifolds did not provide balanced distribution of air flow among all cylinders. This lead to incorrect air/fuel ratios were some cylinders ran too lean at times, while others were running too rich. Incorrect air fuel ratios can lead to excessive exhaust gas temperatures, pre-ignition detonation, causing components such as head gaskets and pistons to fail. The new manifold is aimed at tackling these issues by supplying uniform amount of air to all four cylinders. 3D modeling is done using SolidWorks and the internal flow distribution will be calculated

and tuned using CFD analysis on Star CCM+. A 1D simulation of the complete engine was set up using Ricardo Wave to optimize the runner lengths and to predict the performance of the engine. The intake will be manufactured out of thermoplastic using rapid prototype printing or what is also called as 3D printing.

TABLE OF CONTENTS

ACKNOWLEDGEMENTS........iv

ABSTRACT........v

LIST OF ILLUSTRATIONS........x

LIST OF TABLES........xiii

Chapter 1 INTRODUCTION........1

1.1 Honda CBR250RR........1

1.2 Research Motivation........2

1.3 Application........4

1.4 Thesis Objective........6

1.5 Thesis Organization and Contribution........8

1.5.1 Thesis Organization........8

1.5.2 Contribution........9

Chapter 2 LITERATURE REVIEW........10

2.1 Intake Manifold........10

2.2 Gas Flow through the Intake of the Engine........10

2.3 Design Theory........11

2.3.1 Resonance Charging........12

2.3.2 Dual Volume Plenum........14

2.3.3 Single Volume Plenum........16

2.4 Computational Fluid Dynamics........18

Chapter 3 INTAKE MANIFOLD DESIGN 20
3.1 2015 Formula SAE Powertrains Rules 20
3.2 One Dimensional Simulation 21
3.2.1 Ricardo Wave 22
3.2.2 Modeling the Turbocharger 24
3.2.3 Valve Profile 25
3.2.4 Intake Runner Optimization 25
3.2.5 Predicted Performance of the Engine 28
3.3 Applied Design 29
3.4 3D Models 33
Chapter 4 CFD ANALYSIS 35
4.1 Calculations 35
4.2 Flow Bench Test Results 37
4.3 CFD on 2015 Single Volume 42
4.3.1 CFD Results 46
4.3.2 Pressure and Velocity Contours 50
Chapter 5 INTAKE MANUFACTURING 54
5.1 Rapid Prototype Printing 54
5.2 Assembly 54
Chapter 6 CONCLUSIONS AND FUTURE WORK 57
6.1 Conclusions 57

6.2 Future Work 58
Appendix A HONDA CBR250RR ENGINE SPECIFICATIONS 59
Appendix B PISTON VELOCITY CALCULATIONS................................. 61
Appendix C 3D MODELS 65
Appendix D PHYSICS MODELS USED IN CFD ANALYSIS................... 69
Appendix D PPSF MATERIAL PROPERTIES... 73
REFERENCES.. 76
BIOGRAPHICAL INFORMATION... 78

LIST OF ILLUSTRATIONS

Figure 1.1 Honda CBR250RR Motorcycle Series 1

Figure 1.2Two engines being rebuilt. (a) is MC14E-1157776

and (b) is MC14E-1252690.. 3

Figure 1.3 (a) UTA Race car and team-1998

and (b) UTA Race car and team-2006... 5

Figure 2.1 Column of air-fuel inside the intake port 13

Figure 2.2 Cross-section of a Dual Plenum Manifold................................. 15

Figure 2.3 Flow chart to run CFD analysis 19

Figure 3.1 Intake System Sequence... 20

Figure 3.2 Ricardo Wave Model for CBR250RR 23

Figure 3.3 Mapping Turbine Data using Grabit!... 24

Figure 3.4 (a) is Dial Gauge setup on the camshaft

and (b) Timing Wheel setup on the crank shaft 25

Figure 3.5 Intake Runner Length Optimization 27

Figure 3.6 Brake Power Vs Engine Speed 28

Figure 3.7 Simple Radius Bellmouth 29

Figure 3.8 Airfoil Profile 30

Figure 3.9 Elliptical Profile 30

Figure 3.10 Cross-section of Runner 3.. 31

Figure 3.11 2015 Intake Manifold Assembly.. 33

Figure 4.1 Piston Diagram 35

Figure 4.2 Piston Velocity profile 36

Figure 4.3 Flow Bench trials on F06 Intake Manifold 37

Figure 4.4 Flow Bench results on F06 Dual Volume Plenum Intake 39

Figure 4.5 Flow Bench results on F06 Single Volume Plenum Intake 40

Figure 4.6 Flow Bench results comparison of F06 Single

Volume Plenum Intake Vs F06 Dual Volume Plenum Intake 41

Figure 4.7 (a) is Design1 and (b) is Design2 42

Figure 4.8 (a) is Design 3 and (b) is Final Design 43

Figure 4.9 Volume Mesh on Star CCM+ 45

Figure 4.10 CFD results on 2015 Single Volume Plenum Intake 47

Figure 4.11 CFD results comparison of 2015 Single

Volume Plenum Vs F06 Single Volume Plenum 49

Figure 4.12 Velocity Profile in Runner 1 at Mach0.4 50

Figure 4.13 Velocity Contour in the Plenum for Runner1 at Mach0.4 52

Figure 4.14 Pressure Contour in Runner 1 at Mach0.4 53

Figure 5.1 2015 Single Volume Plenum Intake Manifold Assembly 55

Figure 6.1 Results comparison of F06 Single Volume Plenum-CFD

Vs F06 Single Volume Plenum-Flow Bench 57

Figure C.1 Overall Dimensions 1 Bottom Shell 66

Figure C.2 Overall Dimensions 2 Bottom Shell 66

Figure C.3 Overall Dimensions 1 Top Shell.. 67

Figure C.4 Overall Dimensions 2 Top Shell.. 67

Figure C.5 Honda CBR250RR Engine Assembly.................................... 68

Figure C.6 F06 Chassis with engine assembly.. 68

Figure D.1 PPSF Material Spec Sheet 1 74

Figure D.2 PPSF Material Spec Sheet 2 75

LIST OF TABLES

Table 4.1 CFD results for Volumetric Flow Rate (scfm)

on the 2015 Single Volume Plenum Intake 46

Table 4.2 CFD results for Pressure Drop (kPa)

on the 2015 Single Volume Plenum Intake 47

Table 4.3 Discharge Coefficients on Single Volume Plenum 48

Table 4.4 Percent difference between Discharge Coefficients 48

Chapter 1

INTRODUCTION

This chapter explains the problem statement for this thesis research work. It also describes the motivation for carrying out this research, application of the intake manifold and Honda CBR250RR engine, along with a brief description on the thesis organization.

1.1 Honda CBR250RR

Figure 1.1 Honda CBR250RR Motorcycle Series

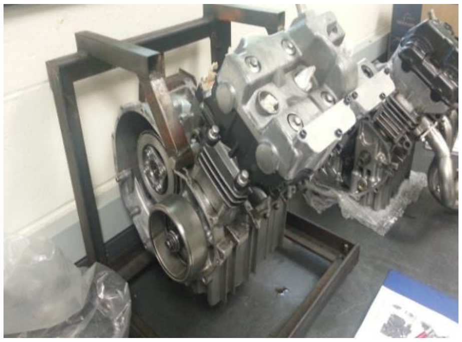
The Honda CBR250RR is a world renowned 250cc sports bike manufactured in Japan between 1986 and 1996, widely regarded as one of the fastest, most powerful and agile learner bikes available. The MC22(Chassis Number) can be found all over the world, and is particularly popular in Australia and New Zealand as the most powerful learner license bike available. It distinctively screams out to high engine speeds yet enjoys a long engine life of 100,000+km. It is a lightweight sport bike with a six-speed gearbox and a 250 cc (15 cu in), four-cylinder, four-stroke engine capable of revving up to 19,000 rpm. Light weight and compact, this racing series bike used a relevantly tiny inline four cylinder to power this machine. What the engine lacks displacement wise was compensated for in the redline range. Minor alterations in the engine during its life cycle lead engine models to produce up to 40 to 45 horsepower when naturally aspirated. The engine specifications are provided in Appendix 1.

1.2 Research Motivation

Formula SAE (FSAE) is an international university-level competition, where university students design, build, and race four-wheel formula-style race cars. Teams are judged based on design, cost, manufacturability and performance on track. The different events have varied point allocations. In order to win, the team should have a car that can maximize points in every event. The students have less than a year to accomplish this.

The race car's performance varies greatly with the type of engine running it. A bigger engine with a higher power capacity can provide much needed power to the car at higher RPM through the later part of the track whereas a smaller engine can save a lot of weight on the car providing quicker acceleration at launch. Since the CBR250 was designed for street riding longevity it makes for a great choice for Formula SAE spec cars, providing easy maintenance, long life cycles on parts and a relatively high power to weight ratio.

(a)



(b)

Figure 1.2Two engines being rebuilt. (a) is MC14E-1157776 and (b) is MC14E-1252690

There are 7 Honda CBR250 engines that were manufactured between 1988-1996 available to be rebuilt and planned to be used on the UTA FSAE race cars. Of these, 2 engines (Engine Number-MC14E-1157776 & MC14E-1252690) are in the process of rebuilt; started back in the summer of 2014 as a part of my course work project. To use this engine on the FSAE racecars and adhere to the rules of the SAE national competitions, certain modifications are made on the stock engine which include redesigning the intake manifold, changing the oil supply system, adding a new turbocharger and changing the fuel injectors and modifying the electronic system.

1.3 Application

The intake manifold is designed to specifically run the four cylinder 250cc CBR engine. Inspiration to use this engine was found during the late nineties by the team members of the Formula SAE at the University of Texas at Arlington. With rules limiting engine displacement to 610cc or less, motorcycle power plants with their internally packaged gearboxes are an ideal choice for the competition. With horsepower being restricted by a spec venturi on the air intake, power to weight ratio plays a crucial role in the development of these race cars. UTA had been using 450cc and 600cc motorcycle engines, naturally aspirated or turbocharged, producing between eighty to ninety horsepower. The new intake manifold aims to harness equal power from the CBR250 with the assistance of a turbocharger, which is relatively small compared to these bigger engines and make it a light weight power alternative,.

(a) *(b)*

Figure 1.3 (a) UTA Race car and team-1998 and (b) UTA Race car and team-2006

The idea was first implemented in 1998 on the UTA FSAE car and repeated again on the 1999 chassis. However, to much regret the promising package did not deliver during the FSAE competitions due to various failures. The 1999 car did manage to win the Formula Student competition in Great Britain, proving its effectiveness against heavier, big engine cars. In 2006 the concept was resurrected once again, to power an FSAE car using a turbocharged CBR250 engine. Personnel and design faults lead the project to being unsuccessful resulting in mechanical failure, etc. Furthermore, attempts to salvage this engine package in late 2010 also lead to failure due to incorrect air/fuel intake charge distribution caused by an improperly designed air intake. Overall, the engine package has run intermittently successful.

An effort to breathe new life into these engines is being made by revising the entire design concept, replacing the engine components and addressing key reasons for failure properly. Of the two engines being rebuilt, one will power the 2006 race car and be used in future competitions. The other will be used an option for the new cars. Every year, there is debate on changing the engine package, the biggest deciding factor being the power to weight ratio among others which include engine rebuilt intervals, long life cycles on parts. Since the CBR250 is a light weight engine designed for longevity and reliability, increasing its output should be manageable as long as key issues are addressed properly. With further development this engine promises to be a very effective power option for the use in future UT Arlington SAE race cars.

1.4 Thesis Objective

Previous intake manifolds did not provide balanced distribution of air flow among all cylinders. This lead to incorrect air/fuel ratios; where some cylinders ran too lean at times, while others were running too rich. The concept of this research is to redesign the intake manifold to tackle this problem and increase the engine's power output by tuning the manifold to have maximum mass flow of air with minimum pressure drop across the intake ports.

The objective of this research is accomplished by carrying out the following steps:-

1. Flow through previous intake manifolds is visualized on a flow bench and CFD analysis. These results are used as a bench mark for the new design.
2. The mathematical model of the powertrain is modeled using Ricardo Wave. This model is used to tune the intake system to provide an additional boost of 8-10 psi using a turbocharger and optimize the intake manifold geometry.
3. The 3D model is drawn using SolidWorks and Computational Fluid Dynamics Analysis is done using Star CCM+.
4. The CFD results are compared to the bench mark values with an aim to maximize the mass flow of air with minimum pressure drop. Flow distribution across the four ports is visualized using graphical contours in Star CCM+ to provide balanced distribution of air.
5. The intake will be manufactured out of PPSF (polyphenylsulfone) using rapid prototype printing or what is also called as 3D printing.

1.5 Thesis Organization and Contribution

1.5.1 Thesis Organization

Chapter 2 provides a literature review of the role of an intake manifold in the combustion process. This chapter explains the physics of gas flow in an internal combustion engine. It also explains the design theory of the intake manifold and the design theories previously used in the UTA FSAE Racing team. It provides an insight into the science of computational fluid dynamics of incompressible internal flow of air. Finally it gives a brief description of the specs of 3D printing and material handling.

Chapter 3 describes the design of the intake manifold and the platform used to implement the design. It also explains 1D Simulation and 3D modeling associated with this design.

Chapter 4 studies the theory of the fluid dynamics in an internal combustion engine and CFD analysis done on the 3D model using Star CCM+. It also discusses the flow bench experiments done on the previous intake manifolds and the results obtained which are then set as a benchmark to compare the CFD results. This chapter also talks on the CFD set up on the oval runner intake and the post processing carried out on the results obtained.

Chapter 5 gives a brief introduction on rapid prototype printing and the material used to manufacture this manifold.

Chapter 6 discusses the CFD results of the new design in comparison to the experimental and CFD results of the old manifolds.

Chapter 7 gives a summary of the findings of this research work and details the conclusions reached. It also discusses several ideas for future work to further improve the flow distribution and reduce the pressure drop across the manifold.

1.5.2 Contribution

The main contribution of this research work is the redesigned intake manifold that aims at achieving a uniform distribution of air across the four intake ports of the engine. Incorrect air fuel ratios can lead to excessive exhaust gas temperatures, pre-ignition detonation, causing components such as head gaskets and pistons to fail. One such unbalanced intake was part of the cause for engine failure in the past. This research effort attempts to refine the old design theory and improve the design to accommodate the most recent developments brought about on the engine.

Chapter 2

LITERATURE REVIEW

2.1 Intake Manifold

The intake manifold connects the throttle body to the intake ports on the cylinder head. A manifold has a plenum or an air chamber where air is stored and a set of runners connected to the intake ports of the engine on the engine head. With port fuel injection, only air flows through the manifold and the top half of the runner. Fuel is injected into the air as it flows through the intake ports. When the throttle valve opens, air flows into the plenum. During the intake stroke, the intake valve opens and air-fuel mixture flows through the runners into the cylinder where combustion takes place. This auto part is not just a passageway for the mixture to flow into but it also contributes to a better distribution of the fuel and air.

2.2 Gas Flow through the Intake of the Engine

The gas flow processes into, through and out of the engine are all unsteady. Unsteady gas is defined as that which the pressure, temperature and the gas particle velocity in a closed duct are variable with time. In the case of induction flow into the cylinder through an intake valve whose area changes with time, the intake pipe pressure alters because the cylinder pressure is affected by the piston motion causing volumetric change within that space. The flow in and out of an internal combustion engine all happens in pulses and for this reason air can be visualized to traveling as pressure waves. Pressure waves are of two types. They are either compression waves or expansion waves. Compression waves are pressure waves where the pressure on any point on the wave is below the ambient pressure and the opposite holds true for expansion waves. The propagation of pressure waves and the mass flow rate that they induce is dependent on the gas properties, the specific heats, pressure and temperature.

2.3 Design Theory

Various design theories have been suggested throughout the history of intake manifold development. The basic layout of the fuel injection manifold will be determined by its application. A racing application will generally tend toward a design with one throttle plate per cylinder. Typically, a street manifold will employ just one throttle plate or one multi-plate progressive throttle body attached to a plenum that that will feed all cylinders. Because we are dealing with everyday user motorcycle engine, we opt for the single throttle body design because it generates a considerably crisper intake manifold vacuum signal. This greatly increases the accuracy with which low speed fuel and ignition can be calibrated and is this thus better suited to a street-driven engine. Of the single plate throttle two plenum design theories have been prominent in the UTA FSAE Racing team. The dual plenum and single plenum design. Though both design theories have their own benefits and disadvantages, the main deciding factor to select one of these plenums is based on a concept called resonance charging.

2.3.1 Resonance Charging

Let us visualize the intake cycle of the engine as air flowing through the intake manifold runner, past the intake valve, and into the cylinder. Everything is fine and dandy until the intake valve shuts. Here is where the law of inertia comes to play -- because the air was in motion, it wants to stay in motion. But the air can't go anywhere because the valve is shut so it piles up against the valve like a chain reaction accident on the freeway. With one piece of air piling up on the next piece of air on the next on the next, the air becomes compressed. This compressed air has to go somewhere so it turns around and flows back through the intake manifold runner in the form of a pressure wave.

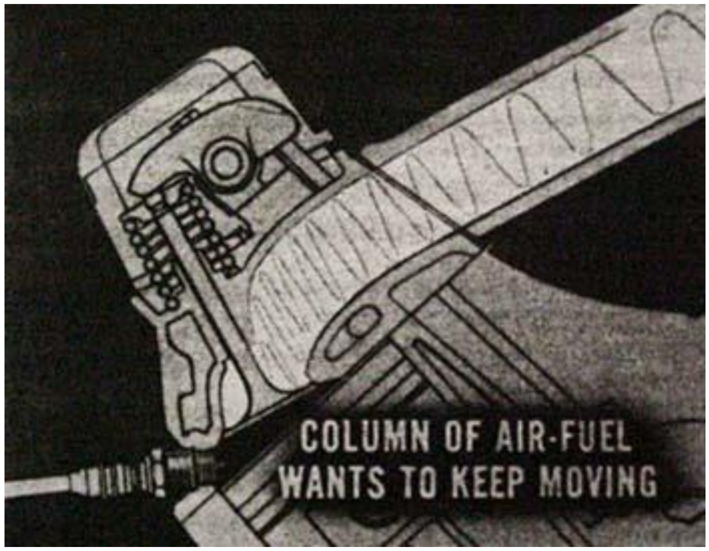


Figure 2.1 Column of air-fuel inside the intake port

This pressure wave bounces back and forth in the runner and if it arrives back at the intake valve when the valve opens, it is drawn into the engine. This bouncing pressure wave of air and the proper arrival time at the intake valve creates a form of supercharging.

In order to create this supercharging, all of the geometric variables of the intake manifold have to be aligned so the pressure wave arrives at the intake valve at the right time. This combination of synchronized events is known as 'resonant conditions'.

The following parameters affect resonance charging:-

1. Engine speed
2. The number of crank rotation degrees the intake valve is closed

3. Length of the intake runner tube
4. Volume of the plenum
5. Diameter of the intake runner

By altering one of the parameters to a suitable value resonance charging or ram induction can be achieved by synchronizing the pressure wave to the valve opening. Now that we have established the principle of resonance charging let us see how the two plenum design are applicable to our requirements.

2.3.2 Dual Volume Plenum

One of the dominant design theories in the previous decade has been implementing a partitioned dual plenum intake. As the name suggests the plenum is split into two halves with one half catering to the two inner cylinders and the other to the two outer cylinders.

Dual partition manifolds generally benefit by improving mid-range torque response. One of the phenomena occurring in such plenums is the advantage of volume resonance charging. Instead of one large plenum resonating depending on engine speed, dual plenums can be tuned to where a positive resonance within one of the volumes aids the intake charge in a runner of the other volume.

Figure 2.2 Cross-section of a Dual Plenum Manifold

The concept of dual plenum partition might seem somewhat conceptually ideal for restricted engines, but a correct balanced application is unlikely to be achieved based on geometric restrictions. One such intake was part of the cause for the most recent engine failures. The inner cylinders were seen to steal air from the outer two due to unbalanced resonance and this made the outer cylinders lean and the inner two run rich. This led to burnt pistons and eventually the engine sized.

Figure 2.3 F06 UTA FSAE Dual Volume Plenum Intake

2.3.3 Single Volume Plenum

This concepts deals with a single plenum catering to all the four runners. With motorcycle four-cylinder engines turning well over 18,000 rpm, dual plenum design theory can be applied to aid the mid power range. However, street cars with these types of systems or dual partitioned manifolds always incorporate a valve which connects the dual partitions into a single volume during higher rpm operation. During high-speed/high-load operations an intake runner ideally wants an infinite volume to draw from in order to minimize losses. During high speed operations seen by motorcycle engines, a larger single volume intake is more beneficial since the intake volume starts acting unanimous at those cycle speeds, giving each cylinder a larger volume to draw from.

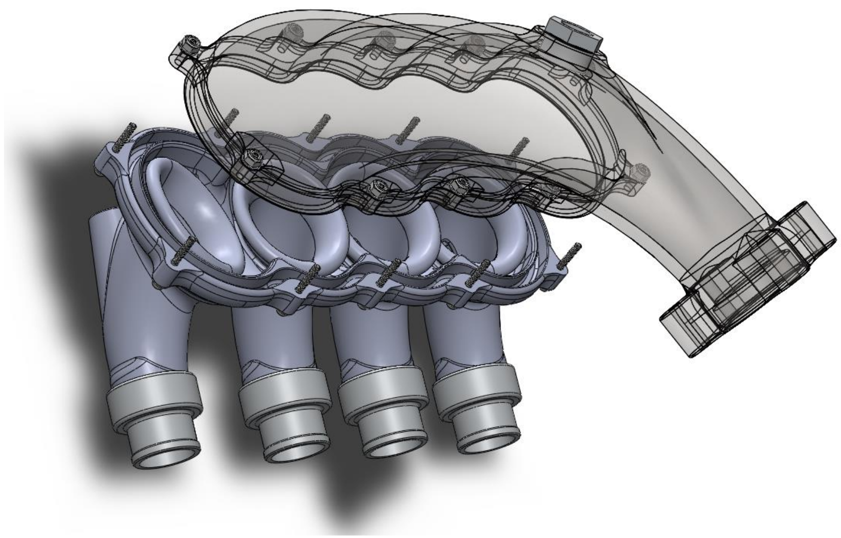


Figure 2.4 F06 Single Volume Plenum Manifold

On restricted engines, such as used in FSAE, where the top end of larger displacement engines becomes choked off anyways, the performance tradeoffs between a single large plenum top end manifold and a partitioned dual volume manifold may be debatable. The hard part is to incorporate the concept where partition resonance and volume are balanced within package. It is possible to build an intake manifold where all four runners share equal geometry as well as the equal geometric dual plenums, but the runners require being lengthy, therefore inducing a higher pressure drop leading to a loss of power. A manifold partitioned or not, must have all intake runners respond equally to static pressure drops as well as dynamic resonance response.

2.4 Computational Fluid Dynamics

Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the millions of calculations required to stimulate the interaction of fluids and gases with the complex surfaces used in engineering. Even with simplified equations and high-speed supercomputers, only approximate solutions can be achieved in many cases. The most fundamental consideration in CFD is how one treats a continuous fluid in a discretized fashion on a computer. One method is to discretize the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid, and Navier-Stokes equations for viscous flow). In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristics of the former is that each cell must be stored separately in memory. Perhaps the most important reason for the growth of CFD is that for much mainstream flow simulations, CFD is significantly cheaper than wind tunnel testing and will become even more so in the future. The improvement in computer hardware and numerical has given us a platform to simulate extreme physical conditions like higher Reynolds Number, higher Mach number, higher temperature etc.

The total process of determining practical information about problems involving fluid motion using computational fluid dynamics happens in three stages.

1. Apply the governing solution involving partial differential equations as a system of discretized algebraic equations so that the computer can be used to obtain the solution.
2. Solve the equations using the known boundary conditions.
3. Deduce the behavior of the flow using numerical data, plots and animations.

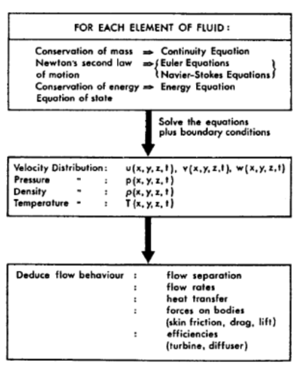


Figure 2.3 Flow chart to run CFD analysis

Chapter 3

INTAKE MANIFOLD DESIGN

3.1 2015 Formula SAE Powertrains Rules

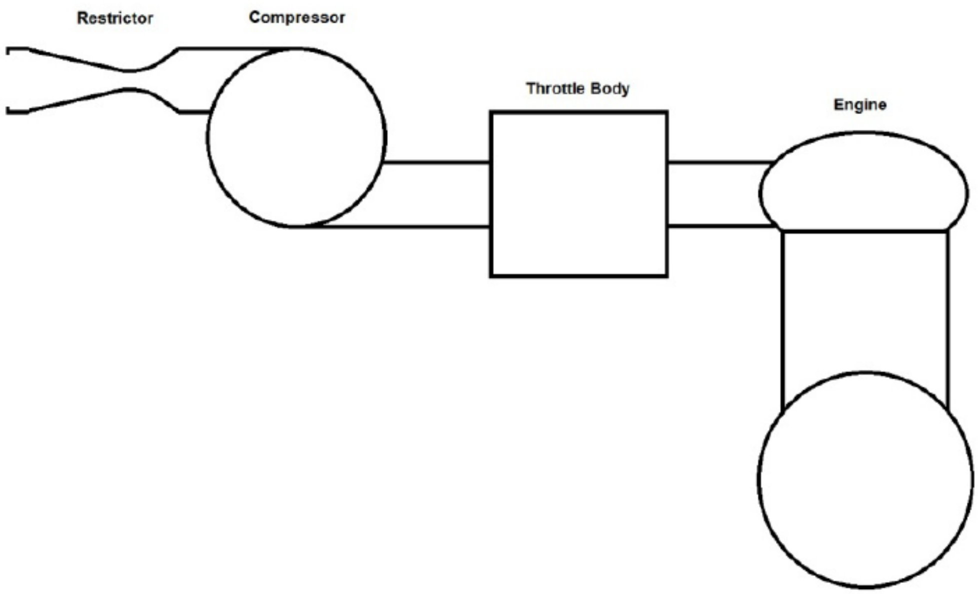


Figure 3.1 Intake System Sequence

2015 FSAE rules state that for a turbocharged or a supercharged engine the sequence should be restrictor, compressor throttle body and engine. This means that the Intake Manifold sits between the throttle body and the engine and plenums anywhere upstream of the throttle body are prohibited.

3.2 One Dimensional Simulation

Now that the gas flow process and the unsteady nature of gas flow in, through and out of the four stroke engine has been established, I will now dwell into the application of 1D Simulation using the available theoretical engine models which can be solved or used to index measured data, on a digital computer. A complete simulation is capable of predicting all of the pressure, temperature and volume variations with time within an engine of specified geometry and of calculating the resulting performance characteristics of power, torque, fuel consumption and air flow; most of the exhaust emission components; and the intake and exhaust noise levels.

Because of the infinite number of conceivable combinations of intake and exhaust geometry that can be used with various types of engine geometry, it is impossible to simulate every combination virtually. Perhaps one of the most useful aspects of engine simulation is that it allowed me to imagine the temporal variations of pressure, temperature, volume and gas flow rate that takes place during the engine cycle. The output generally in the form of numbers and graphs of the analysis helped me observe these parametric changes in a visual manner. The simulations done on this research project are to show the effects of intake tuning and turbocharger matching on the ensuing performance characteristics.

3.2.1 Ricardo Wave

Wave is a 1D engine simulator that works by performing a gas dynamics analysis on a set of ducts set up by the user. The ducts can be set up to simulate an intake. To build a wave model first a complete list of all engine geometric parameters must be created. For a 1D model values for length, diameter, and volumes are entered to build the intake and exhaust system. For the basic model starting with ambient air, an intake inlet was created, and was connected to a throttle body. This was then connected to an orifice to simulate a restrictor, and then a diffuser was connected to a plenum. The plenum is four ducts with a volume assigned to each and they are connected with mass less ducts meaning that the four volumes will act as one. To finish the intake system runners are connected, ducts with a diameter and length, and connected to the intake ports. The exhaust system is basically ducts that are connected to y-junctions, to simulate the collectors. For the engine, values are entered for the number of cylinders, firing order, bore, stroke, and number of valves. The valves are then modeled by assigning lift values and use a high performance valve flow coefficient. This is all the information needed to perform a wave simulation. Once completed, the results can be compared to real world results to calibrate the model, by altering surface temperatures, friction values, and heat transfer coefficients. Default values were used for these variables as they proved to provide usable results. The basic model is shown below.

Shown below is a model of a CBR250RR engine, with 2015 intake and the F06 exhaust models. This model will be altered to perform intake and exhaust optimization. This model was simulated without a turbocharger to begin with. From this standard starting model a simulation was ran to give us a base line performance starting line. To this baseline model, the turbocharger was added to see the performance improvement and results were compared with each other.

To model the turbocharger and the engine, a couple of experiments were conducted which will be discussed later on in this chapter.

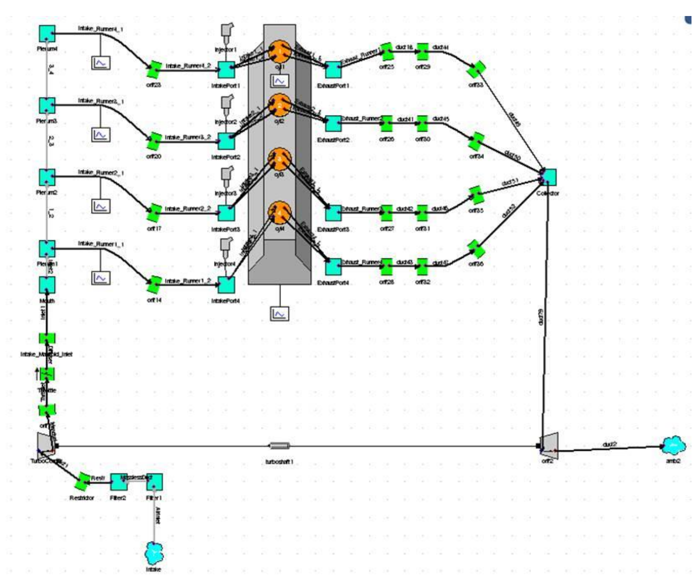


Figure 3.2 Ricardo Wave Model for CBR250RR

3.2.2 Modeling the Turbocharger

The turbocharger comes as a default model in the Wave that contains of three components. The turbine, compressor and the shaft. Turbine and compressor maps were loaded to Wave which uses these maps to interpolate data. A software tool called GrabIt! *(See reference)* to collect data points from the turbo and compressor maps pictures. Grab It! is a Microsoft Excel based application that digitizes data from pictures. Graphs and charts can have data point values digitized or have angle and distance measurements made on scanned photos. Skewed graphs are handled automatically (sometimes scanning isn't perfectly straight) as well as linear, log, date or time charts.

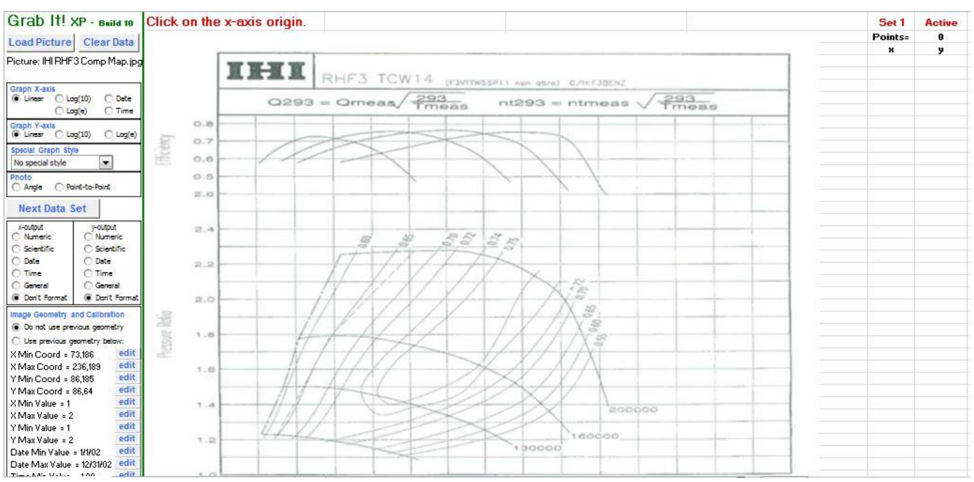
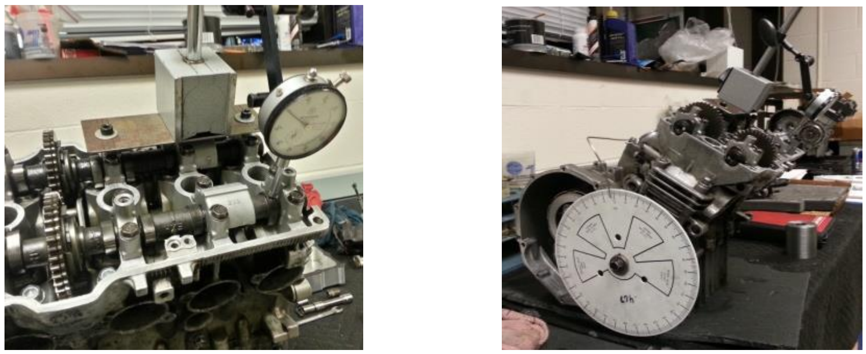


Figure 3.3 Mapping Turbine Data using Grabit!

3.2.3 Valve Profile



(a) (b)

Figure 3.4 (a) is Dial Gauge setup on the camshaft and (b) Timing Wheel setup on the crank shaft

To build a wave profile on the Wave, a dial gauge was mounted on to the cam shafts and the cam lift was tabulated for a complete engine cycle. The crankshaft was rotated for a total of 720° and the lift was noted on the dial gauge for each degree of rotation of the crank shaft. A timing wheel was mounted on the crankshafts to tabulate the crank rotation. The diagram below shows the experimental set up to tabulate the valve profile.

3.2.4 Intake Runner Optimization

The intake system is the most important system to design for this engine, as it has to be completely altered from the stock system. To start the system off a single 36mm throttle body is chosen as that provides the best compromise of throttle response and power. Having too small of a throttle will choke the engine, while having too large of a throttle will make it very sensitive. This throttle is then connected to a 19mm restrictor whose design is similar to the one used on the F13 race car. From there the manifold can begin to be modeled. The actual intake has slightly curved runners, to make the intake path as straight as possible. This cannot be perfectly modeled in Wave so a log intake with the same volume was modeled. The runners were then modeled, keeping the same length and diameter, and a bend was added as in the real intake.

The main purpose of this study is to look at the effect of the plenum volume and runner lengths. The diameters of the runners are fairly fixed as making them a different size than the head will cause a flow problem. Wave has the ability to set geometric values as variables and run an experiment where the geometry is varied between given ranges. The results can then be plotted against results like HP and torque, and the geometry can be displayed that maximizes performance. For the intake experiment the runners were varied from 50mm to 300mm, and the plenum volume was varied from 600cc to 3000cc. A screen shot of the experiment is shown below.

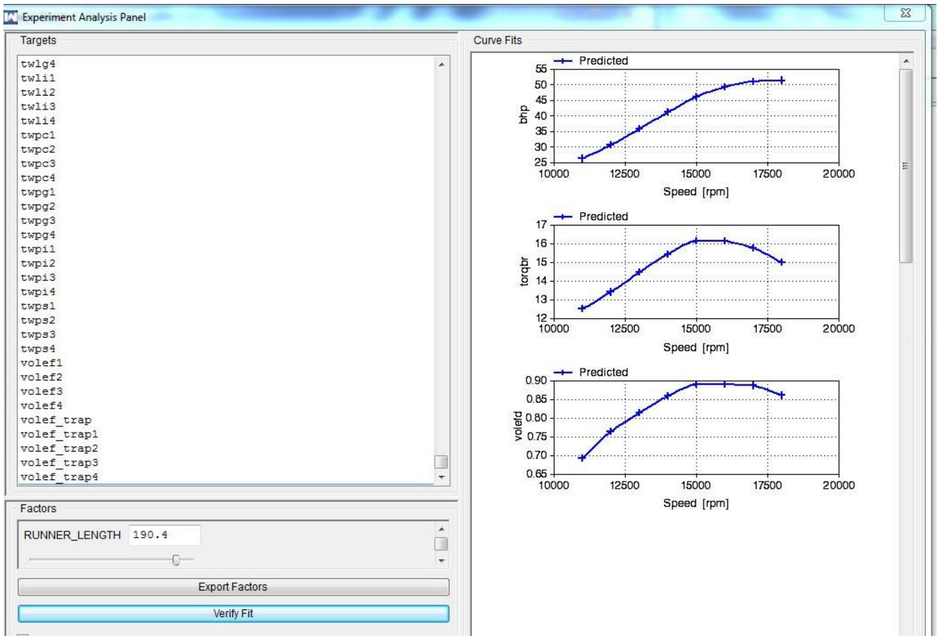


Figure 3.5 Intake Runner Length Optimization

As seen from the results above the optimum runner length is 190mm including the intake port and the hose length. The intake port and the hose make up to 60 mm in length. This fixes the runner length to 130 mm. For the plenum volume it is shown that the larger the plenum the better as this helps to reduce stealing from one cylinder to another. The downside to a very large plenum is that a very large plenum causes server throttle response degradation. Also a large plenum reduces the pulses seen by the restrictor, increasing power. This provides a very complicated situation. However over many years of experimentation, and several independent studies it has been found that for optimum power and throttle response 3 –

3.6 times the displacement of the motor is optimum for the plenum size. The current intake has a plenum that is 3.5 times 250cc, i.e. 875 cc.

3.2.5 Predicted Performance of the Engine

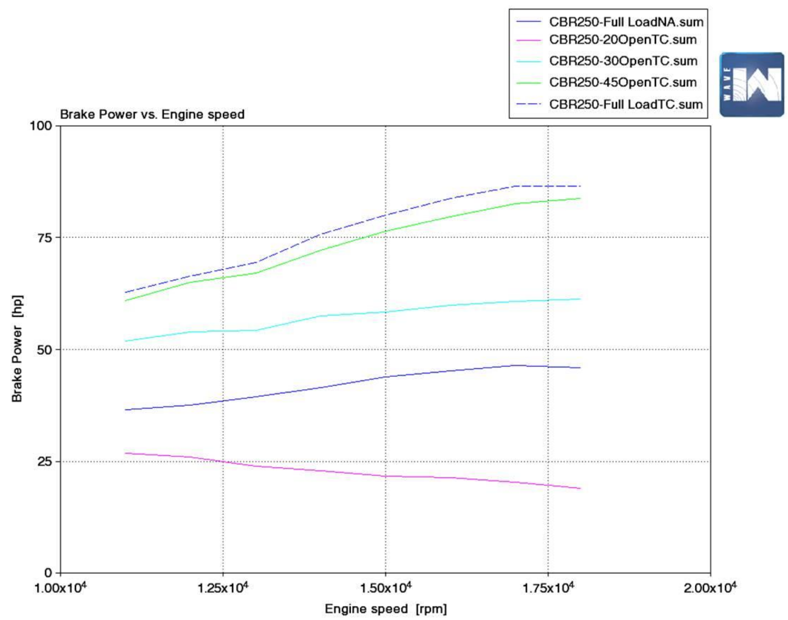


Figure 3.6 Brake Power Vs Engine Speed

Different experiments were set up in Wave to visualize the performance of the engine at partially open throttle to full open throttle for engine speed within the range of 11,000 rpm to 19,000 rpm. As it can be seen from the above graph the engine is expected to pump a maximum of 75 HP of power at 17,500 rpm at full open throttle with a boost of 10 psi using a turbocharger.

Through optimization experiments the geometric features of the intake manifold are found to be:

1. Runner Diameter – 36 mm *(= intake port inner Diameter)*
2. Runner Length – 130 mm
3. Plenum Volume – 875 cc *(= 3.5 times 250cc)*
4. Mouth Diameter – 36 mm *(= throttle body diameter)*

3.3 Applied Design

The design of a bellmouth at the end of the intake tract of an engine has not occupied much pages of the technical literature before. But it should not be concluded that this topic is not of any real significance. There is a considerable amount benefit in mass flow rate by the addition of even a simple radius to the end of a runner. There are three profiles of bellmouth that have been analyzed before.

1. A semi-ball wrap-round radius installed at the end of pipe.

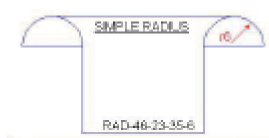


Figure 3.7 Simple Radius Bellmouth

2. A bellmouth with an airfoil profile (NACA Type)

Figure 3.8 Airfoil Profile

3. A Bellmouth with an elliptical profile

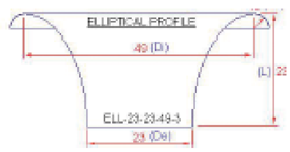


Figure 3.9 Elliptical Profile

All bellmouths are characterized by their basic data for type, length, exit diameter, entry diameter and entry corner radius. Even though the design of the intake bellmouth is not vital to good engine breathing, in racing, where the last few hp is the difference between winning and losing, the design specifications should not to be ignored. Previous studies have shown that the elliptical profile comes out as the winner over the other two profiles. In the all-important pressure ratio range up to 1.1 one can conclude that the best bellmouth has an advantage in discharge coefficient terms of some 3.5% over the simple radius bellmouth. In design terms, we can usefully conclude that “short and fat” is best with an optimum length criterion of one diameter, and an optimum entry diameter of some 2.13 times the exit diameter, and with an elliptical profile. Although the investigations are not presented here, the corner radius can be usefully designed as 0.08 times the entry diameter.

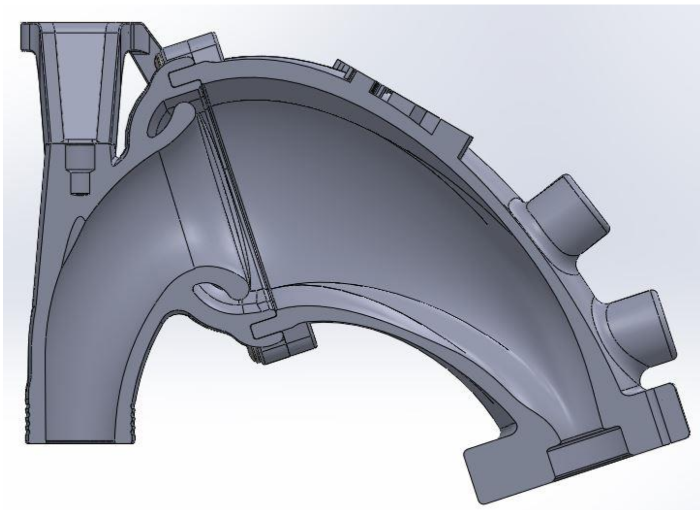


Figure 3.10 Cross-section of Runner 3

To reduce losses, the intake runners were given venturi style elliptical bell-mouth entrances. To further reduce the pressure drop and wall loses, the runners tapper heavily at a ratio of 2.5:1 from bell-mouth to the cylinder head port. To allow for close packaging of runners within the plenum, the bell-mouths were designed in an elliptical shape. This allows a relatively large intake area to draw from while keeping the plenum volume compact.

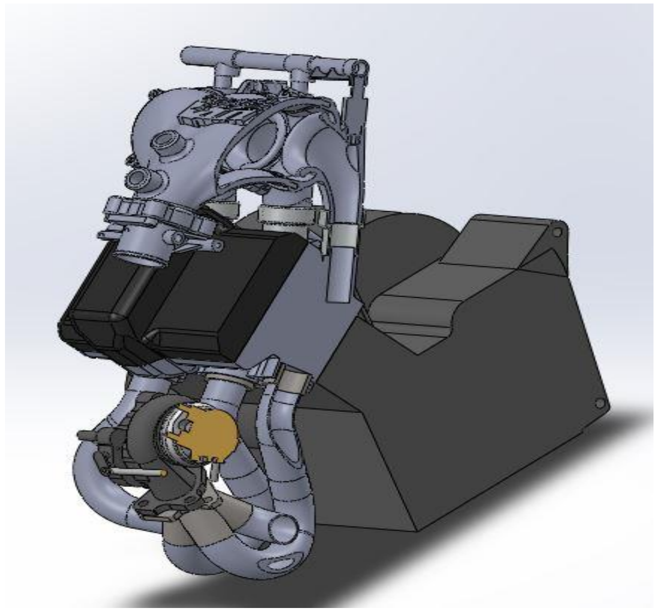


Figure 3.12 Cross section of the Intake Manifold mounted on to the Head

The manifold will be built using rapid prototype three dimensional printing. UTA has successfully built a RPT manifold before which was used on the F09 car. The process involves splitting the design into two shells which will later be joined together by epoxy. It is mandatory to design it as two half's for the printing filler material must be broken out of the printed form, being part of the manufacturing process.

In addition to previous semesters design, the intake has been update by further reducing plenum volume. The semester one intake had a volume of roughly 2100cc. By curving the intake inward between the bell mouth runner inlets, the volume was further reduced to below 1100cc. The additional curvature rib structure also increases rigidity and improves the overall pressure vessel design. The smaller volume reduces the chances of the building up unwanted velocity vortices in the plenum that would increase the pressure drop across the manifold.

3.4 3D Models

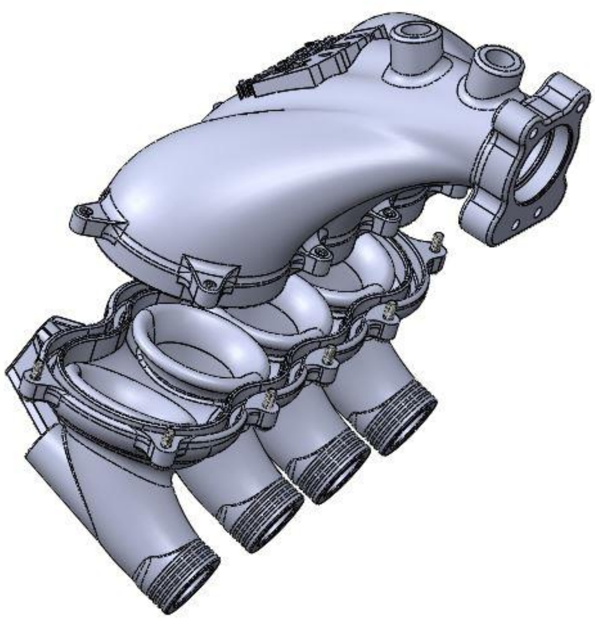


Figure 3.11 2015 Intake Manifold Assembly

The intake ports on the engine are not symmetrically spaced from each other. This gives to rise to a huge issue of runner 4 getting less air than the others if the plenum design does not supply adequate amount of air. So a total of four different 3D models were created using SolidWorks 2015 with different runner orientations. The final design has an upper shell that is symmetric over the axis of the mouth of the plenum and placed exactly at the center of the engine head. Different runner orientations were then analyzed using CFD and the orientation that had the least pressure drop was finalized. The volume of the plenum was also varied between 1800 cc to 1030 cc to see its effect on the velocity vortices. The plenum takes a 90° bend across the engine head to orient towards the compressor. The inside loft of the plenum was modeled to provide a streamline flow towards the runners and to minimize separation of flow near the walls. The mouth of the plenum is designed to house the throttle body and the runner outlets mount on to carburetor boots.

Chapter 4

CFD ANALYSIS

4.1 Calculations

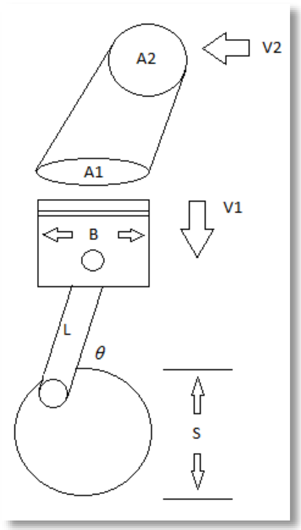


Figure 4.1 Piston Diagram

Where*:*

B = bore (mm)

S = stroke (mm)

L = connecting rod length (mm)

θ = crank angle (rad)

V1 = Velocity of the piston (m/s)

A = Area (m^2)

Before getting to use any software, calculations had to be done to find out the values of the desired inputs that will be required by the software and the theoretical values to compare analysis with. In this case it was required to know the velocity of the air at the intake port along with the volumetric flow rate at that velocity and the theoretical pressure drop. To be able to find the velocity at the port one must first find the maximum velocity of the piston. The volumetric velocity is then calculated using the velocity found in the previous calculation multiplied by the area of the bore. The port velocity and volumetric velocity can be found by a simple mass flow equations using the results from the above calculations. Please refer to the appendix for a detailed explanation of the calculations.

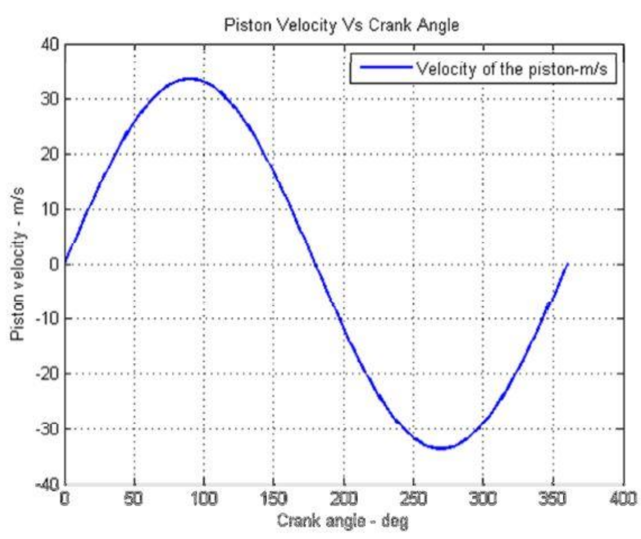


Figure 4.2 Piston Velocity profile

A mathematical model of the piston was set-up in Matlab to see the behavior of air at the intake port. This gave the values of the desired inputs that will be required by the software and the theoretical values to compare analysis with

At the intake port @ 19,000 rpm with a boost of 10 psi

- Max Velocity V = 108.5 m/s
- Max Volumetric Flow Rate Q = 0.0621 m^3/s (131.6 scfm)
- Max Pressure Drop P = 10.8 kPa

4.2 Flow Bench Test Results

To have something to compare the data to, from the final intake design, flow bench experiments were conducted on two of the previous intake manifolds.

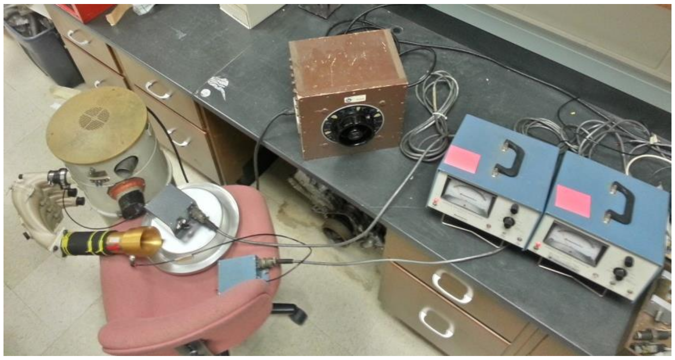


Figure 4.3 Flow Bench trials on F06 Intake Manifold

The intake manifold was set up on a flow bench as shown in Figure 4.3 to tabulate the pressure drop across the manifold at various voltages. The flow bench set up is basically a pump that pulls vacuum. Two pressure sensor probes are connected, one on the nozzle connected to the mouth of the intake plenum and other on the pump. The pressure drop readings are taken on an electronic manometer. Here pressure gets tabulated against voltage. So as to compare the four intake runners the equation below gets used.

$$\Delta P = HQ^2$$

Where ΔP is the pressure drop in kPa, H is the discharge coefficient and Q is the volumetric flow rate in scfm. Modification of the H is required so as to find the best fit line for the data. After all the plots are completed comparing the H values is critical to see how much different the flow for each runner is.

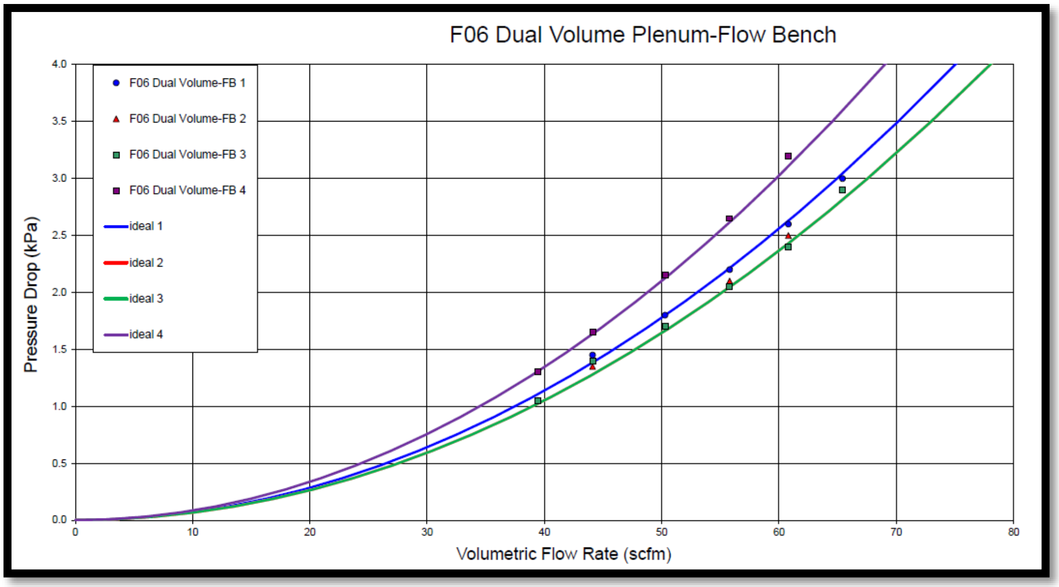


Figure 4.4 Flow Bench results on F06 Dual Volume Plenum Intake

As it can be seen from figure 4.5 the 4 runners on the F06 Dual Volume Intake were not receiving uniform flow of air. A flow difference of 3.5% was seen between runners 1 & 2 and 1 & 3. Runners 2 and 3 were receiving 3.5% less air than runner 1 and a flow difference of 8% was seen between runners 1 & 4. Runner 4 was receiving 8% less air than runner 1.

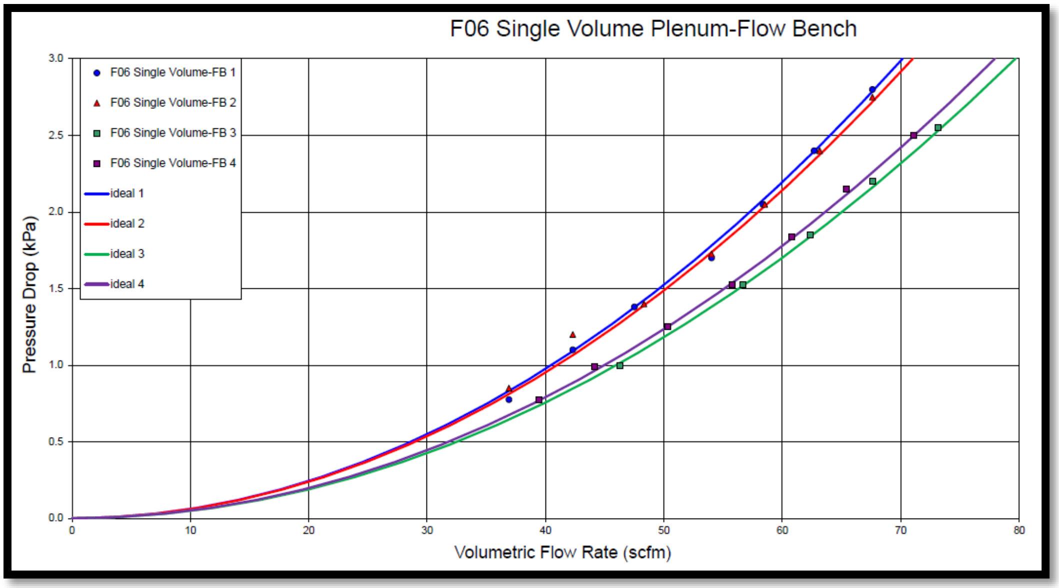


Figure 4.5 Flow Bench results on F06 Single Volume Plenum Intake

On the F06 Single Volume Intake, the flow looked more uniform than the dual volume intake, although there was considerable flow difference between the runners. 3.65% difference between runner 1 & 2, 11.9% difference runner 1 & 3 and 11.1% difference between runner 1 & 4. Runners 3 and 4 received more air than 1 and 2 because the mouth of the plenum was tilted towards runners 3 and 4.

The flow bench results were compared with each other by interpolating one graph over the other. As it can be seen in the figure below, the single volume plenum had a lesser pressure drop compared to the dual volume plenum and the flow looked more uniform.

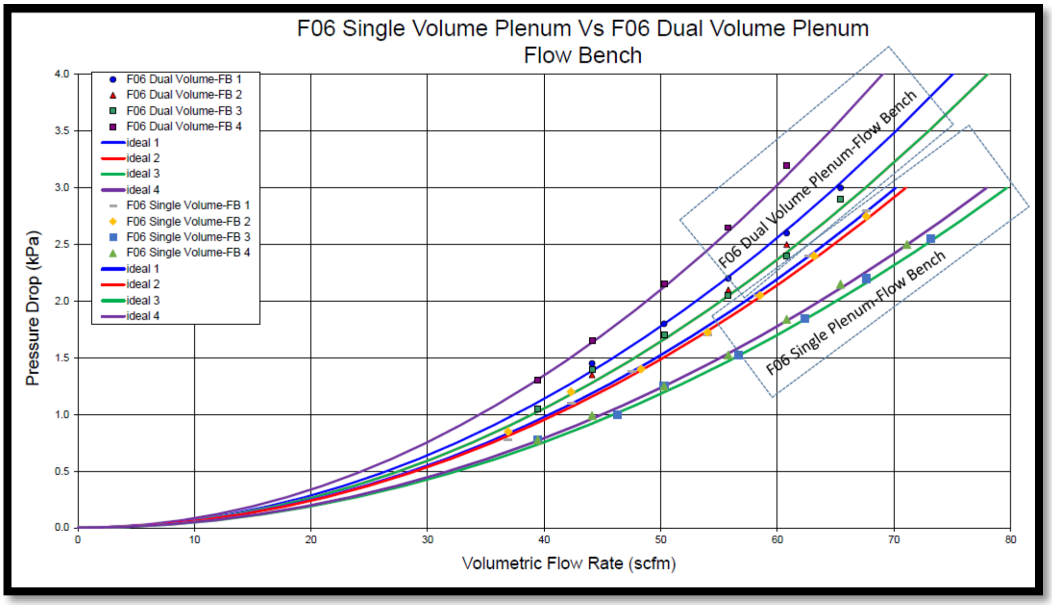


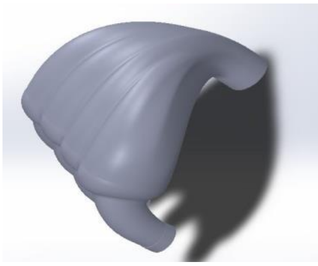
Figure 4.6 Flow Bench results comparison of F06 Single Volume Plenum Intake Vs F06 Dual Volume Plenum Intake

A volumetric flow of 60.8 scfm at a pressure drop of 2.4 kpa was observed on runner 3 on the dual volume plenum and 62.4 scfm at 1.84 kpa was observed on the single plenum volume. For a given volumetric flow rate of about 60 scfm pressure drop dropped by 23.33% on the single volume plenum intake. Although this was a significant improvement the runners on the single volume plenum did not have uniform flow through all the 4 runners.

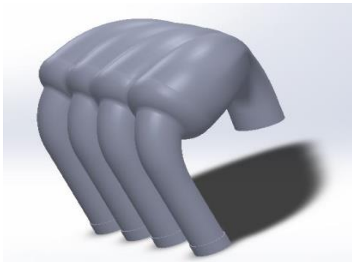
4.3 CFD on 2015 Single Volume

The 3D assembly of the intake manifold created in SolidWorks 2015 was exported as a single surface in the form an IGES file. The initial designs were modeled as one section and features that didn't take part in the internal flow were suppressed.

Meshing and CFD analysis were done using Star CCM+. Star CCM+ is a comprehensive engineering simulation package for solving problems involving flow (of fluids or solids), heat transfer and stress.



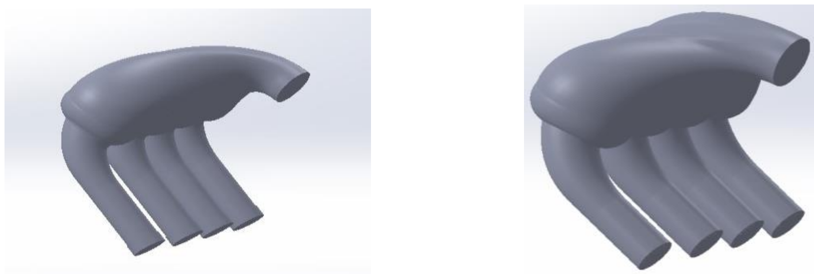
(a)



(b)

Figure 4.7 (a) is Design1 and (b) is Design2

Design 1 had a plenum volume of 1800 cc and Design 2 had a plenum volume of 1400 cc. The runners 3 and 4 in these two designs are oriented towards the mouth of the plenum.



(a) *(b)*

Figure 4.8 (a) is Design 3 and (b) is Final Design

Design 3 has a plenum volume of 1030 cc and all the 4 runners oriented towards the mouth of the plenum. The final design has a volume of 1030 cc and the mouth of the plenum located centrally on the engine head and all the runners oriented towards the mouth. The carburetor boot length and the intake port length were added to the runner length to observe the pressure drop accurately across the manifold.

Although the same thought process carried out during the flow bench experiments applies to analyzing the results from CFD, a different approach was taken to set up the simulation in Star CCM+. Instead of applying different pressure drops on each runner, a velocity field was implemented. In other words instead of applying a pressure drop ranging from 1000 Pa to 10000 Pa a velocity field ranging from Mach 0.1 to Mach 0.4 was applied to the runners. So as to be able to apply a velocity field in Star-CCM appropriate models need to be picked as shown below.

Under Continua in the tool bar:

Click Models:

Two Layer All y + Wall Treatment

Realizable K- Epsilon two Layer

K- Epsilon Turbulent

Reynolds- Average Navier- Stokes

Turbulent

Segregated Fluid Temperature

Ideal Gas

Segregated Flow

Gas

Steady

Gradient

Three Dimensional

When importing the surface mesh into Starr-CCM there will be a menu that will ask how the boundaries want to be set up. In this case they were set up as shown below:

New Region

One Boundary for all Faces

One Region for all Bodies

Very Fine Tessellations

Once all the models were selected the meshing models were then chosen which are found under continua in the tool bar. Total of 145947 domains were created.

The models that were selected are shown below:

Prism Layer Mesher

Polyhedral Mesher

Surface Remesher

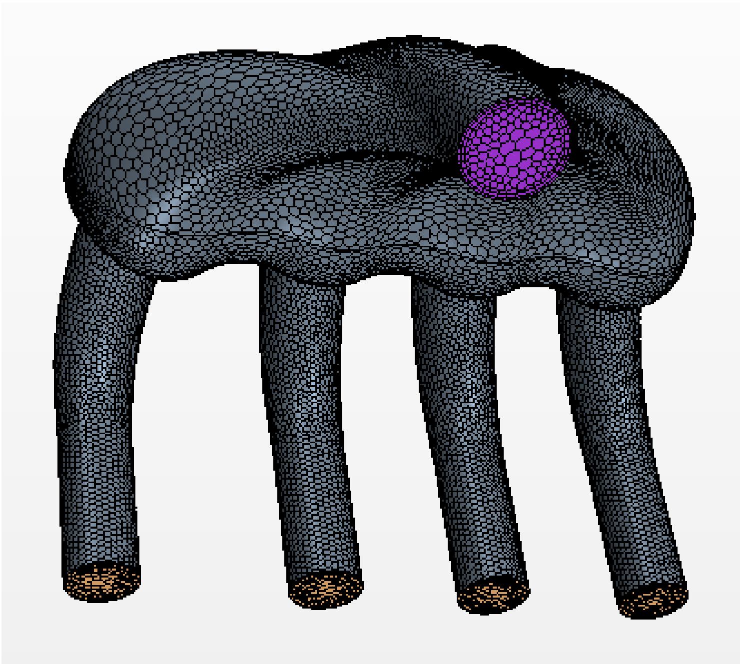


Figure 4.9 Volume Mesh on Star CCM+

Once these models were selected, under reference values in the toolbar the base size picked was .007 m. These are the models in which

Star-CCM uses so as to solve for the solution. A detailed explanation on the physics and meshing models selected are given in the appendix. One simulation run on each of the 4 designs had 4 iterations to be done for Mach 0.1 to Mach 0.4 on each runner making it a total of 16 iterations on each design.

4.3.1 CFD Results

The data that is wanted to be found from the CFD simulation is the volumetric velocity, mass flow rate and pressure drop from each outlet. So as to be able to get these data from Star-CCM, reports have to be created and to help visualize what is going on inside the intake, different scenes need to be set up and hardcopies need to be saved. Once the simulation has run, the data can be retrieved and put into an excel spreadsheet and the discharge coefficient can be used to compare flow across each runner.

Table 4.1 CFD results for Volumetric Flow Rate (scfm) on the 2015 Single Volume Plenum Intake

	Runner			
Mach Number	1	2	3	4
	Volumetric Flow (scfm)			
0.1	39.83	39.83	39.83	39.83
0.2	76.28	76.28	76.28	76.28
0.3	110.39	110.39	110.39	110.39
0.4	142.18	142.18	142.18	142.18

Table 4.2 CFD results for Pressure Drop (kPa) on the 2015 Single Volume Plenum Intake

	Runner			
Mach Number	1	2	3	4
	Pressure Drop (kPa)			
0.1	0.71	0.72	0.71	0.70
0.2	2.55	2.60	2.56	2.54
0.3	5.23	5.35	5.26	5.24
0.4	8.53	8.74	8.60	8.74

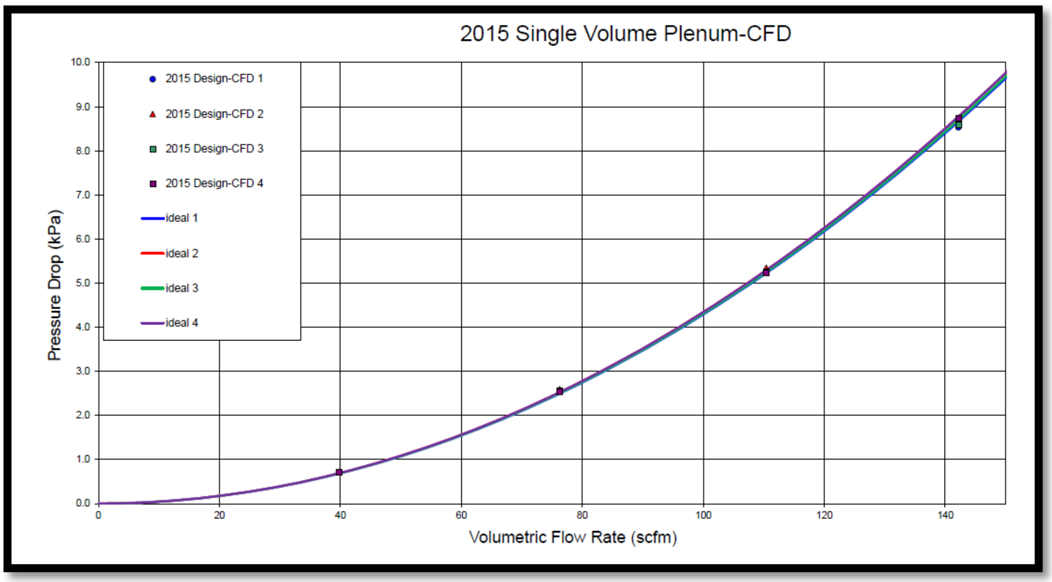


Figure 4.10 CFD results on 2015 Single Volume Plenum Intake

The graph shows that the flow is practically the same on all the 4 runners even at high Mach numbers of 0.4.

Table 4.3 Discharge Coefficients on Single Volume Plenum

Runners	Discharge Coefficient
1	48.3
2	48
3	48.2
4	48

As it can be seen the H values differ only by 0.1 which is good because it means the runners are flowing relatively the same amount of air.

Table 4.4 Percent difference between Discharge Coefficients

	Percent difference in Discharge Coefficients %			
Runners	1	2	3	4
1	0.0	0.62	0.2	0.62
2	0.62	0	0.41	0
3	0.21	0.41	0	0.41
4	0.62	0	0.41	0

Comparing the values found CFD to the theoretical values from it can be seen that the values are more than what is required by the engine. The reasoning for why the calculated value is higher than the one found by the Star-CCM+ is that the simulation takes in account a higher Mach number. The fact that the values are higher than the theoretical ones is good news because it means that the simulation is providing good data and that the intake is capable of providing more than air to all the runners.

When we compare the volumetric flow rate of the 2015 Single Volume Plenum Manifold to the F06 Single Volume Plenum Manifold, the flow looks more uniform throughout all the 4 runners. At 110.3 scfm of volumetric flow rate, the pressure drop on runner 3 on the F06 Single Volume Plenum Intake is 5.76 kPa. At 110.4 scfm, the pressure drop on the runner 3 on 2015 Single Volume Plenum Intake is 5.24 kPa. That is close to 9% reduction in the pressure drop for a given flow rate of about 110 cfm. This is a major improvement from the old design.

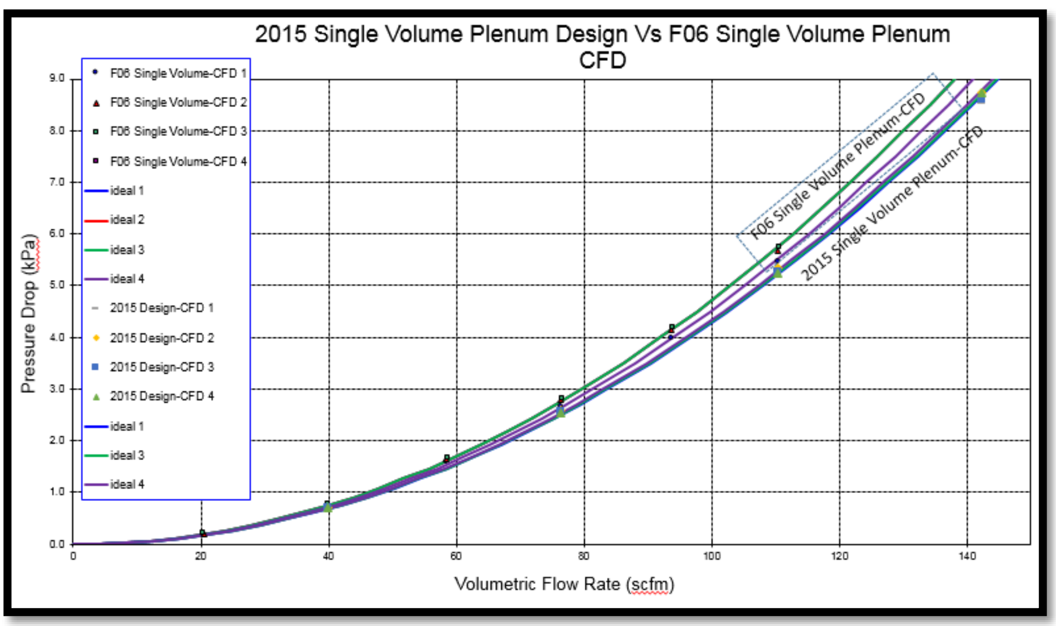


Figure 4.11 CFD results comparison of 2015 Single Volume Plenum Vs F06 Single Volume Plenum

4.3.2 Pressure and Velocity Contours

Scalar and vector scenes can be set up in Star CCM+ to observe the physical characteristics like pressure distribution and velocity profile of air flowing through the entire intake manifold. The contours can either viewed as a Smooth profile as used in pressure contours or in Glyph contour with arrows that can used to view velocity profile.

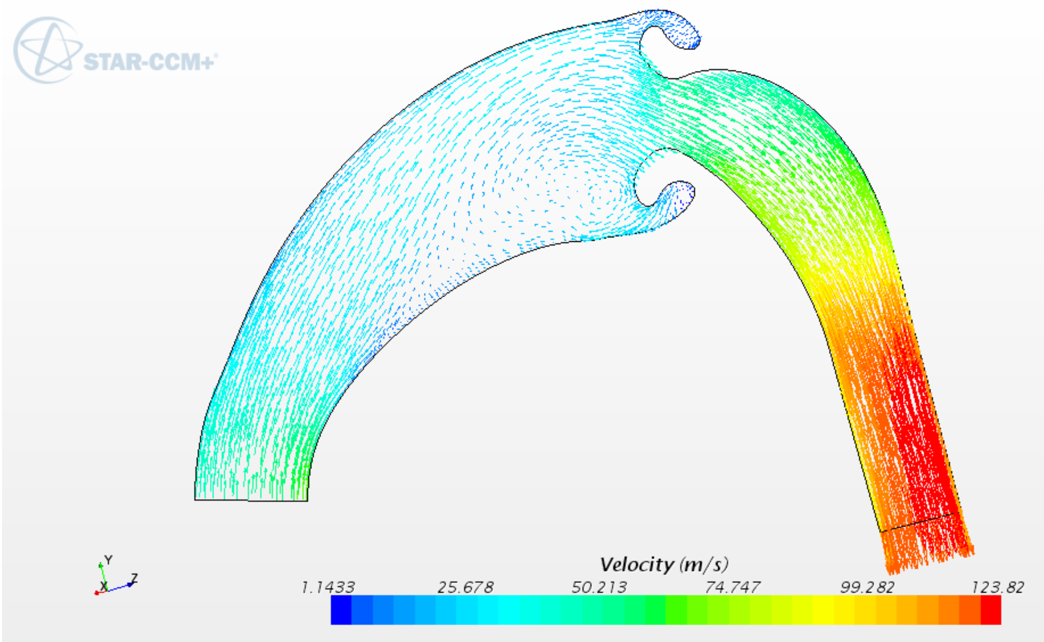


Figure 4.12 Velocity Profile in Runner 1 at Mach0.4

In figure 4.12 it can be seen that the flow throughout the plenum of the intake has minimal amount of separation. There is one spot near the middle of the plenum but it is negligible since that area does not affect the bell mouth. It can be clearly seen the air that remains un-separated from the inlet continues directly to the bell mouth so the area below is not contributing nor is it hurting the airflow into the runner. That vortex is created because of the elliptical section at the bottom of the bellmouth which is acting as a stagnation point in that area. This can be reduced by decreasing the diameter of the bellmouth and having a smoother entry at the bottom of the bell mouth. But this would come at the cost of compromising on the area available for air to enter into the runner from the plenum volume. Because we are dealing with a high speed, high power engine the intake valves are practically open all the time and the runner would need an infinite amount of air available from the plenum. For this reason, it is better to have a wider bell mouth with a bigger area available for the runner.

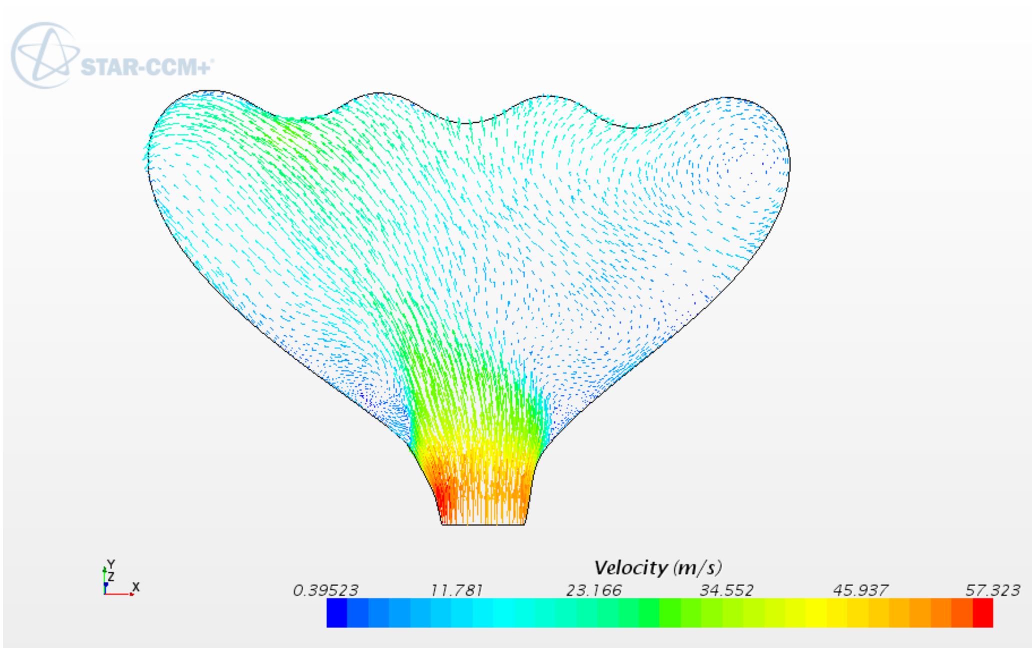


Figure 4.13 Velocity Contour in the Plenum for Runner1 at Mach0.4

As it can be seen from the velocity profile in the plenum volume during intake stroke in cylinder 1 there are two vortices formed; one near the mouth of the runner 4 and other near the left side of the mouth of the plenum. There is an abrupt turn in the streamline flow which gives rise to this vortex. The space restriction between the mouth and compressor prohibits a longer and a smoother transition from the mouth towards the plenum. There is no change in the velocity near the outlet of the runner or any flow separation seen on the runner wall because of these vortices.

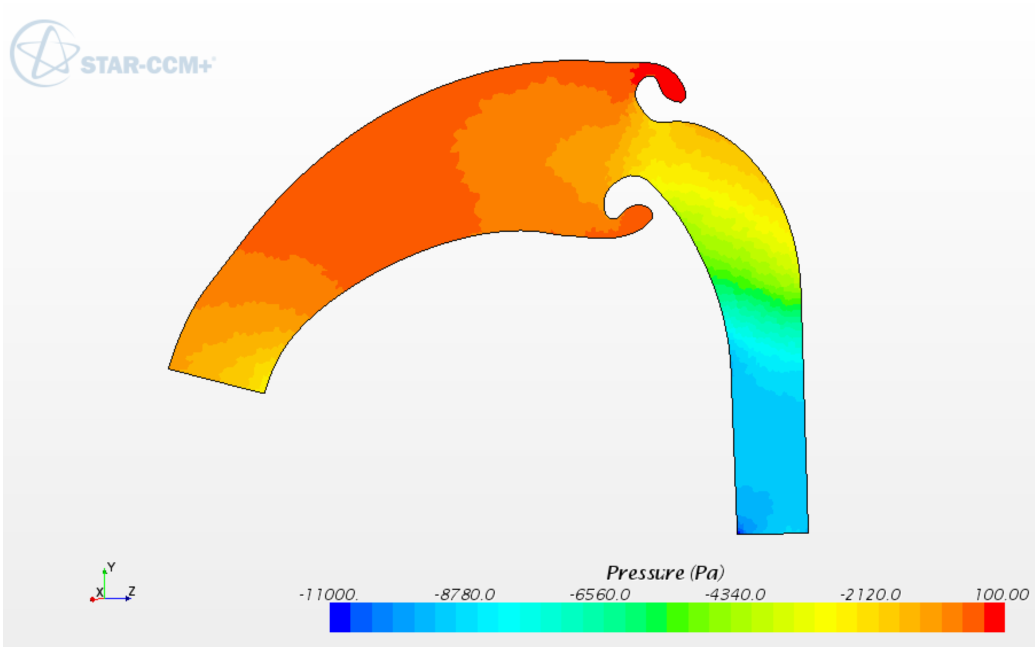


Figure 4.14 Pressure Contour in Runner 1 at Mach0.4

The figure above shows the pressure drops throughout the plenum. As it can be seen there is a very even pressure drop from the inlet towards the bell mouth. There is one spot near the center of the plenum which is very low meaning that that area is not contributing to the airflow. As the air enters the runner the pressure gradually increases and reaches near maximum. This means that the oval design runner is preventing a sudden large pressure drop which is what is wanted because what is being sought is a large volumetric flow for a minimal pressure drop.

Chapter 5

INTAKE MANUFACTURING

5.1 Rapid Prototype Printing

Rapid prototyping is a group of techniques used to quickly fabricate a scale model of a physical part or assembly using three-dimensional designs. Construction of the part or assembly is usually done using 3D printing or additive layer manufacturing technology.

The intake will be rapid prototype printed out of Polyphenylsulfone **(**PPSF- material spec sheet available in appendix) which has the highest heat and chemical resistance out of any polymer based RPT technology available. Plenum was also now split into two sections for manufacturability. Due to the nature of rapid prototype technology, inside cavities need to be accessible to clear out the breakaway printing material.

5.2 Assembly

The upper and lower half of the plenum are joint together by a grove and slot type geometry which get bonded together using epoxy. A tolerance of 1/5000 of an inch is given to the assembly to accommodate for the epoxy. In addition, 8 number 10-32 brass heat inserts will be soldered on the bottom shell and head socket cap screws reinforce the upper shell on to bottom shell to assure pressure will not deteriorate or tug apart the bond through cyclic loading. Furthermore two bungs are added on top of the manifold to support a tapped 3/8NPT intake air temperature sensor and the MAP sensor. The runners have bungs to support the fuel injector and the bottom shell houses the mounts for the fuel rail. The fuel rail will be assembled on to bottom shell using 10-32 brass heat inserts.

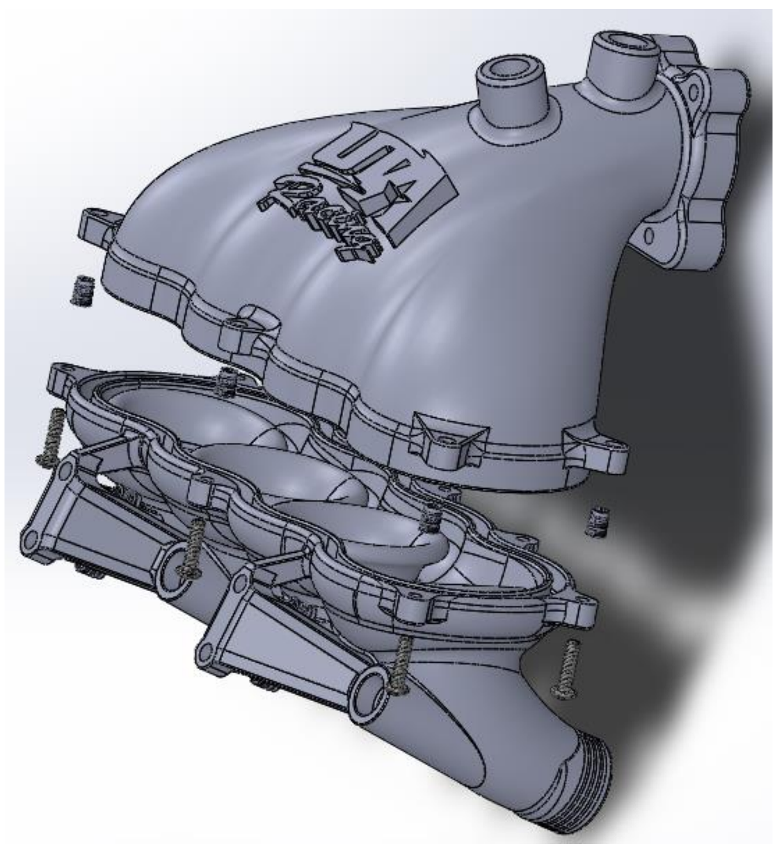


Figure 5.1 2015 Single Volume Plenum Intake Manifold Assembly

The throttle body mounts to the mouth of the plenum using 4 number quarter inch bolt-nut. The throttle will also host an O-ring to prevent any air leaks near the mouth. The intake will mount on the engine head through rubber carburetor boots securing this firmly and minimize any vibrations during engine operation.

Chapter 6

CONCLUSIONS AND FUTURE WORK

6.1 Conclusions

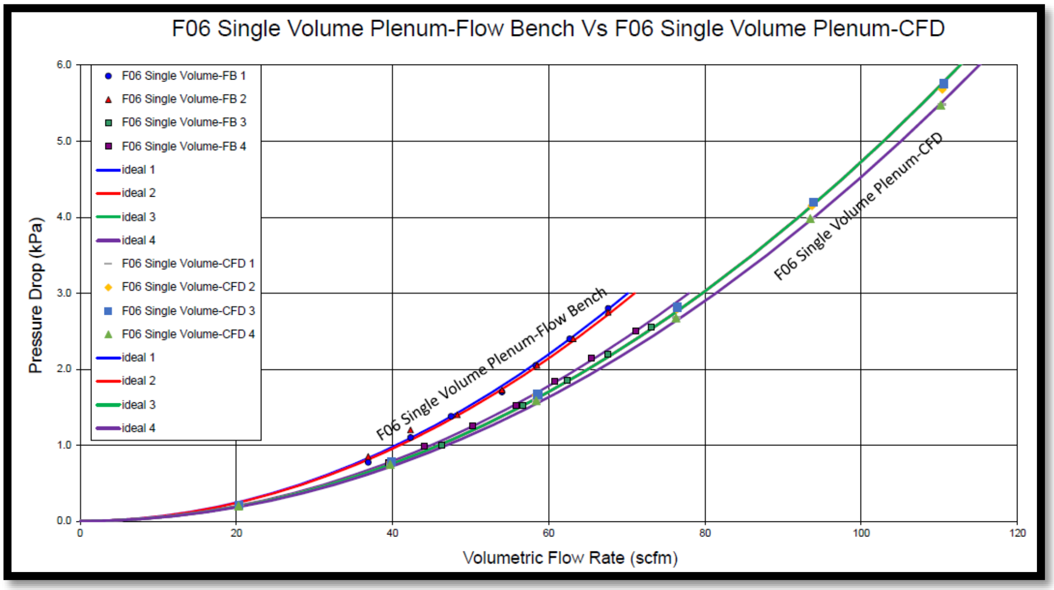


Figure 6.1 Results comparison of F06 Single Volume Plenum-CFD Vs F06 Single Volume Plenum-Flow Bench

Looking at the data from the simulation it can be seen that the differences between volumetric flow rate and pressure drop among the different intake runners are very close to one another and may in fact be negligible. This means that the bellmouth is a very efficient design without any significant flow separations. Any flow separation that happens along the plenum wall or in the plenum volume does not affect the flow through the runners.

By comparing the CFD results with the flow bench results on the F06 Single Volume Plenum Intake Manifold as shown in Figure 6.1, we can predict the performance of the new design. F06 single volume plenum intake manifold had a pressure drop of 1.84 kPa at 60.8 scfm of flow when measured on the flow bench on runner 4. The CFD analysis on runner 4 showed a pressure drop of 1.69 kPa at 60.8 scfm of flow. There was a pressure loss of about 8%. The new design is aimed to follow a similar flow trend but with reduction in the pressure loss and provide uniform flow of air to all the runners.

6.2 Future Work

. Looking at the percent difference between the discharge coefficients which is 0.62% means that when compared to previous designs, is a major improvement and that there is very little else that can be done to improve that number. More research and designing should take place to reduce vortices generated inside the plenum volume near the bellmouth without compromising on the area available for the flow of air. Flow-wiz tests should be set up on the manifold to observe the fluid dynamics inside the plenum accurately.

Appendix A

HONDA CBR250RR ENGINE SPECIFICATIONS

Displacement	:	249.00 ccm (15.19 cubic inches)
Engine type	:	In-line four, four-stroke
Power	:	45.00 HP (32.8 kW)) @ 14500 RPM
Torque	:	2.40 Nm @ 11500 RPM
Top speed	:	185.0 km/h (115.0 mph)
Max RPM	:	19000
Compression	:	11.5:1
Bore x stroke	:	48.5 x 33.8 mm (1.9 x 1.3 inches)
Valves per cylinder	:	4
Fuel system	:	Carburetor, (replaced by F.I)
Fuel control	:	DOHC
Cooling system	:	Liquid
Gearbox	:	6-speed manual
Final drive	:	Chain
Clutch	:	Wet Multi-plate

Appendix B

PISTON VELOCITY CALCULATIONS

Maximum Velocity of the piston is given by the equation:

$$Vp = \left[\frac{S}{2}\sin(\theta) + \frac{\left[\frac{S}{2}\right]^2 \sin\theta\cos\theta}{\sqrt{L^2 - \left[\frac{S}{2}\right]^2 \sin^2\theta}} \right]\omega$$

Volumetric Flow Rate at the port is given by:

$$Q = AV$$

Where

B = bore (m)

S = stroke (m)

L = connecting rod length (m)

θ = crank angle (rad)

ω = engine speed (rad/sec)

V_p = Velocity (m/s)

A = Area (m^2) = $\pi * B^2 / 4$

Mass flow equation to calculate the flow rate in the intake port:

$$(V_p max)(A_p) = (V_{ip})(A_{ip})$$

Initial Conditions:

ρ = 1.1455;	Density of air at standard conditions (kg/m^3)
γ = 1.4;	Ratio of Specific Heats of gas in Isentropic Expansion
S = 0.0338;	Stroke (m)
B = 0.0485;	Bore (m)
L = 0.0998;	Connecting Rod Length (m)
Dip = 0.027;	Diameter of the intake port (m)
ω = (19000/60)*2*pi;	Maximum Speed of the engine (rad/s)
θ = 0:360;	Crank angle (degree)
Ap = (pi*B^2)/4;	Area of the piston top surface (m^2)
Aip = (pi*Dip^2)/4;	Area of the intake port (m^2)
$V_p$max = 33.6255 m/s;	Maximum velocity of the piston

To calculate the Turbo Boost:

Pbst = 10;	Boost desired (psi)
Patm = 14.7;	Atmospheric pressure - gauge (psi)
Pr = (Patm+Pbst)/Patm;	Pressure ratio
Pr = 1.6803	

To Calculate the Maximum Velocity and Volumetric Velocity of air at the Intake Port:

Vip = Ap*Vpmax/Aip; Velocity at the intake port (m/s)

Qip = Vip*Aip; Volumetric Velocity at the intake port (m^3/s)

Pna = (Ro*Qip^2)/(2*Aip^2); Max Pressure at the Intake Port (Pa)

Pb = Pna * 1.6; Max Pressure at the Intake Port (Pa)

Vip = 108.4988 m/s

Qip = 0.0621 m^3/s

Pna = 6.7424 kPa;

Pb = 10.7 kPa

Appendix C

3D MODELS

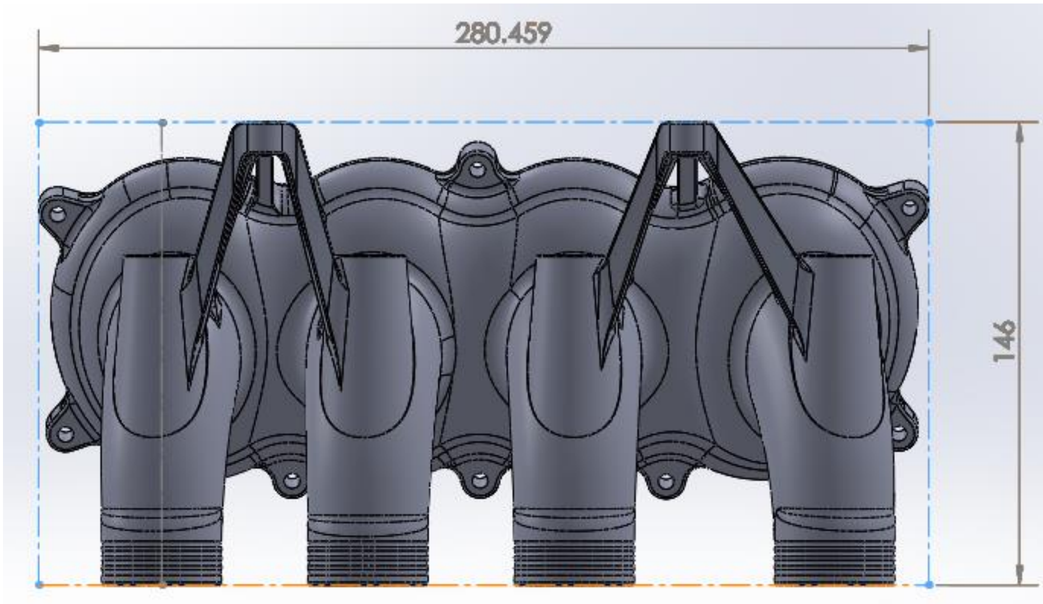


Figure C.1 Overall Dimensions 1 Bottom Shell

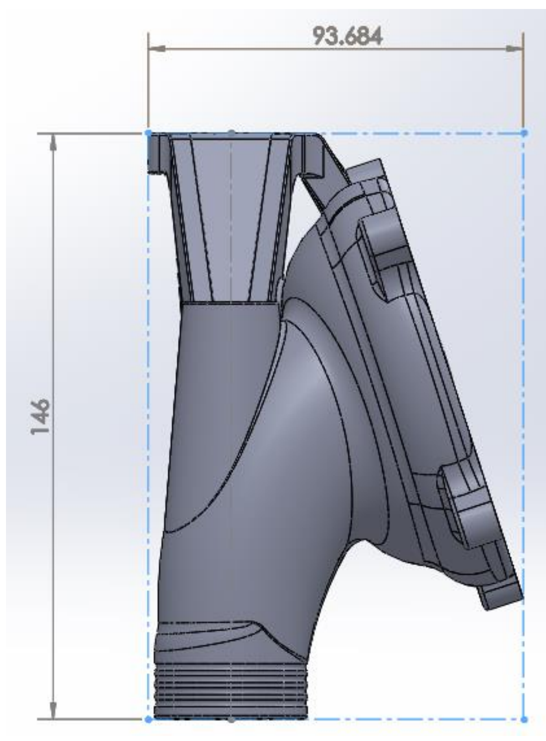


Figure C.2 Overall Dimensions 2 Bottom Shell

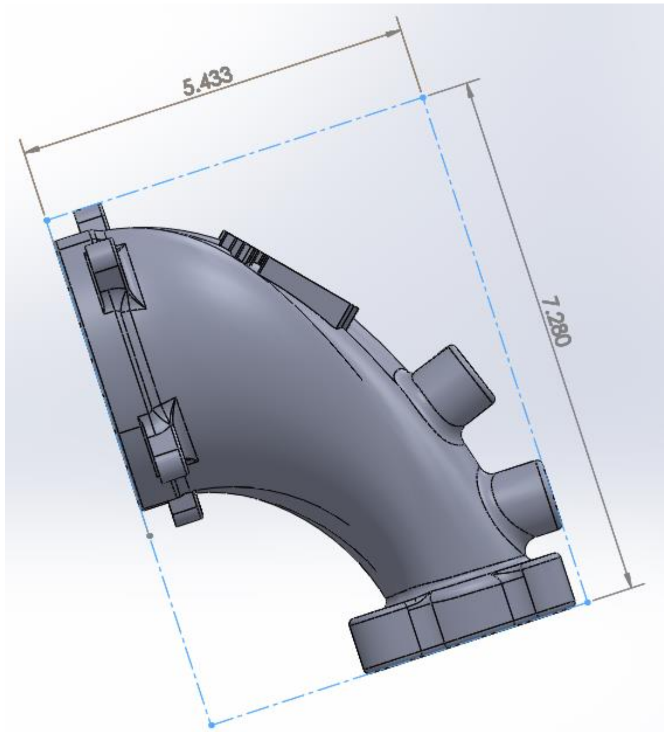


Figure C.3 Overall Dimensions 1 Top Shell

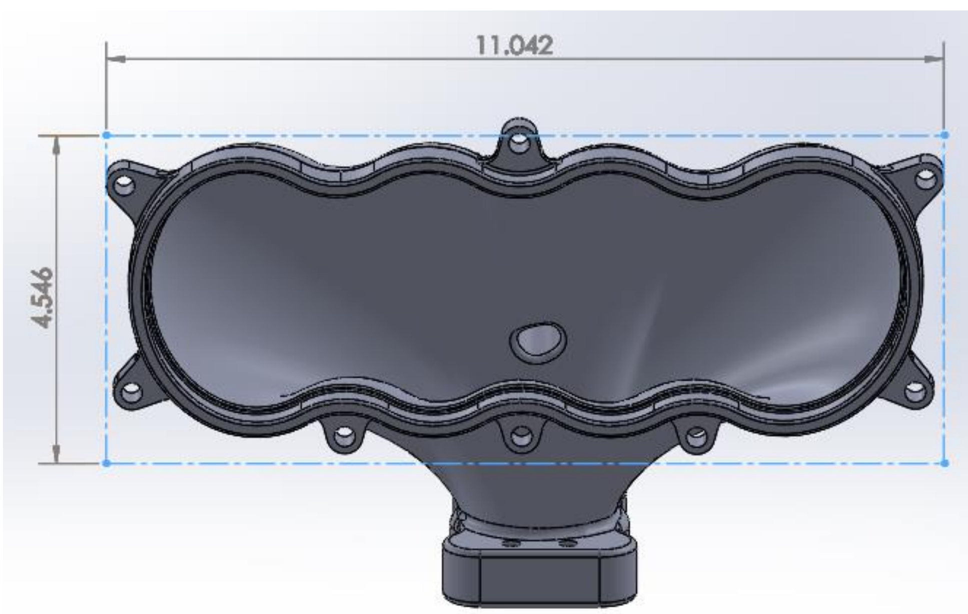


Figure C.4 Overall Dimensions 2 Top Shell

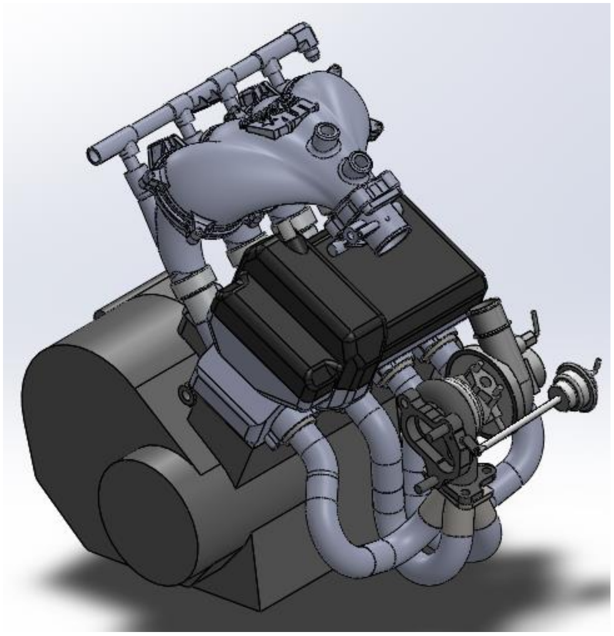


Figure C.5 Honda CBR250RR Engine Assembly

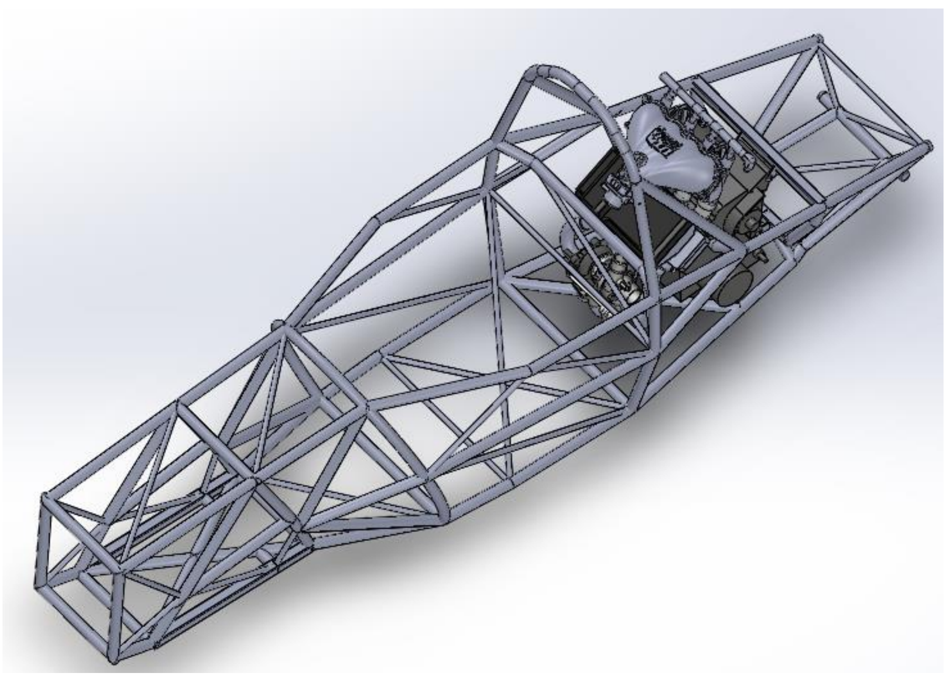


Figure C.6 F06 Chassis with engine assembly

Appendix D

PHYSICS MODELS USED IN CFD ANALYSIS

Turbulence Model:

It is widely acknowledged that turbulence models are inexact representations of the physical phenomena being modeled, and no single turbulence model is the best for every flow simulation. Consequently, it is necessary to provide a suite of models that reflect the current state of the art.

To obtain the Reynolds-Averaged Navier-Stokes (RANS) equations, the Navier-Stokes equations for the instantaneous velocity and pressure fields are decomposed into a mean value and a fluctuating component. The averaging process may be thought of as time averaging for steady-state situations and ensemble averaging for repeatable transient situations. The resulting equations for the mean quantities are essentially identical to the original equations, except that an additional term now appears in the momentum transport equation. This additional term is a tensor quantity, known as the Reynolds stress tensor.

Realizable K-Epsilon Turbulence Model:

A K-Epsilon turbulence model is a two-equation model in which transport equations are solved for the turbulent kinetic energy and its dissipation rate. K-Epsilon turbulence models provide a good compromise between robustness, computational cost and accuracy. They are generally well suited to industrial-type applications that contain complex recirculation, with or without heat transfer.

A Realizable K-Epsilon turbulence model is substantially better than the standard K-Epsilon model for many applications, and can generally be relied upon to give answers that are at least as accurate. This model contains a new transport equation for the turbulent dissipation rate. Also, a critical coefficient of the model, is expressed as a function of mean flow and turbulence properties, rather than assumed to be constant as in the standard model. This procedure lets the model satisfy certain mathematical constraints on the normal stresses consistent with the physics of turbulence (realizability). The concept of a variable is also consistent with experimental observations in boundary layers.

Two-Layer Approach:

The two-layer approach, is an alternative to the low-Reynolds number approach that allows the K-Epsilon model to be applied in the viscous sublayer. In this approach, the computation is divided into two layers. In the layer next to the wall, the turbulent dissipation rate and the turbulent viscosity are specified as functions of wall distance. The values of dissipation rate specified in the near-wall layer are blended smoothly with the values computed from solving the transport equation far from the wall. The equation for the turbulent kinetic energy is solved in the entire flow. In

STAR-CCM+, the two-layer formulations work with either low-Reynolds number type meshes y+ ~ 1 or wall-function type meshes y+ > 30.

Two layer all y+ Wall Treatment:

The wall functions are a set of semi empirical functions used to satisfy the physics of the flow in the near wall region. Turbulence is affected in many ways by the presence of the wall through the non-slip condition that must be satisfied at the wall. Four areas in the near wall region are defined, the laminar sub - layer, the blending region, the log law region and the outer region. Each region has a different effect on turbulence and a particular care must be taken to the y+ position of the first cell in the boundary layer. The all y+ wall treatment is a hybrid treatment that attempts to emulate the high wall treatment for coarse meshes and the low y+ wall treatment for fine meshes. It is also formulated with the desirable characteristic of producing reasonable answers for meshes of intermediate resolution.

Appendix D

PPSF MATERIAL PROPERTIES

PPSF (polyphenylsulfone) material has the excellent heat and chemical resistance – ideal for aerospace, automotive and medical applications. PPSF parts manufactured are not only mechanically superior, but also dimensionally accurate; to better predict end-product performance. Users can also sterilize PPSF via steam autoclave, EtO sterilization, plasma sterilization, chemical sterilization and radiation. PPSF gives you the ability to manufacture parts direct from digital files that are ideal for conceptual modeling, functional prototyping, manufacturing tools and end-use-parts.

Mechanical Properties[1]	Test Method	English	Metric
Tensile Strength (Type 1, 0.125", 0.2"/min)	ASTM D638	8,000 psi	55 MPa
Tensile Modulus (Type 1, 0.125", 0.2"/min)	ASTM D638	300,000 psi	2,100 MPa
Tensile Elongation (Type 1, 0.125", 0.2"/min)	ASTM D638	3%	3%
Flexural Strength (Method 1, 0.05"/min)	ASTM D790	15,900 psi	110 MPa
Flexural Modulus (Method 1, 0.05"/min)	ASTM D790	320,000 psi	2,200 MPa
IZOD Impact, notched (Method A, 23°C)	ASTM D256	1.1 ft-lb/in	58.7 J/m
IZOD Impact, un-notched (Method A, 23°C)	ASTM D256	3.1 ft-lb/in	165.5 J/m

Thermal Properties[3]	Test Method	English	Metric
Heat Deflection (HDT) @ 264 psi	ASTM D648	372°F	189°C
Glass Transition Temperature (Tg)	DMA (SSYS)	446°F	230°C
Coefficient of Thermal Expansion	ASTM D696	3.1^{-05} in/in/°F	5.5^{-05} mm/mm/°C
Melting Point	- - - - - - - - -	Not Applicable[2]	Not Applicable[2]

Electrical Properties[4]	Test Method	Value Range
Volume Resistivity	ASTM D257	1.5×10^{14} - 5.0×10^{13} ohms
Dielectric Constant	ASTM D150-98	3.2 - 3.0
Dissipation Factor	ASTM D150-98	.0015 - .0011
Dielectric Strength	ASTM D149-09, Method A	290 - 80 V/mil

Figure D.1 PPSF Material Spec Sheet 1

Environmental Resistance[5]	24 hours @ 23°C (73°F)	24 hours @ 100°C (212°F)
Antifreeze (Prestone), 50%	Passed	Passed
Gasoline-Unleaded	Passed	Not tested
Motor Oil 10W-40	Passed	Passed
Power Steering Fluid	Passed	Passed
Transmission Fluid	Passed	Passed
Windshield Washer Fluid, 50%	Passed	Not tested

Other[3]	Test Method	Value
Specific Gravity	ASTM D792	1.28
Flame Classification	UL 94	V-0
Rockwell Hardness	ASTM D785	M86
UL File Number	- - - - - - - - -	E345258

System Availability	Layer Thickness Capability	Support Structure	Available Colors
Fortus 400mc™	0.013 inch (0.330 mm)	Breakaway	Tan
Fortus 900mc™	0.010 inch (0.254 mm)[6]		

Figure D.2 PPSF Material Spec Sheet 2

REFERENCES

[1] Gordon P Blair, “Design and Simulation of Four-Stroke Engines”, Society of Automotive Engineers, 1998, SAE reference R-186,

[2] William Harry Crouse, Donald L. Anglin, “Automotive Mechanics”, McGraw-Hill College, 1993

[3] John Heywood, “Internal Combustion Engine Fundamentals”, McGraw-Hill Education, April, 1988

[4] Corky Bell, “Maximum Boost, Designing, Testing and Installing Turbocharger Systems”, Bentley Publishers, 1997

[5] Wes Ruebman and Travis Butler, “Data Analysis and Selection of IHI and Garrett Turbo Systems”, February 2015

[6] Sagar KC, Pablo Lopez, Robert Lopez, Tobias Overdiek, Sebastian Peters, Luis Ulloa, “CBR 250 Turbo Package-Improvements And Revisions”, August 2012

[7] Chris Craig, “Dynamic Analysis of FSAE Intake Manifolds Using WAVE Engine Simulation Software”, 2006

[8] Jacob Bell, Akula Aditya, Subhash Seshadri, “Honda CBR 250R (MC14E) Engine Rebuild”, August 2014

[9] Jacob Bell, Travis Butler Lauren Middleton, Wes Ruebman, Austin Williams, “Design of 250CC Four Cylinder Racing Engine”, Maverick Performance And Engineering, May 2015

[10] Gordon P Blair, W. Melvin Cahoon, “Best bell, Special Investigation: Design of an Intake Bellmouth”, September 1, 2010

[11] Robert Woods, “Turbo Engine Model”, 16 Jan 2006

[12] Rodi W., “Experience with Two-Layer Models Combining the k-e Model with a One-Equation Model Near the Wall”, 29th Aerospace Sciences Meeting, January 7-10, Reno, NV, AIAA 91-0216, 1991.

[13] Shih T.H., Liou W.W., Shabbir A., Yang Z. and Zhu J., “A New k-Eddy Viscosity Model for High Reynolds Number Turbulent Flows-Model Development and Validation”, NASA TM 106721, 1994

[14] Star CCM+ 10.02.010 User Guide and Tutorials

[15] Ricardo Wave 2015.1 Help System

[16] Stratasys Rapid Prototyping, Material Spec Sheet, PPSF, http://www.stratasys.com/materials/fdm/ppsf-ppsu

BIOGRAPHICAL INFORMATION

The author’s tryst with race car engineering stemmed out from an early interest he had in the sports of F1 racing. The urge to push technology to its limits and to enjoy, even if vicariously, the thrill and excitement of the rush in motorsport racing inspired the author to be a part of UTA FSAE Racing team. After working in a reputed automotive design company in Karnataka, India, for two years, the author came to UT Arlington in 2012 to pursue graduate study in Mechanical Engineering. The author has been a part of the team of enthusiastic individuals working on the Honda CBR250RR engine since summer 2014.

The author would like pursue a career in automotive design and CFD analysis contributing to the advancement of design engineering and to pioneer new technological frontiers that make a positive impact on our lives.