International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2015): 78.96 | Impact Factor (2015): 6.391

Modal Analysis of Intake Manifold of a Carburetor

Dr. A. John Presin Kumar[1], P. M. Selvaganapathy[2]

[1]Professor, Mechanical Engineering, Hindustan Institute of Technology and Science, Chennai, India

[2]P.G Student, Mechanical Engineering, Hindustan Institute of Technology and Science, Chennai, India

Abstract: ***Aim of the CFD Analysis is to develop engine performance and condense the emissions. To achieve the extreme mass flow rate and the output velocity should be high with an even distribution to each cylinder for efficient working of the engine. These factors are influence the engine performance such as compression ratio, fuel injection pressure, and quality of fuel, combustion rate, air fuel ratio, intake temperature and pressure, inlet manifold, and combustion chamber designs etc. Optimized geometrical scheme of intake manifold is one of these methods for the better performance of an engine. Optimized Intake manifolds provide better Air motion to the chamber. In the present research two different Manifold Designs viz. With C-D nozzle and without C-D nozzle are used for the Computational Fluid Dynamic(CFD) analysis using k- ε model to find which model gives maximum Mass flow and Output velocity and hence the performance of Engine can be improved.***

Keywords: Intake temperature, Combustion ratio, CFD

1. Introduction

The primary function of the intake manifold was to deliver the air / air-fuel mixture to the engine cylinder through the intake port with least flow losses. Certain intake manifolds were designed to enhance the flow swirl in the intake manifold to improve the combustion in the engine cylinder.

Also, based on the engine cylinder firing order, the intake manifold must supply evenly split air flow among the cylinders. This had been investigated in this work for a 4-

Cylinder IC engine intake manifold. Recent developments in the computer simulation based methods for designing automotive components had been gaining popularity. Even though the results obtained from these numerical simulations (CFD) were comparable with the experimental studies, there's been continuous research to improve the simulation accuracy.
In the available literatures, the intake manifold CFD simulations were performed by considering all the ports to be open. But, in actual conditions for the 4-cylinder engines, only two ports – based on the firing order- would be open. The other two ports would remain closed.

A similar condition was imposed in this study. The flow outlet condition was imposed for the two ports and the wall boundary conditions for the remaining two ports.

2. Problem Statement

In engineering field, the result of failure must be exactly true. Finite element analysis will be able to analysis the created design as well when all the specification is known, then, that can show the better result. From the review, there are several problems should be highlighted in this project.

These include:
1) Poor carburetor design may lead to high fuel consumption.
2) The pressure drop will also leads to poor air-fuel mixture ratio.
3) Uneven Distribution of Air will Leads to poor Mass flow and Output velocity

3. Objective

There are three main objectives that must be achieved:
1) To develop the geometry of the carburetor using CATIA Engineer software and FEA analysis by using CFD Software.
2) To investigate the pressure drop across all locations.
3) Experimental data and the modeling has provided a good insight into the flow details and also optimization of geometrical design to get a good mixing efficiency and get maximum Mass flow and Output velocity .

1) Modelling and Analysis
Here the Carburetor model is done by using CATIA Software. A product and its entire bill of materials can be molded accurately with fully associative engineering drawings and revision control systems. The associatively functionality in Pro/E enables users to make changes in the and automatically update downstream deliverables. This capability enables concurrent engineering design, analysis and manufacturing engineers working in parallel and streamlines development process.

a) *Specifying Geometry*: This can be done either by entering the geometric information in the finite element package through the keyboard or mouse, or by importing the model from a solid modeler like Pro/ E.
b) *Specify Element type & material properties*: In an elastic analysis of an isotropic solid, these consist of the Young's modulus and the Poisson's ratio of the material.
c) *Mesh the Object*: Then, the structure is broken into small elements. This involves defining the types of elements into which the structure will be broken, as well as specifying how the structure will be subdivided into elements.
d) *Apply Boundary Conditions & External Loads:* Next, the boundary conditions and the external

loads are specified.

e) Generate solution: Then the solution is generated based on the previously input parameters.

f) *Post-Processing*: Based on the initial conditions and applied loads, data is returned after a solution is processed. This data can be viewed in a variety of graphs and displays.

g) *Refine the mesh:* FEM are approximation increases with the number of elements used. The number of elements needed for an accurate model depends on the problem and the specific results from a single finite element run, you need to increase the number of elements in the object and see if or how the results change.

h) *Interpreting Result*: This step is perhaps the most critical step in the entire analysis because it requires that the modeler use his or her fundamental knowledge of mechanics to interpret and understand the output of the model. This is critical for applying correct to solve real engineering problems and in identifying when modeling mistakes have to be been made

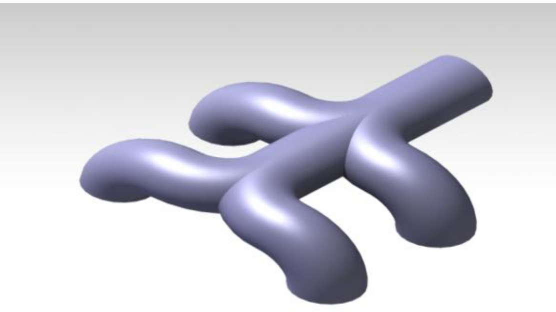

Figure 1.1: Carburetor without C-D Nozzle

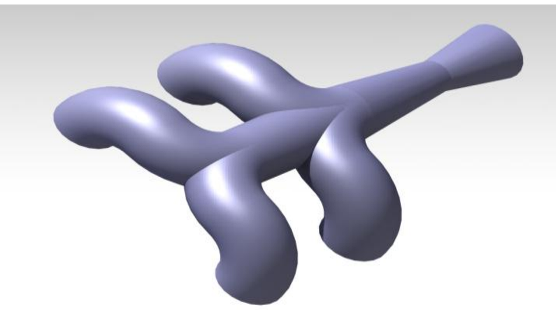

Figure 1.2: Carburetor with C-D Nozzle

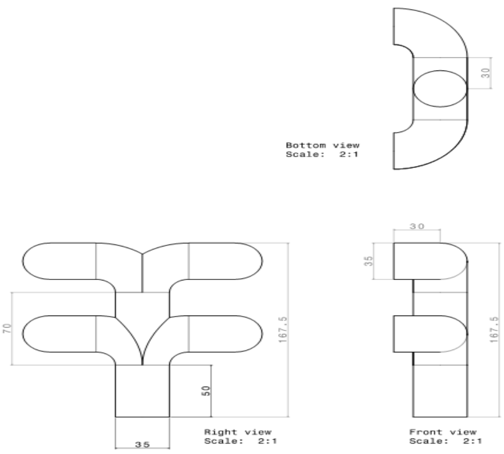

Figure 1.3: Drafted View

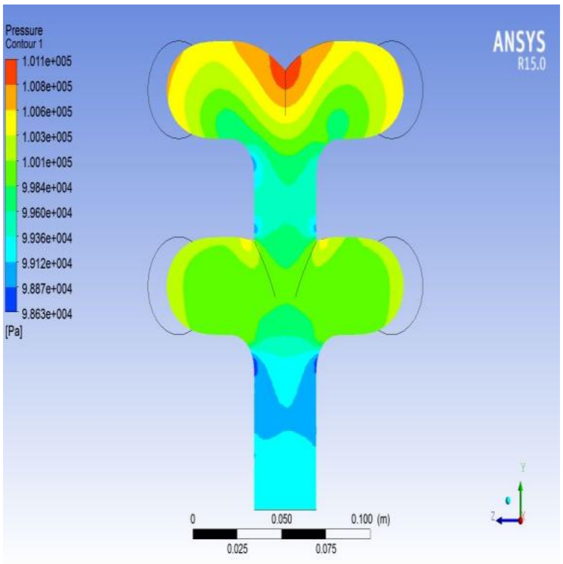

Figure 1.4: Pressure Plot without C-D Nozzle

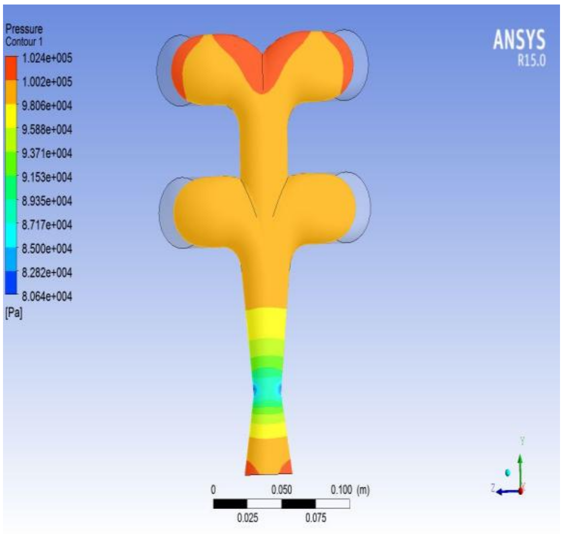

Figure 1.5: Pressure Plot with C-D Nozzle

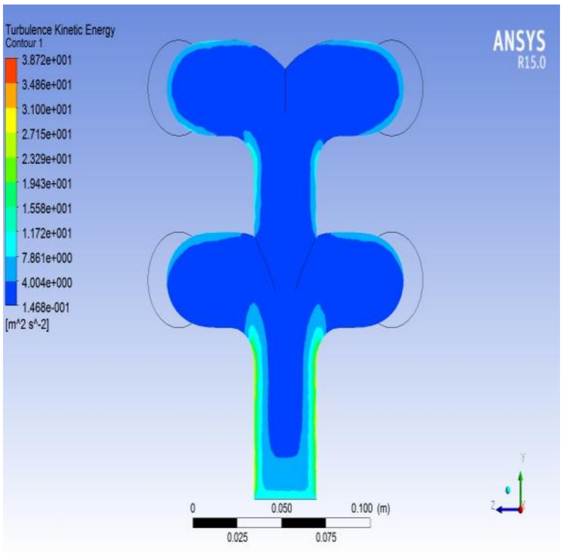

Figure 1.6: Turbulance Plot without C-D Nozzle

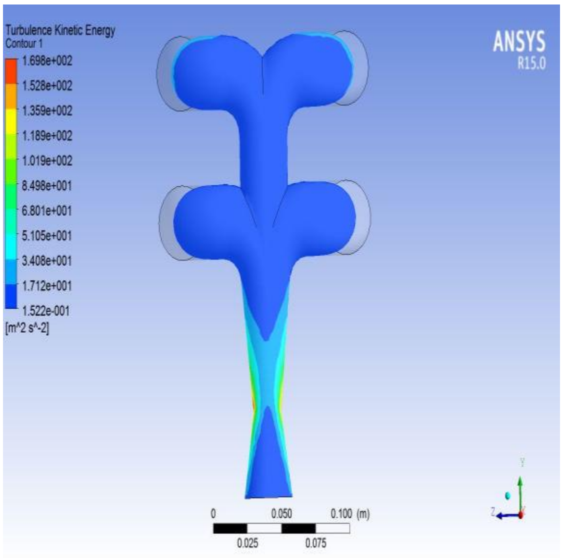

Figure 1.7: Turbulance Plot with C-D Nozzle

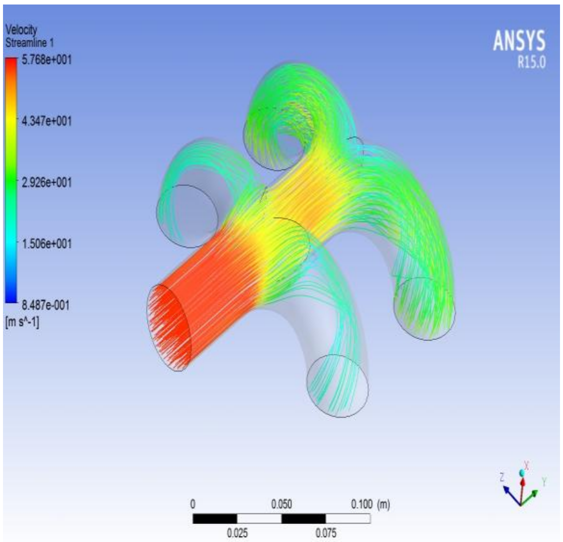

Figure 1.8: Streamline Velocity without C-D Nozzle

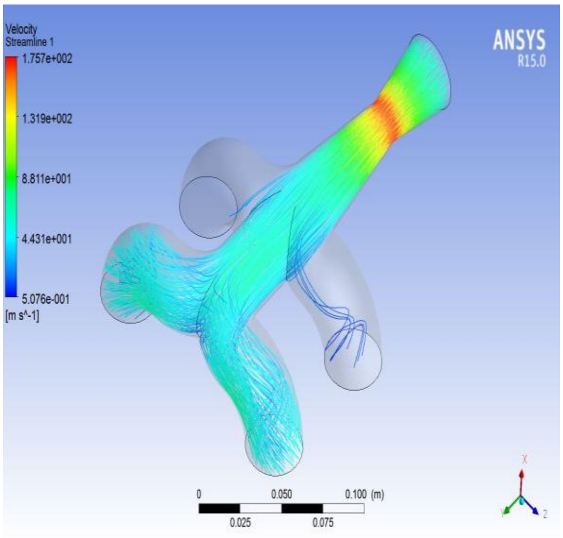

Figure 1.9: Streamline Velocity with C-D Nozzle

2) Analysis with Ports Closed

Streamline Velocity

The streamlines for Case A and D for both the firing order had been provided in images [Fig 10 - 13]. The flow swirl could be observed from these plots. The flow to the Port-2 appears to have the high swirling as compared to the remaining ports. An equally strong swirling in Port-1 could also be noted from these streamline plots.

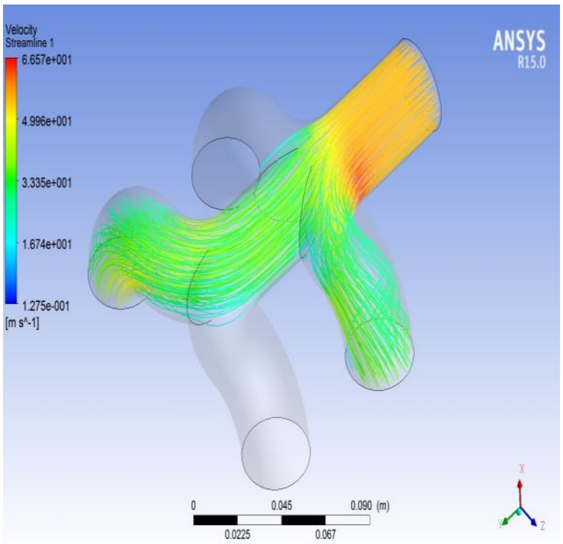

Figure 2.1: Streamline Velocity without C-D Nozzle Port 2 & 4 Closed

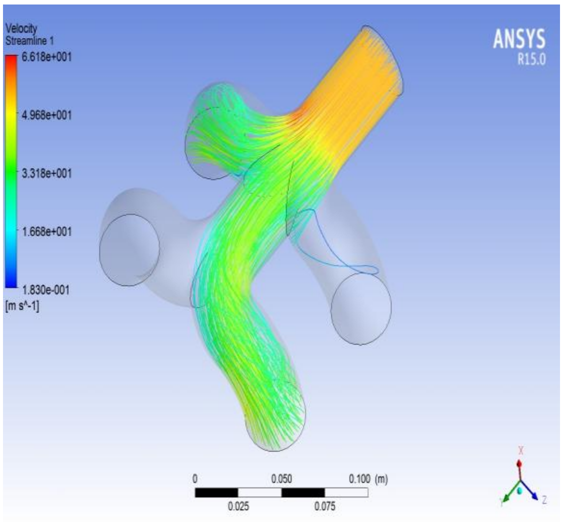

Figure 2.2: Streamline Velocity without C-D Nozzle Port 1 & 3 Closed

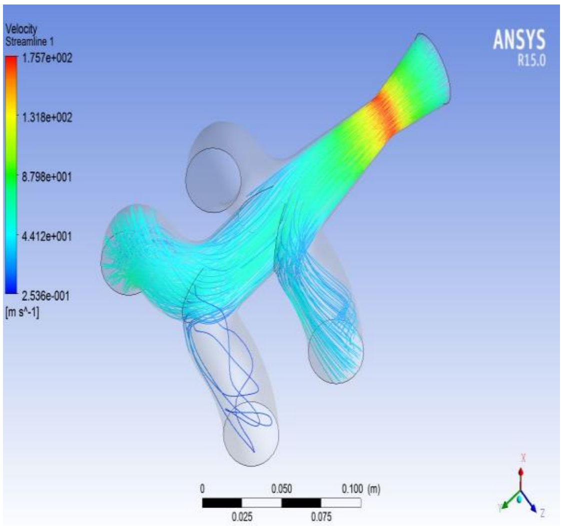

Figure 2.3: Streamline Velocity with C-D Nozzle Port 2 & 4 Closed

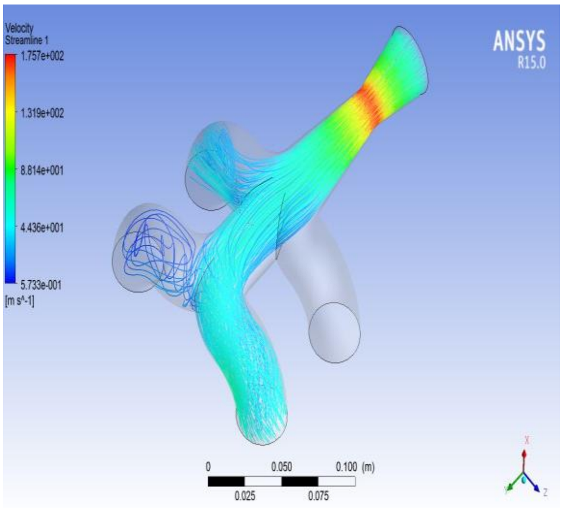

Figure 2.4: Streamline Velocity with C-D Nozzle Port 1 & 3 Closed

Velocity Plot

The flow loss had been defined as the total pressure drop from the inlet to the port outlets. It was expected that the manifold must offer least flow losses for the high engine performance. But, un-even mass flow split was observed for the Firing-Order 2-4 configurations. With Port-2 was receiving lower mass flow rate (35%) as compared to the Port-4 for all operating conditions.

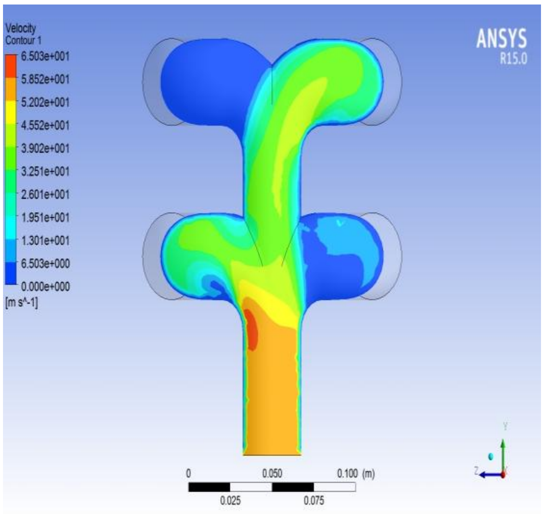

Figure 2.5: Velocity Plot without C-D Nozzle Port 2 & 4 Closed

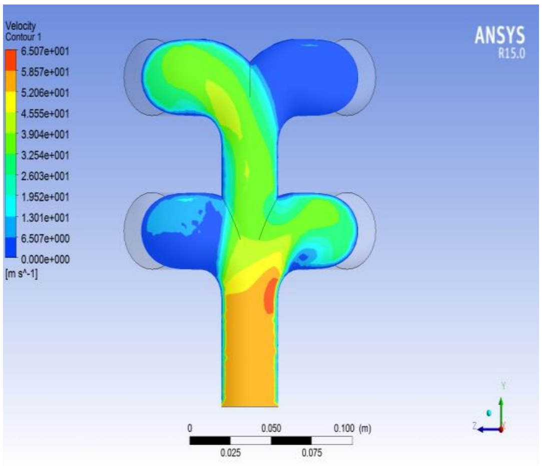

Figure 2.6: Velocity Plot without C-D Nozzle Port 1 & 3 Closed

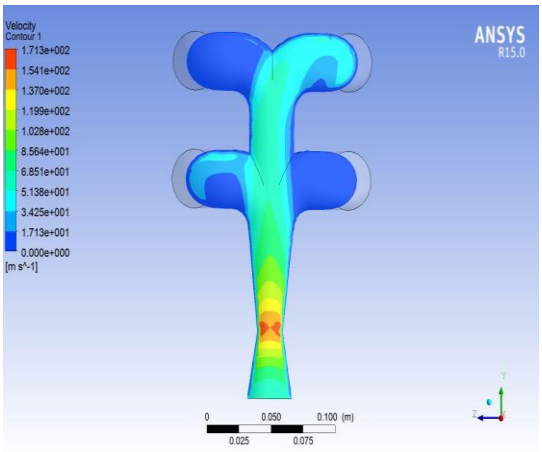

Figure 2.7: Velocity Plot with C-D Nozzle Port 2 & 4 Closed

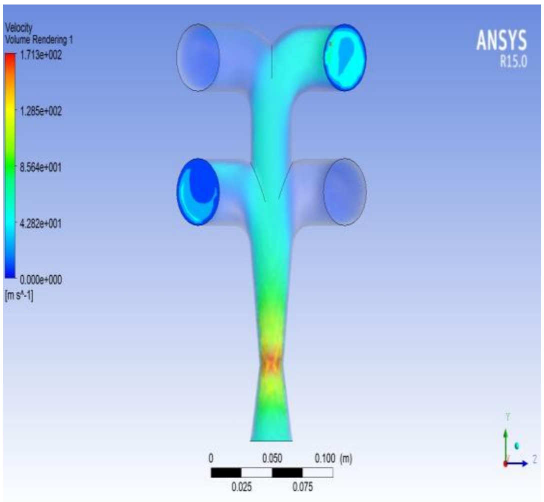

Figure 2.8: Velocity Plot with C-D Nozzle Port 1 & 3 Closed

Exit Velocity

The flow swirl at the port inlet could enhance better engine combustion characteristics. Since the identical flow patterns with different velocity magnitude was observed for the remaining cases, those were not presented here.

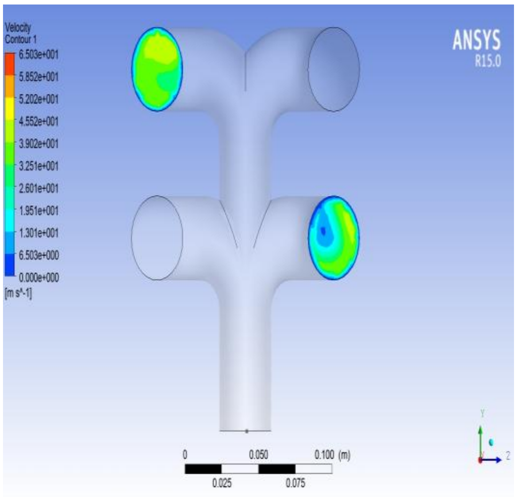

Figure 2.9: Exit Velocity without C-D Nozzle Port 2 & 4 Closed

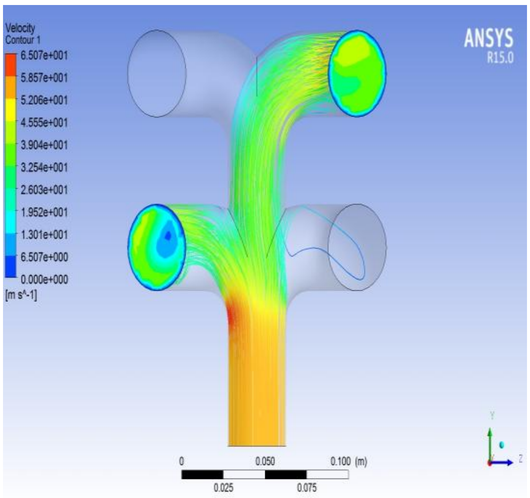

Figure 2.10: Exit Velocity without C-D Nozzle Port 1 & 3 Closed

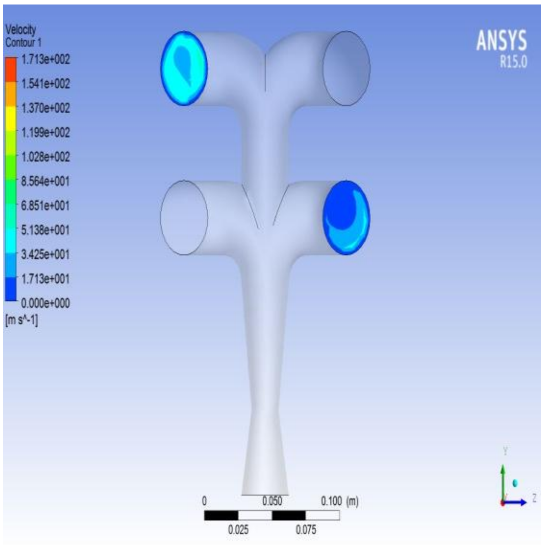

Fig 2.11: Exit Velocity with C-D Nozzle Port 2 & Closed

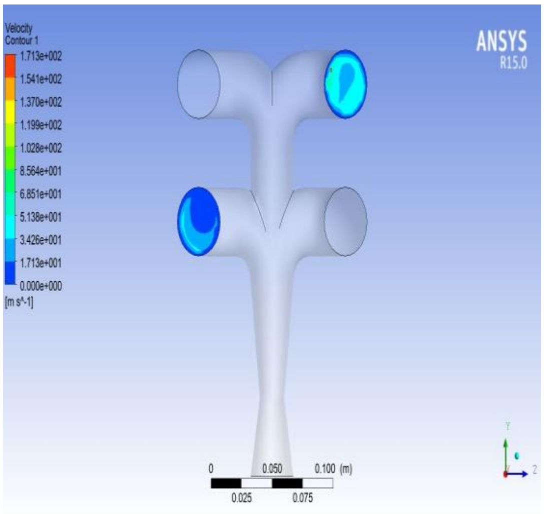

Figure 2.11: Exit Velocity with C-D Nozzle Port 1 & 3 Closed

3) CFD Analysis

The CFD simulations were performed using CFD software. The pre-processing activities that were needed for the CFD simulations like geometry clean-up, meshing, applying boundary conditions were completed in the CFD. The polyhedral mesh element type was chosen for meshing the computational volume of the Intake manifold.

A comparative study between the experimental and the CFD simulations was performed in terms of the flow losses. This was done all the 10 configurations. The results were plotted in the following images.

Boundary Conditions

S.no	Boundary	Values
1	Inlet Mass Flow Rate	0.063 Kg/s
2	Outlet Pressure	1 Bar
3	Wall no slip Pressure	Adiabatic
4	Model Type	K-Epsilon
5	Mesh Type	Tetra Hedral
6	Type of Fluid	Air at 25 degree

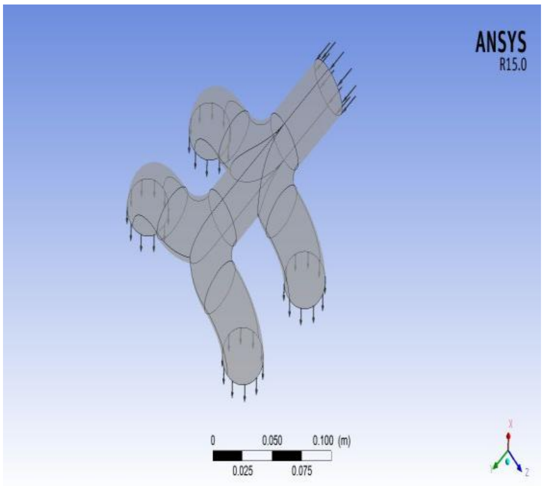

Figure 3.1: CFD Analysis of a carburetor without C-D Nozzle

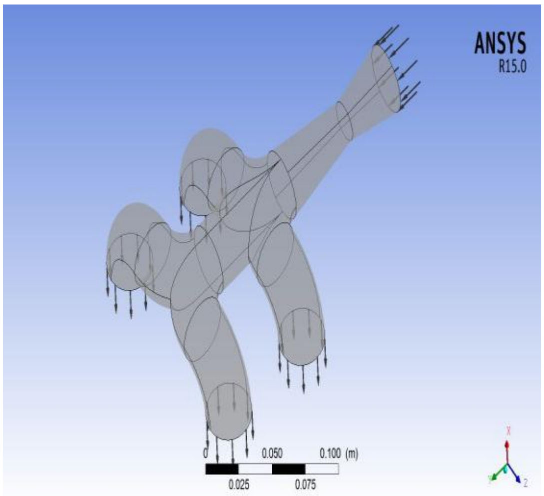

Figure 3.1: CFD Analysis of a carburetor with C-D Nozzle

4. Results

For without C-D Nozzle

S.no	Port 1	Port 2	Port 3	Port 4
For 1& 3	m=0.047 Kg/s	0	m=0.053 Kg/s	0
For 2 & 4	0	m=0.053 Kg/s	0	m=0.048 Kg/s

For with C-D Nozzle

S.no	Port 1	Port 2	Port 3	Port 4
For 1& 3	m=0.043 Kg/s	0	m=0.063 Kg/s	0
For 2 & 4	0	m=0.047 Kg/s	0	m=0.072 Kg/s

5. Conclusion

Based on this CFD Analysis, the following were the conclusions

- CFD simulation methodology –in terms of boundary conditions - for the intake manifold was proposed. The results are comparing with and without C-D Nozzle analysis.
- High flow swirl had been noted for all the Boundary conditions which could enrich the engine combustion features.
- The flow path for the Firing Order 2-4port with C-D Nozzle provided Greater mass flow rate at exit of the intake manifold system and it will helps to increase the combustion intensity and performance of the engine with compare to Without C-D Nozzle.

References

[1] http://ac.els-cdn.com/S2212827115008203/1-s2.0-S2212827115008203-main.pdf?_tid=3a119ccc-3591-11e7-973f-00000aacb35d&acdnat=1494428531_53885f1afd657b9bf270cd4488134ba8
[2] http://ac.els-cdn.com/S2212827115008203/1-s2.0-S2212827115008203-main.pdf?_tid=3158c2cc-3591-11e7-b24d-00000aab0f02&acdnat=1494428517_ccd8f6c9d608f6faa5d43eee04696899
[3] http://dellorto.it/wp-content/uploads/2015/10/dellorto_manual.pdf
[4] http://ethesis.nitrkl.ac.in/2296/1/final_report.pdf
[5] https://www.researchgate.net/publication/260477966_Analysis_and_Optimal_Design_of_a_Producer_Carburetor