PRESSURE DROP ANALYSIS OF 1.6L CAR AIR INTAKE SYSTEM

MOHAMAD SAFWAN BIN AHMAD LOTHFY

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive.

Signature

Name of Supervisor: Mr. Devarajan Ramasamy

Position: Lecturer

Date: 24 November 2009

STUDENT'S DECLARATION

I hereby declare that the work in this project is my own except for quotations and summaries which have been duly acknowledged. The project has not been accepted for any degree and is not concurently submitted for award of other degree.

Signature

Name: Mohamad Safwan Bin Ahmad Lothfy

ID Number: MH 06039

Date: 24 November 2009

Dedicate to my beloved dad,lovely mom and my honour siblings

ACKNOWLEDGEMENTS

First and foremost, grateful to Allah SWT for making it possible for me to complete this project on time. It is pleasure to acknowledge the help and support of everyone concerned with this project. I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. To my father, mother and brothers who have been so keen in helping and encouraging me during the several months it took me to complete this thesis.

I would like to thank and express my deepest appreciation to my supervisor, Mr. Devarajan Ramasamy for providing me opportunity for his helpful reviews, comments and suggestion during the progression of this project. I am truly grateful for his progressive vision about my training in science, his tolerance of my naïve mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

Finally, my sincere thanks go to all my labmates, members of the staff of the Mechanical Engineering Department, UMP: Mr. Mohd fadzil B. Abdul Rahim and Mr. Ismail. Many special thanks go to member research group for their excellent co-operation, inspirations and supports during this study. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this study. Lastly I would like to thank someone who always make me happy and give her love to me.

ABSTRACT

Today, air intake system and filter play major role in getting good quality air into automobile engine. The intake system has improves the combustion efficiency and also reduces air pollution. This paper focuses on the pressure drop analysis of 1.6L car air intake system. The thesis describes the CFD analysis techniques to predict the pressure drop and identify the critical locations of the components. The air intake system is work as to supply the engine with clean air and correct amount for the required air to burn in the manifold chamber. This research is to analyze the model and pressure drop of Proton Waja intake system. The thesis describes the CFD analysis techniques to predict the pressure drop and identify the critical locations of the components. 3D viscous CFD analysis will carry out for an existing model to understand the flow behavior through the intake system, air filter geometry and filter media. Results obtain from CFD analysis of the existing model show good correlation with experimental data. Based on existing model CFD results, it can show the airflow in the intake system and the pressure drop will see by pressure visualization. The time and cost are reducing by using 3D CFD analysis for air intake system in automobile industry.

ABSTRAK

Dewasa ini, sistem pengambilan udara dan penapisan memainkan peranan penting dalam mendapatkan kualiti udara yang memasuki injin automobil. Sistem ini meningkat kecekapan pembakaran dan mengurangkan pencemaran udara. Projek ini memberi fokus kepada analisis tekanan yang jatuh dalam sistem pengambilan udara bagi kereta 1.6 liter. System ini juga berfungsi membekalkan udara bersih dan jumlah udara yang diperlukan untuk pembakaran di dalam kebuk pancarongga. Penyelidikan ini bertujuan mengkaji model dan tekanan yang jatuh untuk sistem pengambilan udara bagi kereta Proton Waja. Analisis CFD dianalisakan ke atas model bagi mamahami corak sistem pengaliran udara melalui sysem pengambilan udara luar, geometri penapisan udara dan media penapisan. Keputusan yang diperolehi dari analisa model CFD menunjukkan bahawa corak sistem ini mempunyai perkaitan yang baik. Daripada keputusan yang dikeluarkan oleh CFD, ia menunjukan pengaliran udara dalam sistem ini dan melihat bagaimana tekanan jatuh melalui visual takanan. Dengan menggunakan analisis dari CFD, ia dapat mengurangkan kos dan masa untuk mengkaji sisten pengambilan udara di dalam industri automotif.

TABLE OF CONTENTS

List of Tables

LIST OF FIGURES

LIST OF SYMBOLS

μ	Dynamic viscosity
V	Fluid velocity
ρ	Density
υ	Kinematic viscosity
m	Molar mass
$\dot{m}$	Mass flow rate
A_c	Area of the nozzle
A_s	Area of the wall
f	Friction factor
K_L	Loss coefficient
h_L	Head loss
Re	Reynolds number
g	Gravity
D	Pipe diameter
Δp	Pressure loss

List of Abbreviations

FYP	Final year project
vs	Versus
3-D	Three Dimension

CHAPTER 1

INTRODUCTION

1.1 PROJECT BACKGROUND

This thesis focuses on the intake system pressure drop analysis using CFD analysis results and experiment. In the end of this project, the pressure drop of air flow cross the air intake can be analyzed. The engine of a car needs air for the combustion process in the cylinders. Air intake system and filter play major role in getting good quality air into automobile engine. It improves the combustion efficiency and also reduces air pollution. For this thesis, the Proton Waja 1.6 Air intake system has been choosing to analyze. The main function of an air intake system is to supply the engine with clean air and correct amount for the required air to burn in the manifold chamber. Air enters the filter through dirty pipe and inlet side plenum, which guides the flow uniformly through the filter media. Optimum utilization of filter can significantly reduce the cost of filter replacements frequently and keep the filter in use for longer time. To optimize intake system and filter, thorough understanding of flows and pressure drop through the system is essential. Computational Fluid Dynamics (CFD) is considered to be the most cost effective solution for flow analysis of intake system along with filter media. Air intake systems employ specially-shaped intake tubes designed to straighten airflow as much as possible while looking great in engine compartment. These pipes are typically mandrel-bent, a process that doesn't crimp the pipe diameter at the bend.

1.2 PROJECT PROBLEM STATEMENT

Car air intake systems allow the car to breathe easier creating more horsepower and greater gas mileage. Grabbing a high-performance air intake is the quick and easy route to several benefits, including:

- An instant increase in horsepower
- A noticeable boost to your throttle response
- Improved fuel economy
- A long-life, washable performance air filter
- Specialized engineering that's fine-tuned to your specific vehicle
- Straightforward, simple installation virtually anyone can complete Horsepower increase from a performance air intake

The flow efficiency of the intake system has a direct impact on the power the engine is able to deliver. This project is to analyze the pressure drop of Proton Waja intake system. If the flow in the air intake determined to have less turbulent flow and decrease the wake projection and there is less pressure drop across the intake system it will increase the efficiency of combustion of the air in the intake system. The CFD will be use to analyze the internal flow of air intake and get the initial result. From the analysis, the value of pressure drop in certain rpm of engine power can be determined. The difference speed of air flow based on the lower until maximum rpm of engine will be used.

For performance intake draws in a higher volume of air which may be much cooler, your engine can breathe easier than with a limiting stock system. With combustion chamber filled by cooler, oxygen-rich air, fuel burns at a more efficient mixture. It will get more power out of every drop of fuel when it's combined with the right amount of air. With more air in the chamber, it can also burn more fuel than before. That's how a performance intake puts power at the pedal for reducing air temperatures, balancing fuel mixtures and providing more air for combustion.

1.3 OBJECTIVES

The objectives of this project are:

i. To determine and analyze the pressure drop in the air intake system.
ii. To analyze the model of Waja 1.6 Air Intake System using the CFD
iii. To estimate flow rates of the air intake system across the intake system.
iv. To analyze the air flow affected by the minor losses.

1.4 SCOPES OF STUDY

The scope of this project will comprise the boundaries of project study. The pressure drop analyses of air intake system are wide range of study. Many characteristic should be bound in order to make this project achieve the objectives. First of all, the study of this project is using Bernoulli Equation to determine the pressure drop in calculation. This equation is very useful to identify the velocity, density and pressure of the air flow. Furthermore, this project study using the Solidworks to design the air intake system model of the Proton Waja 1.6L. The CFD will be use to analyze the internal flow of air intake and get the result in difference speed of engine from minimum until maximum. The speed of engine start from 1000 rpm until 7000 rpm will be use in the simulation.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

This chapter presents a review of literature on the efforts relating to the pressure drop analysis of air intake system. It attempts to establish what the factors are affecting the performance of intake system and how this intake system affected the car engine performance. The discussions are focus on the flow management in the intake system as a tool to improve the car engine performance.

2.2 AIR INTAKE SYSTEM

For an engine equipped with a carburetor, this is pretty straightforward. Air comes in the air filter housing, passes through the air filter, into the carburetor where the fuel is mixed with it. Then it passes through the intake manifold and is drawn into the cylinders [1]. The most advanced part of the system was an Air Temperature Sensor in the air intake. It was used to measure the air temperature and, by opening and closing a flap, allow cool air in through the air horn or heated air piped in from around an exhaust manifold [2]. This was to prevent carburetor icing that would cause the car to stall and die out. It also facilitated vaporization of the fuel into the air stream [1].

In a fuel-injected car it's a whole different ball game. Air is drawn in through the air intake. This is usually a long plastic tube going into the air filter housing. The reason the intake tube is long is to get the air moving in a fairly steady, coherent stream. It then passes through the air filter and then through an Air Flow Meter [3].

The intake system of an engine has three main functions. Its first and usually most identifiable function is to provide a method of filtering the air to ensure that the engine receives clean air free of debris. Two other characteristics that are of importance to the engineers designing the intake system are its flow and acoustic performance [5]. The flow efficiency of the intake system has a direct impact on the power the engine is able to deliver. The acoustic performance is important because government regulations dictate the maximum noise level that vehicles can make during a pass-by test. The speed of air generated by the intake system can be a significant contributor to this pass-by noise and separated flow [6]. It may be noted that since the loss pressure from the intake duct towards atmosphere, this paper assumes the inlet is at the intake manifold and air filter duct and the outlet is at atmosphere.

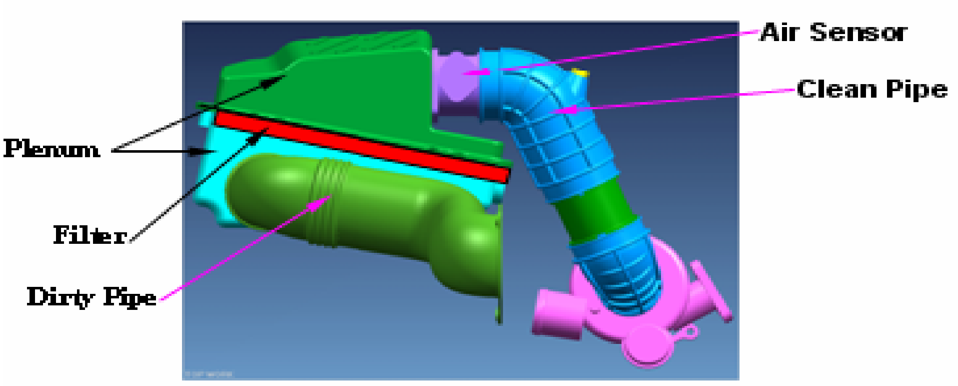


Figure 2.1: An illustration of the air intake system structure

Source: Ravinder Yerram and Nagendra Prasad
Quality Engineering and Software Technologies (QUEST), Bangalore

Air intake systems employ specially-shaped intake tubes designed to straighten airflow as much as possible while looking great in your engine compartment. These pipes are typically mandrel-bent, a process that doesn't crimp the pipe diameter at the bend. Special care is given to locating the intake tube, air box and filter in the position that best fosters maximum performance. The materials used are also selected with optimum engine conditions in mind.

The fundamentals of installing a performance air intake on your vehicle not only is a performance air intake one of the most essential upgrades to your vehicle, it happens to be one of the easiest additions to install. With little more than a common socket set, a couple of screwdrivers and half an hour's time, you can have your new air intake in place and ready to roar [5].

Detailed instructions are included with every intake kit. These instructions go through the simple process of removing your stock intake system (including the tube and air box), and installing the new air intake in just a few minutes to stock mounting positions. No cutting, drilling or other modifications are required [2].

2.3 BERNOULLI EQUATION

The Bernoulli equation is a useful equation as it relates pressure changes to velocity and elevation changes along a streamline. Streamlines are lines drawn in a flow field so that at a given instant they are tangent to the direction of the flow at every point in the flow field. Since they are tangent to the velocity vector at every point in the flow field, there can be no flow of fluid across a streamline [4].

$$\frac{P_1}{\gamma} + \frac{V_1^2}{2g} + Z_1 = \frac{P_2}{\gamma} + \frac{V_2^2}{2g} + Z_2 + h_L \qquad (2.1)$$

The Bernoulli equations give correct results when certain restrictions are applied. These are as follows:

1. Steady flow
2. Incompressible flow
3. Frictionless flow
4. Flow along a streamline.

$$\gamma = \rho g$$

2.3.1 Major Loss

The most pipe or duct system consists of the straight pipe at this point head loss due to viscous effect $Major\ losses, h_{L\ major}$ can determined by equation:

$$Major\ losses, h_{L\ major} = f\frac{\ell}{D}\frac{V^2}{2g} \tag{2.2}$$

Friction factor, f to be determine using Moody chart. Using the Reynolds number and for the plastic surface of the AIS we look the graph curve at the graph smooth line [4]. Because the roughness, ε. For the laminar developed flow, the value of f is simply:

$$f = \frac{64}{Re}$$

2.3.2 Minor Loss

In the intake system we found the system of the pipe more than a straight pipe. These additional components (valves, bends, tees, and the like) add the overall head loss of e system [8]. Such losses are generally termed $Minor\ losses$, with the corresponding head loss denoted $h_{L\ minor}$.

$$Minor\ losses, h_{L\ minor} = K_L\ \frac{V^2}{2g} \tag{2.3}$$

The most common method used to determine these head losses or pressure drops is to specify the loss coefficient, K_L which is defined as:

$$K_L = h_{L\ minor}.\frac{2g}{V^2} = \frac{\Delta p}{\frac{1}{2}\rho V^2} \tag{2.4}$$

Pressure drops, Δp:

$$\Delta p = K_L \frac{1}{2}\rho V^2 \qquad (2.5)$$

2.3.3 Total Pressure

Total Pressure is obtained when the flowing fluid is decelerated to zero speed by a frictionless process [4]. In an incompressible flow the Bernoulli equation can be used to relate the changes in speed and pressure along a streamline for such a flow. Neglecting elevation, then equation becomes:

$$\frac{P_1}{\rho} + \frac{V_1^2}{2} = \frac{P_2}{\rho} + \frac{V_2^2}{2} \qquad (2.6)$$

$$\frac{P_1}{\gamma} + \frac{V_1^2}{2g} = constant$$

If the static pressure P_1 is at a point in the flow where the speed is V_2, then the total pressure P_1, where the stagnation speed, V_1,, is zero, then the equation becomes:

$$P_1 = P_2 + \rho \frac{V_2^2}{2} \qquad (2.7)$$

2.4 MASS AND VOLUME FLOW RATE

The amount of mass flowing through a cross section per unit is called the mass flow rate and its denoted by $\dot{m}$ [8]. The dot over a symbol is used to indicate *time rate change*.

$$\dot{m} = \rho V_{avg} A_c \qquad (2.8)$$

We defined the average velocity V_{avg} average value across the entire cross section of the pipe, where A_c is the area of the cross section normal t the flow direction.
The volume of the fluid flowing through a cross section per unit time is called volume flow rate,$\dot{V}\ or\ Q$.

$$\dot{V} = V_{avg}A_c = VA_c$$

The mass and volume flow rates are related by:

$$\dot{m} = \rho\dot{V}$$

2.5 COMPUTATIONAL FLUID DYNAMICS (CFD)

Air was used as fluid media, which was assumed to be steady and incompressible. High Reynolds number k-ε turbulence model was used in the CFD model. This turbulence model is widely used in industrial applications. The equations of mass and momentum were solved using SIMPLE algorithm to get velocity and pressure in the fluid domain. The assumption of an isotropic turbulence field used in this turbulence model was valid for the current application. The near-wall cell thickness was calculated to satisfy the logarithmic law of the wall boundary. Other fluid properties were taken as constants [11]. One of the most important requirements before a CFD computation can be performed is the available of a suitable grid. Inability to construct a grid quickly and reliably often rules out a CFD analysis. Linear methods such as the Panel Method need only a grid on the body surface (and road). Generation of the grid on the surface of a real vehicle so as to correctly capture the critical flow phenomenon is not a trivial problem. The Computer-Aided Design (CAD) surface definition data created for body panel manufacture in the industry are helpful in generating such grids [12]. The nonlinear CFD methods (Euler, NS) need a body-surrounding spatial grid to solve the partial differential equations. The inner boundary of this grid is the body surface and the outer boundary is the bounding surface of a sufficiently large computational domain around the body

The aim of CFD is to resolve the equations that drive theoretically every kind of flow:

- The continuity equation
- The momentum equations
- The energy equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k}\left(\rho u_k\right) = 0 \tag{2.9}$$

$$\frac{\partial \rho u_i}{\partial y} + \frac{\partial}{\partial x_k}\left(\rho u_i u_k - \tau_{ik}\right) + \frac{\partial P}{\partial x_i} = S_i \tag{2.10}$$

where u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass, E is the total energy per unit mass, Q_H *is a heat source per unit volume,*

$$\frac{\partial(\rho E)}{\partial y} + \frac{\partial}{\partial x_k}\left(\left(\rho E + P\right)u_k + q_k - \tau_{ik} u_i\right) = S_k u_k + Q_H \tag{2.11}$$

is the viscous shear stress tensor and q_i is the diffusive heat flux.

2.5.1 Simulation Benefits

Technical Advantages

- Faster evaluation of new ideas, products and processes
- New insights into your process and performance
- Maximise effectiveness of your manufacturing resources
- Save time and cost, and get better results

Business Advantages

- Reduce risk and increase confidence in technical projects
- Increase customer confidence
- Increase credibility with customers
- Win more business

CHAPTER 3

METHODOLOGY

3.0 INTRODUCTION

In this project, simulation will be conduct by vary the velocity of air based on the speed of engine. Research and approach will be described clearly in flow chart, procedures, dimension measurements, modeling and simulation. The collected data from the simulation will be use for further analysis.

3.1 FLOW CHART

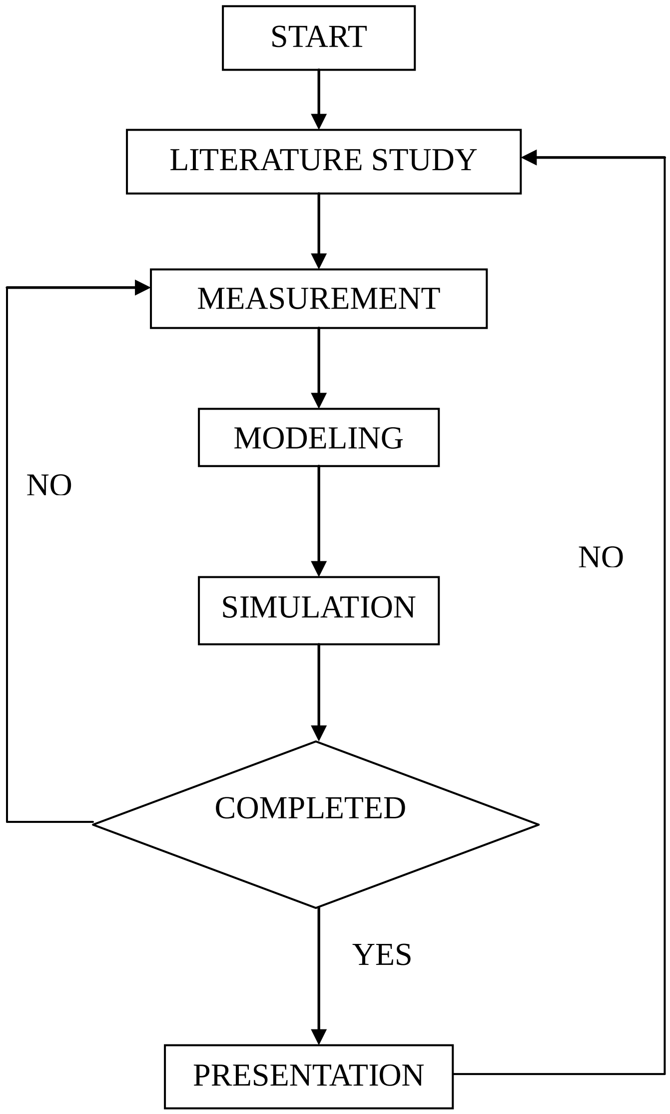


Figure 3.1: (Flow Chart)

3.2 DATA COLLECTING

Figure 3.2: Proton Waja air intake system

The data of dimension for air intake system of Proton Waja was collecting from measuring then modeling the body by SolidWorks software. Data collecting of dimension as accurate as possible is very important for air intake to simulating the model in CFD.

To measure the area, venire caliper had been used.

Inlet cross section area = $0.03756 m^2$

Thickness of plastic = 29.5 mm

Porosity (filter) = 0.85

3.3 STRUCTURAL MODELING – USING SOLIDWORK

After measure all dimension of the air intake, the model has been design by using solidwork software. Every single part of air intake has been drawn and finally all part will assemble.

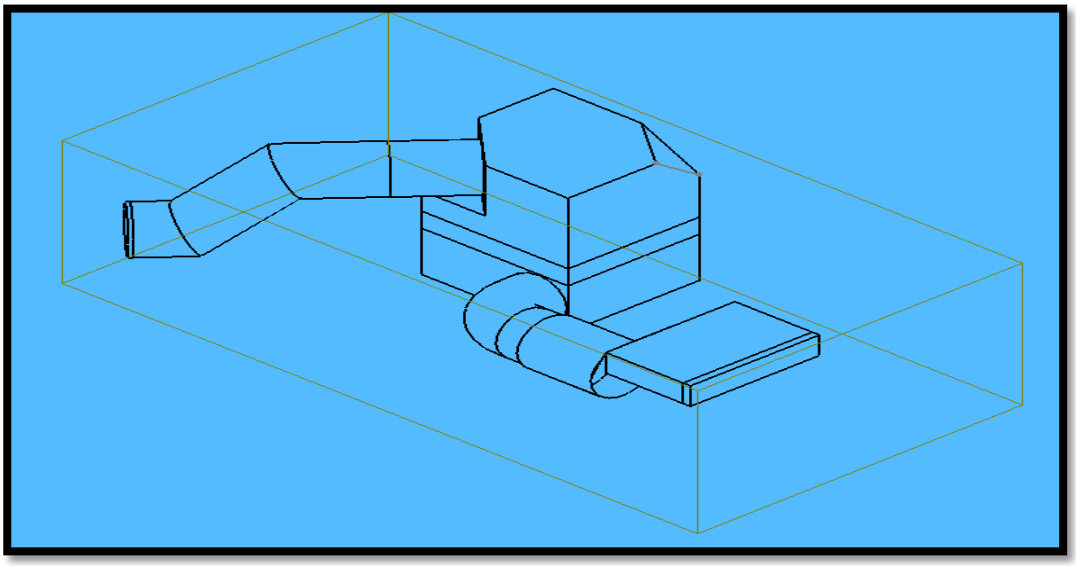


Figure 3.3: 3D of air intake system

The first part: intake pipe

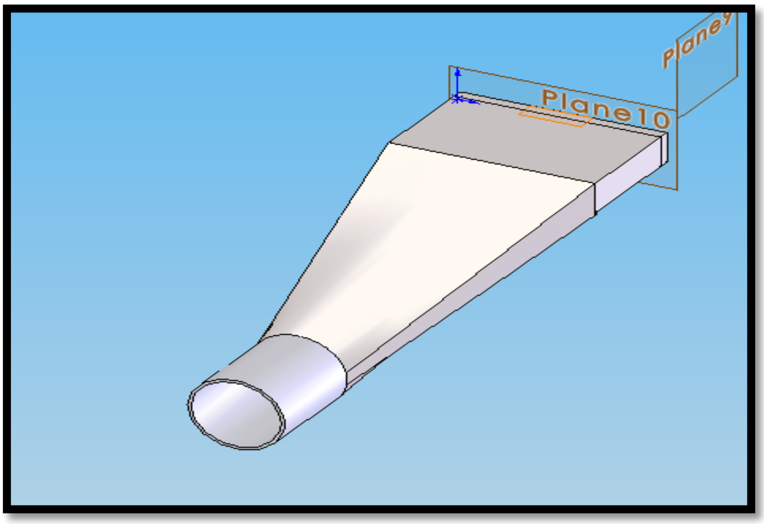


Figure 3.4

Air Pipe

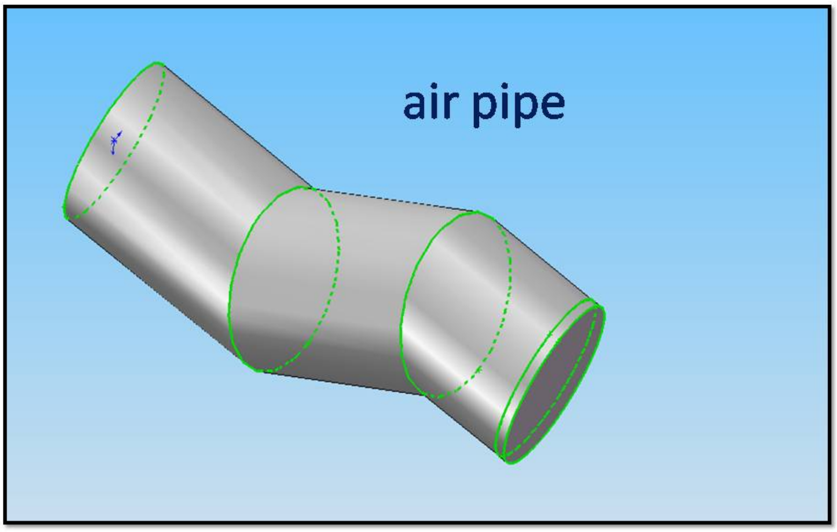


Figure 3.5

Upper box

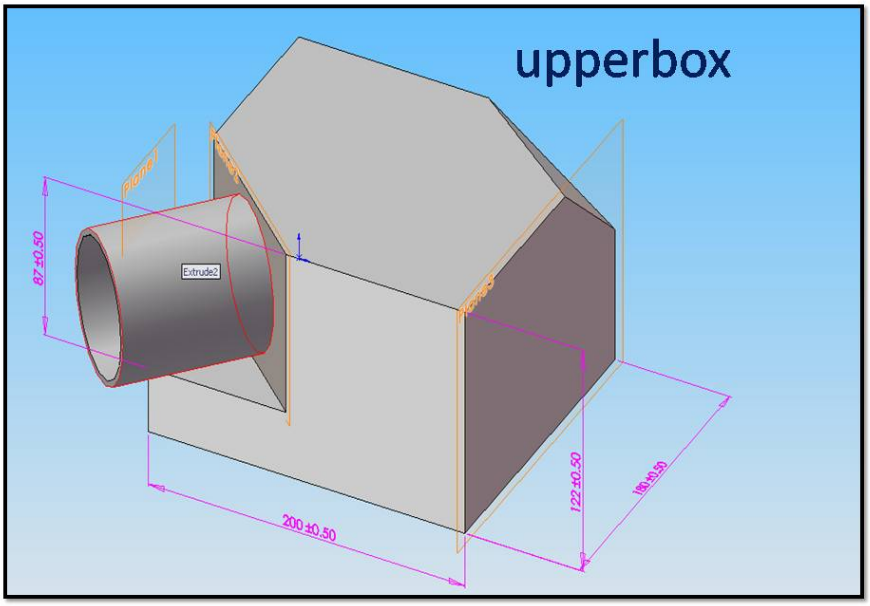


Figure 3.6

Down box

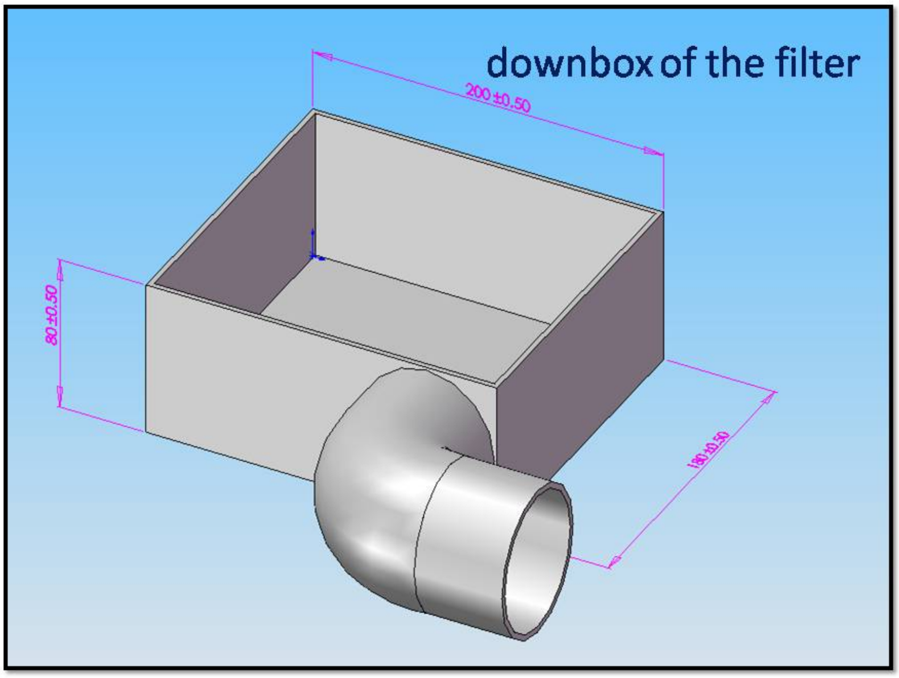


Figure 3.7

Filter

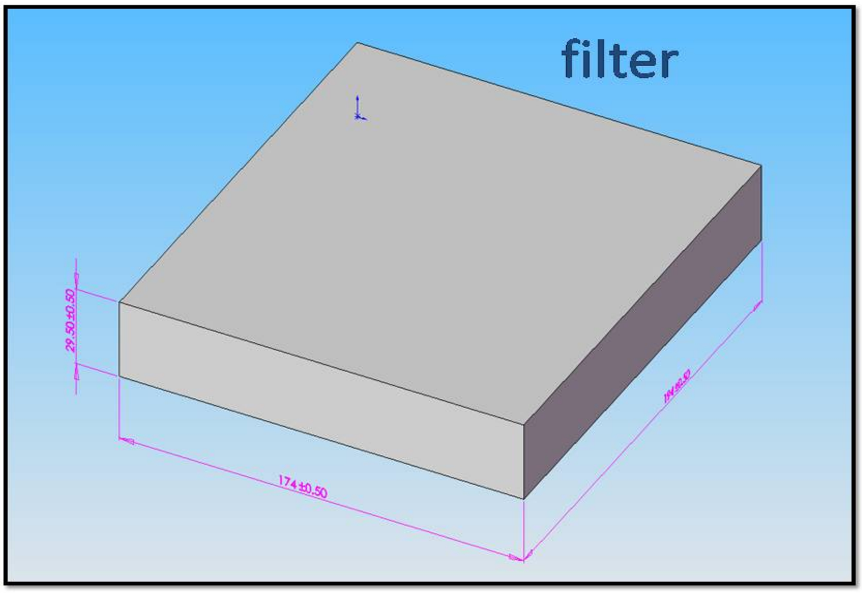


Figure 3.8

After Assemble

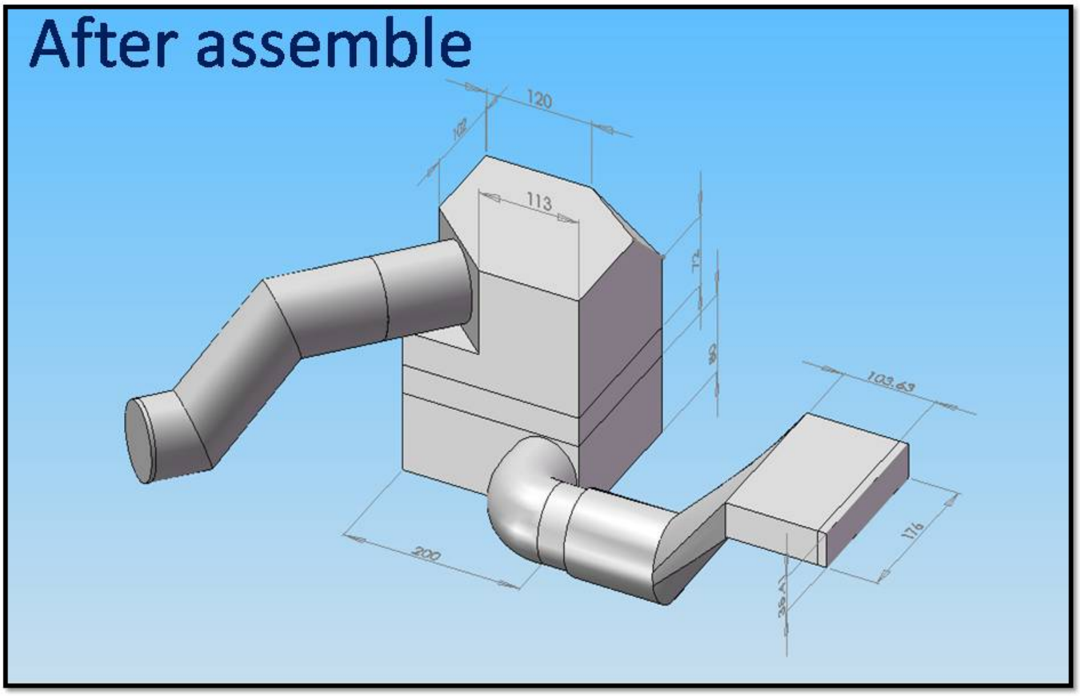


Figure 3.9

Orthographic View

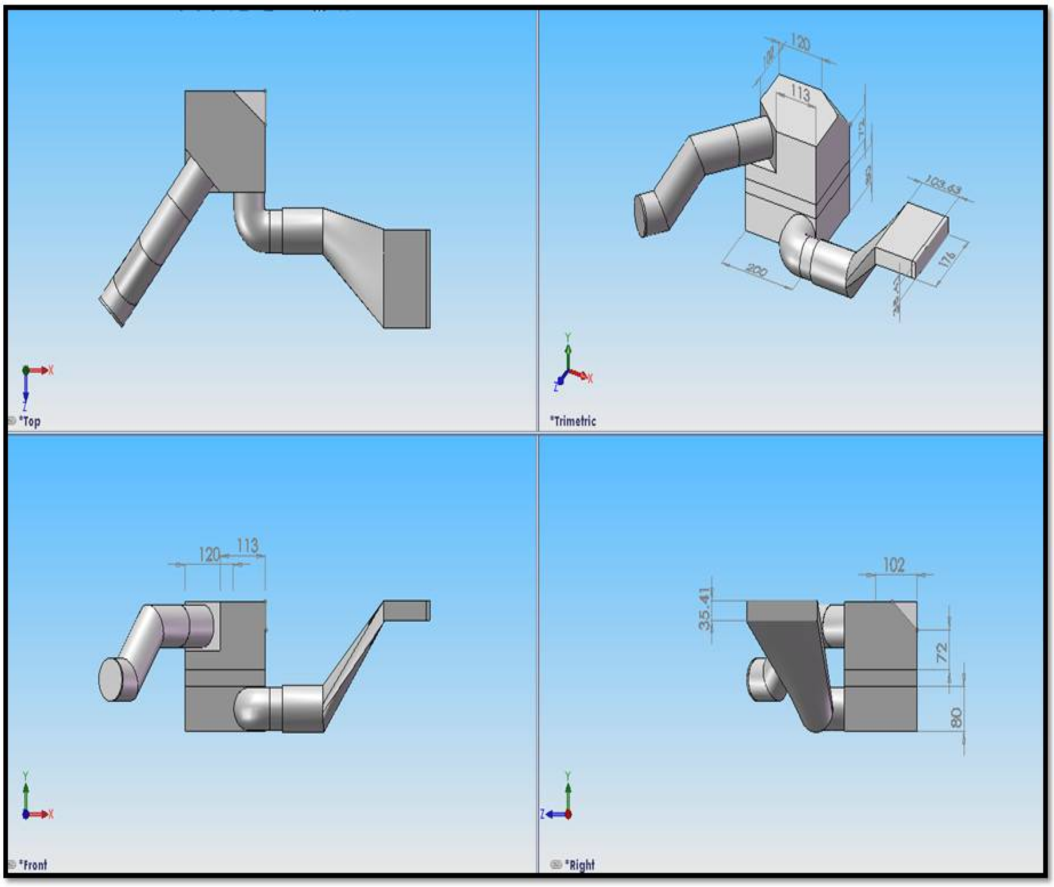


Figure 3.10

3.4 SIMULATION – USING CFD

After finish the designed, import the design from solidwork into CFD for analysis. For this project, different velocity of air based on the minimum rpm until maximum rpm of Proton Waja engine has been used. Select the power engine from minimum until maximum value: 1000 rpm, 2000rpm, 3000rpm, 4000 rpm, 5000 rpm, 6000 rpm and 7000 rpm.

3.4.1 Boundary Condition

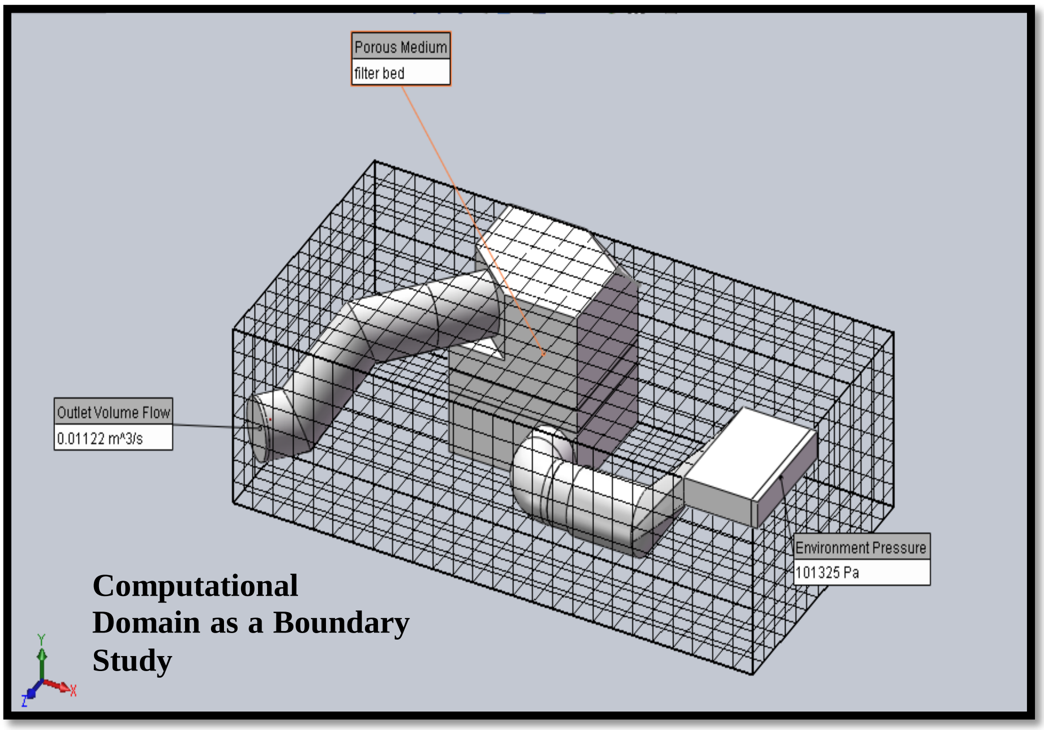


Figure 3.11: Boundary Condition of CFD analysis

The first step in simulation is setup boundary condition like Figure 3.11 Various boundary conditions for the different components applied to this study were as follows:

For inlet, the mass flow rate was imposed using the fixed mass inlet boundary condition. The value of density (1 kg/m3), total pressure (1 atm) and turbulence intensity (5%) were specified at the inlet boundary. For outlet, outflow boundary condition was imposed with flow rate weighting of 1. No slip boundary condition was applied on all wall surfaces. For main filter media, porous media boundary was imposed with $\alpha i = \beta I = 3000$. Whole domain was considered at 1 atm and at 298 K as initial condition.

3.4.2 Mesh

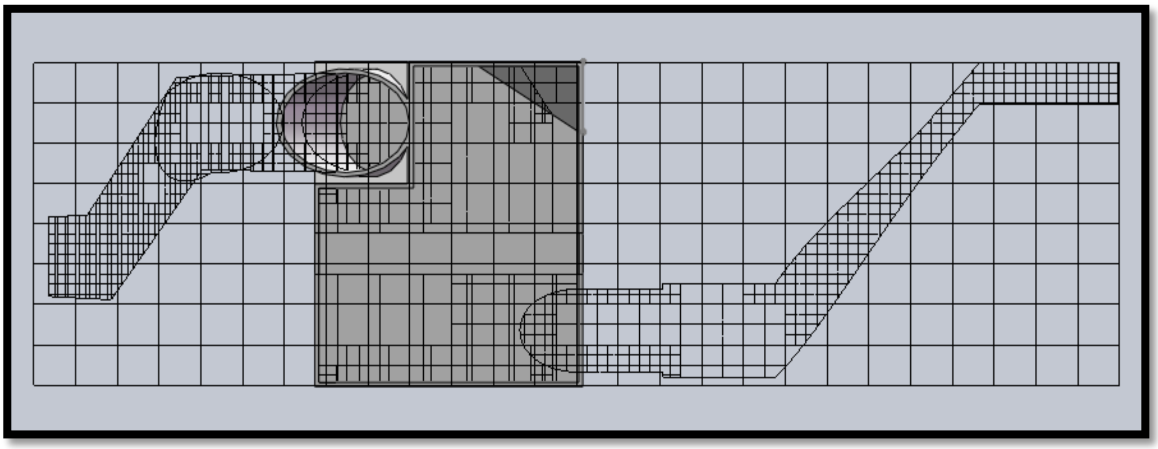


Figure 3.12: Generate mesh

To capture the three-dimensional flow inside the domain with reasonable accuracy, one needs good quality mesh. Multi-block structured hexagonal mesh was considered to be the best for this case and was created using commercial mesh generator. The model was approximately 0.55 million hexagonal fluid elements. Boundary layer was resolved for y+ of 40 to 200 to capture physics inside the complicated regions. Figure 3.12 shows hexahedral mesh of intake system fluid domain.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

In this chapter, all of the raw results will be rearranged and the selected finding will be discussed briefly to give the proper explanation about the analysis and the important point of the result. The relationship of different engine speed on the pressure drop will be discussed.

4.1 Data Collections

4.1.1 Reference Point of Flow Analysis

(a) Fluid type: Air

(b) Ambient parameter conditions

 i. Ambient Pressure: 101325 Pa

 ii. Ambient Temperature: 300.15 K

 iii. Density of Air: 1.225 kg m^{-3}

(c) Analysis type: Internal Flow

(d) Wall Condition: Adiabatic (Incompressible Flow)

4.2 VELOCITY VISUALIZATION

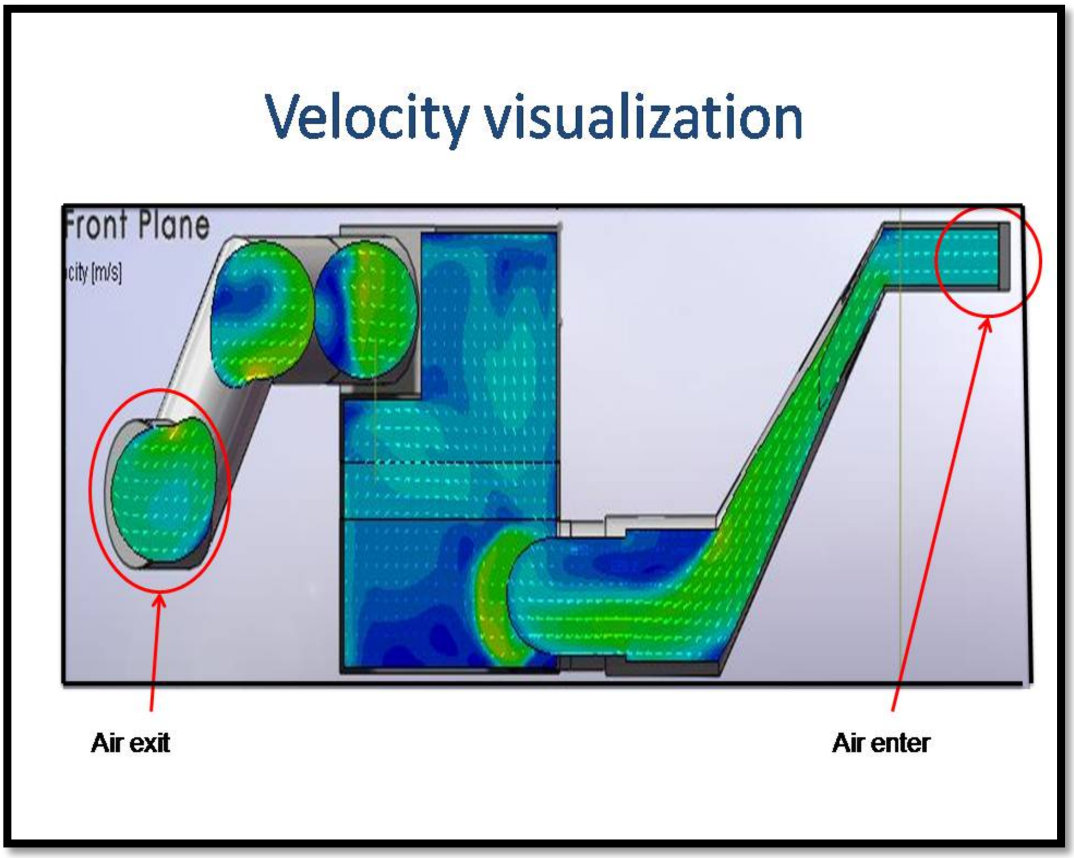


Figure 4.1: The visual shows vector and contour of velocity

The air is taken in through the air intake at the front of the engine. It passes through the intake system, air flow sensor and the throttle body. The figure shows velocity pressure visualization. The air enters from intake and the vector of air flow through the intake system. Figure 4.1 shows the contour plot of velocity for the ranging velocity's analysis. The blue and green color from the contour plot of velocity shows a difference velocity. If looked detail, the big slight differences contour plot of velocity at the pipe before entering the filter box.

4.3 PRESSURE VISUALIZATION

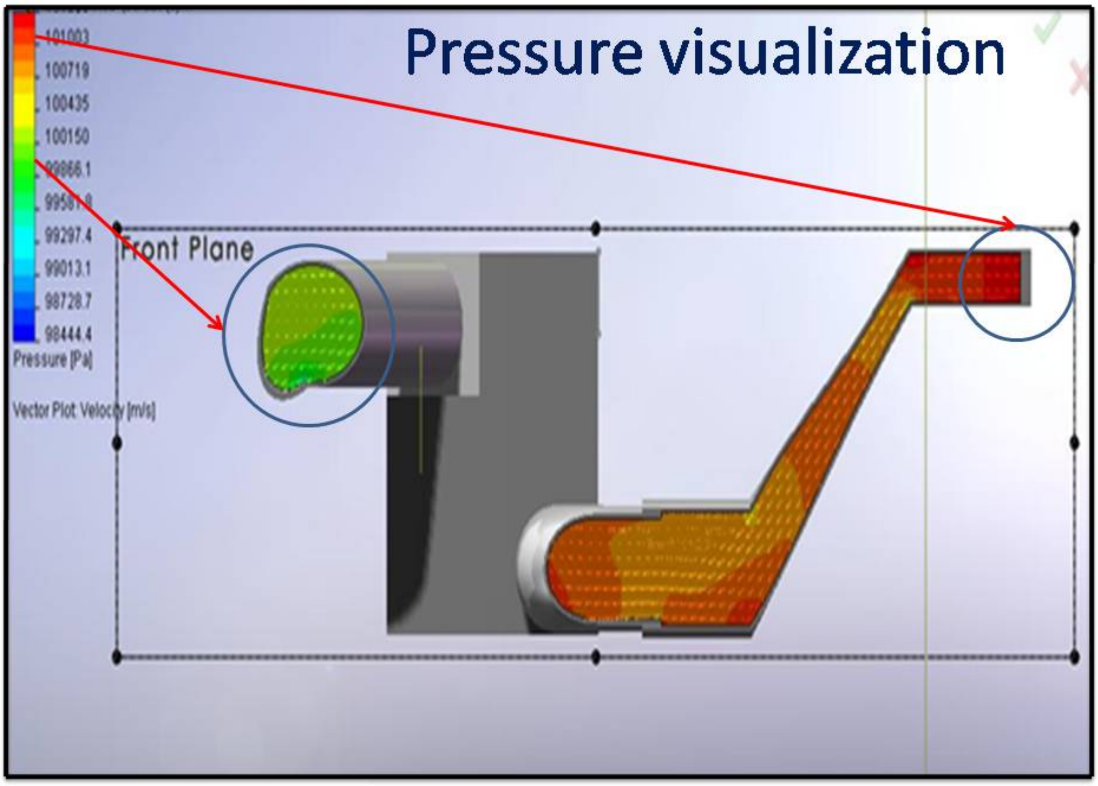


Figure 4.2

From Figure 4.2, the air is entering the intake system with high pressure and exit with low pressure. This means the pressure drop happen when the air across the intake system. The red and green color from the contour plot of pressure shows high and low pressure. The clear differences can look at the green region when the air flows exit the air intake.

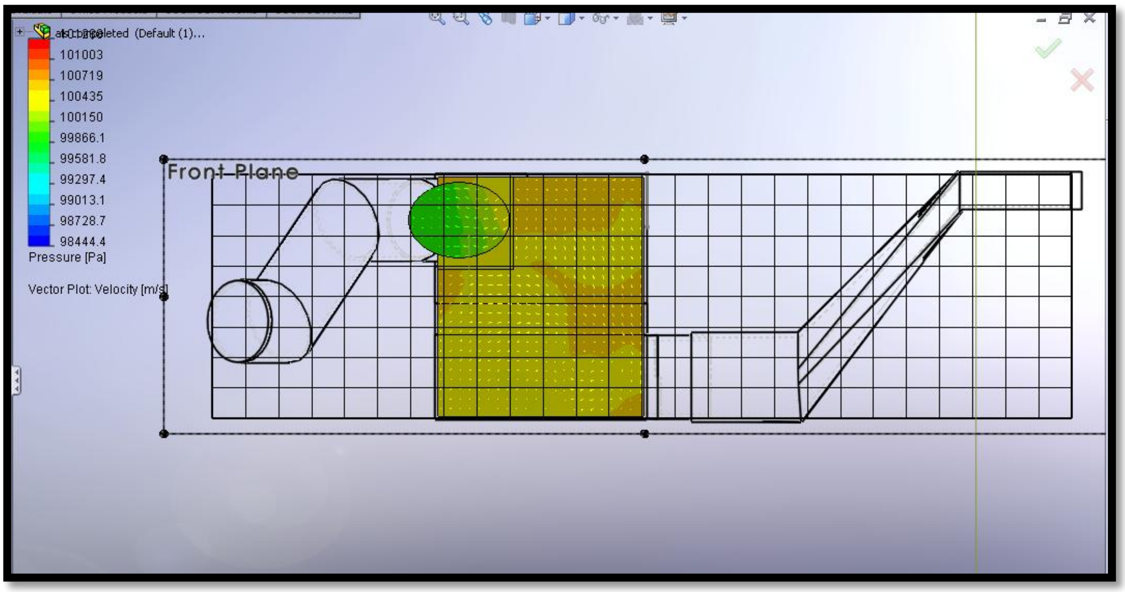


Figure 4.3: Visual of pressure at filter box

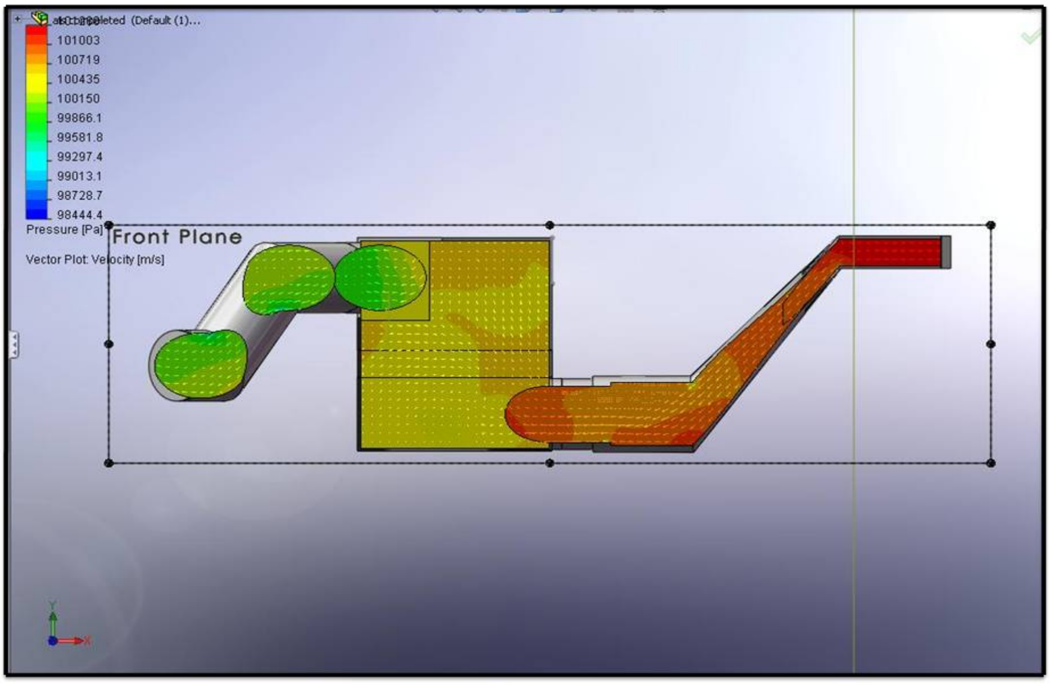


Figure 4.4: Full pressure visualization of air flow

4.4 SIMULATION ANALYSIS

From the CFD analysis, the result and the pressure drop will be determined. This is the preliminary result at the entering part (inlet pipe) when the power of engine is 1000 rpm. All parameter will come out from the analysis result. The analysis of simulation finished until iteration 121.

Table 4.1: Simulation result at entering pipe for 1000 rpm (iteration: 121)

Local parameters

Parameter	**Minimum**	**Maximum**	**Average**	**Bulk Average**
Pressure [Pa]	101321	101323	**101322**	101322
Temperature [K]	293.2	293.2	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	1.991	2.611	2.256	2.265
X-component of Velocity [m/s]	-2.611	-1.99	-2.256	-2.265
Y-component of Velocity [m/s]	-0.009	0.032	0.001	0.001
Z-component of Velocity [m/s]	-0.025	0.013	-0.012	-0.013

This result at the exit part (air pipe)

Table 4.2: Simulation result at exit pipe for 1000 rpm (iteration: 121)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101289	101298	**101295**	101295
Temperature [K]	293.196	293.199	293.198	293.198
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	2.835	3.477	2.918	2.917
X-component of Velocity [m/s]	-3.391	-1.424	-2.072	-2.068
Y-component of Velocity [m/s]	-0.973	1.380	-0.053	-0.061
Z-component of Velocity [m/s]	0.767	2.530	1.949	1.953

4.4.1 Calculation

Sample of calculation at 1000 rpm

101322 – 101295 = 27 Pa

The pressure drop:

Δp = 27 Pa

= 0.027 kPa

4.5 COMPLETE RESULT

For 2000 rpm (inlet)

Table 4.3: Simulation result at entering pipe for 2000 rpm (iteration: 115)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101184	101223	**101211**	101211
Temperature [K]	293.185	293.197	293.192	293.192
Density [kg/m^3]	1.202	1.202	1.202	1.202
Velocity [m/s]	5.671	6.921	5.835	5.830
X-component of Velocity [m/s]	-6.647	-2.937	-4.208	-4.197
Y-component of Velocity [m/s]	-1.378	3.092	-0.016	-0.041

2000 rpm (outlet)

Table 4.4: Simulation result at exit pipe for 2000 rpm (iteration: 115)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101310	101315	**101313**	101313
Temperature [K]	293.199	293.2	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	4.017	5.059	4.515	4.532
X-component of Velocity [m/s]	-5.059	-4.017	-4.515	-4.532
Y-component of Velocity [m/s]	-0.006	0.025	0.001	0.001

For 3000 rpm (inlet)

Table 4.5: Simulation result at entering pipe for 3000 rpm (iteration: 111)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101293	101303	**101298**	101297
Temperature [K]	293.2	293.2	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	6.027	7.281	6.729	6.757
X-component of Velocity [m/s]	-7.281	-6.027	-6.729	-6.757
Y-component of Velocity [m/s]	-0.018	0.016	-0.001	-0.001

Air pipe (outlet)

Table 4.6: Simulation result at exit pipe for 3000 rpm (iteration: 111)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101056	101132	**101105**	101106
Temperature [K]	293.165	293.194	293.182	293.182
Density [kg/m^3]	1.200	1.201	1.201	1.201
Velocity [m/s]	8.508	10.385	8.791	8.782
X-component of Velocity [m/s]	-10.047	-4.245	-6.400	-6.394
Y-component of Velocity [m/s]	-2.496	4.806	-0.178	-0.235

For 4000 rpm (inlet)

Table 4.7: Simulation result at entering pipe for 4000 rpm (iteration: 151)

Local parameters

Parameter	**Minimum**	**Maximum**	**Average**	**Bulk Average**
Pressure [Pa]	101268	101285	**101276**	101276
Temperature [K]	293.199	293.2	293.2	2932
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	8.159	9.759	9.008	9.041
X-component of Velocity [m/s]	-9.759	-8.159	-9.008	-9.041
Y-component of Velocity [m/s]	-0.010	0.028	0.001	0.001

Air pipe (outlet)

Table 4.8: Simulation result at exit pipe for 4000 rpm (iteration: 151)

Local parameters

Parameter	**Minimum**	**Maximum**	**Average**	**Bulk Average**
Pressure [Pa]	100827	100957	**100919**	100920
Temperature [K]	293.142	293.187	293.174	293.174
Density [kg/m^3]	1.197	1.199	1.198	1.199
Velocity [m/s]	11.343	13.762	11.586	11.579
X-component of Velocity [m/s]	-12.173	-6.258	-8.139	-8.118
Y-component of Velocity [m/s]	-1.631	7.418	0.169	0.123

For 5000 rpm (inlet)

Table 4.9: Simulation result at entering pipe for 5000 rpm (iteration: 147)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101236	101262	**101249**	101248
Temperature [K]	293.199	293.2	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	10.195	12.154	11.238	11.279
X-component of Velocity [m/s]	-12.154	-10.195	-11.238	-11.279
Y-component of Velocity [m/s]	-0.012	0.029	0.001	0.001

Air pipe (outlet)

Table 4.10: Simulation result at exit pipe for 5000 rpm (iteration: 147)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	100565	100768	**100691**	100691
Temperature [K]	293,112	293,184	293,16	293,16
Density [kg/m^3]	1,19491	1,19723	1,19634	1,19635
Velocity [m/s]	14,1781	16,548	14,5265	14,5209
X-component of Velocity [m/s]	-14,9808	-7,54785	-10,4616	-10,4462
Y-component of Velocity [m/s]	-4,29134	8,09947	-0,0005922	-0,0537897

For 6000 rpm (inlet)

Table 4.11: Simulation result at entering pipe for 6000 rpm (iteration: 135)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101198	101236	**101216**	101215
Temperature [K]	293.199	293.201	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	12.182	14.552	13.453	13.502
X-component of Velocity [m/s]	-14.552	-12.182	-13.453	-13.502
Y-component of Velocity [m/s]	-0.014	0.0312	0.001	0.001

Air pipe (outlet)

Table 4.12: Simulation result at exit pipe for 6000 rpm (iteration: 135)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	100226	100488	**100416**	100417
Temperature [K]	293.077	293.17	293.14	293.14
Density [kg/m^3]	1.190	1.193	1.193	1.193
Velocity [m/s]	17.011	20.529	17.376	17.365
X-component of Velocity [m/s]	-15.673	-7.189	-11.990	-11.989
Y-component of Velocity [m/s]	-4.415	11.487	0.545	0.475

For 7000 rpm (inlet)

Table 4.13: Simulation result at entering pipe for 7000 rpm (iteration: 138)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	101153	101204	**101177**	101176
Temperature [K]	293.199	293.201	293.2	293.2
Density [kg/m^3]	1.203	1.203	1.203	1.203
Velocity [m/s]	14.190	16.931	15.667	15.712
X-component of Velocity [m/s]	-16.931	-14.190	-15.667	-15.712
Y-component of Velocity [m/s]	-0.015	0.020	0.001	0.001

Air pipe (outlet)

Table 4.14: Simulation result at exit pipe for 7000 rpm (iteration: 138)

Local parameters

Parameter	Minimum	Maximum	Average	Bulk Average
Pressure [Pa]	99889	100226	**100128**	100129
Temperature [K]	293.032	293.164	293.119	293.119
Density [kg/m^3]	1.187	1.190	1.189	1.189
Velocity [m/s]	19.845	23.887	20.239	20.228
X-component of Velocity [m/s]	-18.281	-9.189	-13.971	-13.960
Y-component of Velocity [m/s]	-5.043	13.112	0.543	0.465

4.6 PRESSURE DROP RESULT

Base on the simulation results, the absolute value of pressure was tabled into Table 4.15 then plotted into graph. The differences pressure between at the inlet and outlet was calculated to define the pressure drop.

Table 4.15: Result of pressure drop based on engine speed

Engine speed (RPM)	Intake pressure (Pa)	Air pipe pressure (Pa)	Pressure drop (Pa)
1000	101322	101295	27
2000	101313	101211	102
3000	101298	101105	193
4000	101276	100919	357
5000	101249	100691	558
6000	101216	100416	800
7000	101177	100128	1049

4.7 PRESSURE DROP ANALYSIS

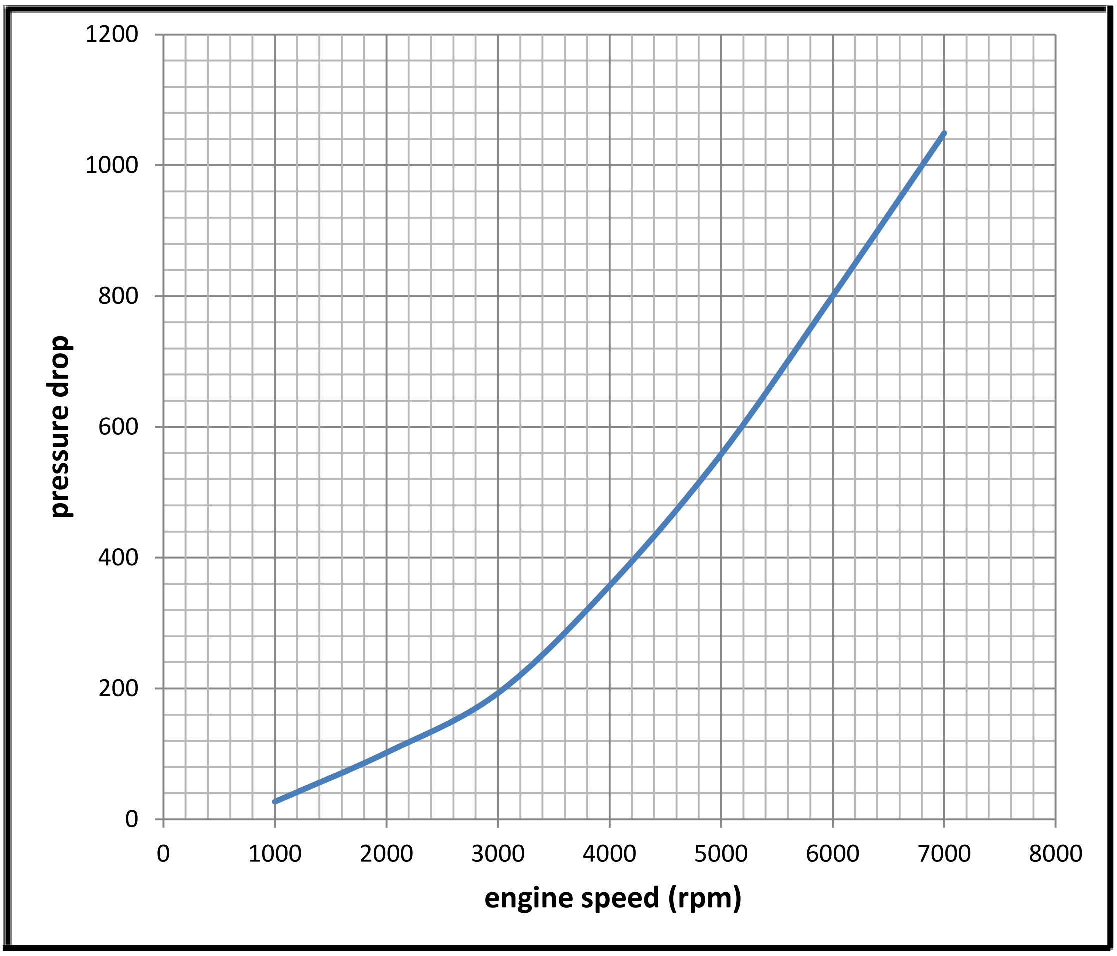
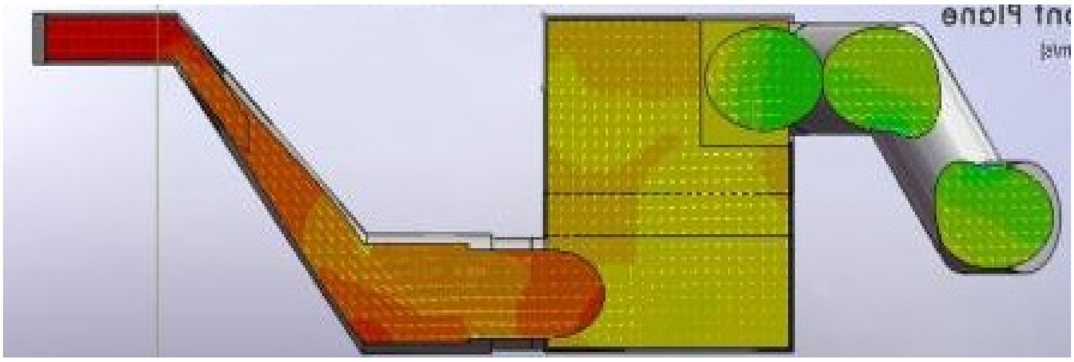


Figure 4.5: Pressure drop graph based on engine speed

The graph has shown the pressure drop again engine speed. The pressure drop has increased when the engine speed was increased. The high engine speed was produced the high velocity of air flow through the intake system. In this case, the less pressure drop across the intake system, it will increase the efficiency of combustion of the air in the intake system.



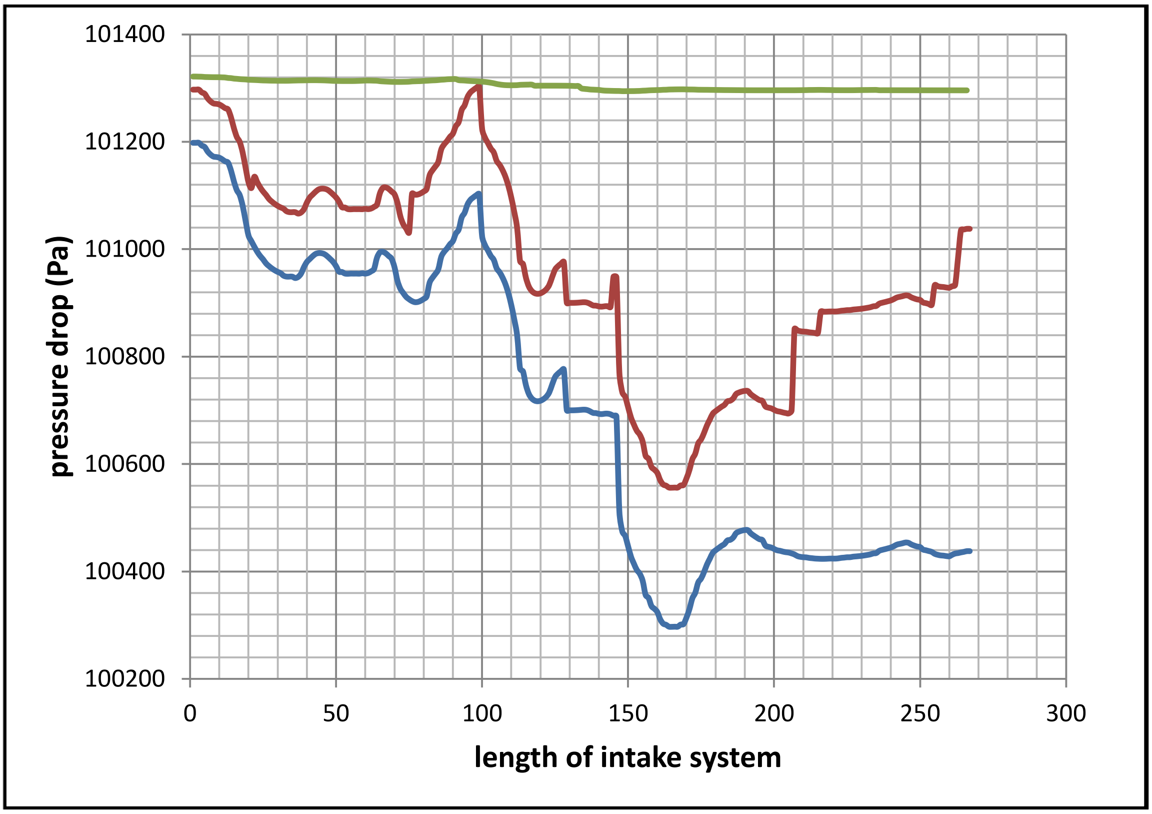


Figure 4.6: graph of pressure drop based on different engine speed

The differences between line in the graph based on the engine speed and the pressure drop of air flow that across the intake system. When the engine in high speed condition, the pressure drop will increase and the line shown the pressure loss looks drastically when the air flow came out from the inlet pipe into the filter box. The volume of intake was influent the pressure drop of the air flow.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

From the analysis, the value of pressure drop in certain rpm of engine power was determined by using CFD analysis. By using 3D CFD analysis, optimal design of the intake system for an automobile engine is achieved with considerable reduction in development time and cost. Learning process base on the estimation of pressure drop achieved using CFD specifically for internal body as well as the study of internal air flow on the pipe and air intake system. The analysis show pressure loss in term of pressure drop was proportionally increased with air velocity.

The contour plot of velocity and pressure were shown the rationalization of pressure drop graph analysis as a visualization analysis. The patent of visualization for every velocity depict quite same either for velocity contour plot or pressure contour plot. For the plot of pressure, the difference clearly shows at 7000 rpm of engine speed.

5.2 RECOMMENDATION

At some point in this project, there are some recommendations in order to overcome the constraints during finish the project. This recommendation can be used for the future in order to improve this project more successful and achieve more quality finding.

The experiment should be done to get the experimental result. This analysis can be done by using flow bench SF-1020 in lab that can be setup the speed of the Proton Waja 1.6 from maximum speed to minimum. Experiment result can be use for validation.

Figure 5.1: Flow bench machine

REFERENCES

[1] Graham Bell .2006. *Four-Stroke Perfomance Tuning*. Third edition.

[2] Gordon P. Blair .1999. *Design Simulation Four-Stroke Engines*.

[3] Collin R. Ferguson, Allan T. Kirkpatrick .2001. *Internal Combustion Engines, Applied Thermodynamics*. Second edition.

[4] Patankar, S.V .1980. *Numerical Heat Transfer and Fluid Flow*. Hemisphere, Washinton D.C

[5] Ed May and Les Simpson .2007. *Automotive Mechanics*. Seventh edition.

[6] Anthony E. Schwaller .2005. *Total Automotive Technology*. Forth edition.

[7] Stone, R. 1999. *Introduction to internal combustion engines.* USA: Society of Automotive Engineers, Inc.

[8] Bruce R. Munsan. Donald F. Young and Theodore H. Okiishi. 2006. *Fundamental of Fluid Mechanics*. Fifth Edition. John Wiley & Sons (Asia), Inc.

[9] Heinz Heisler. 2002. *Advanced Vehicle Technology.* Second Edition. Elsevier Butterworth Heinemann.

[10] Amann, C.A. 1999. Evaluating alternative internal combustion engines: 1950-1975. *Journal of Engineering for automotive engine and Power.* **121**(3): 540-545.

[11] Marit Kleven* and Morten C. Melaaen. *Fluid Flow And Particle Deposition Simulations In The Human Nose.* Telemark University College (HiT-T
Telemark R&D Centre (Tel-Tek)

[12] *FundamentalsGuide*. COSMOSFloWorks 2003. COSMOS. 2003