Performance Analysis Of Carburettor Using CFD (Ansys Fluent)

Mohammad Javed[1], Dr.S.K.Biradar[2]
Department of Mechanical Engineering
[1, 2]MSS's College of Engineering and Technology

Abstract-*Today the whole world is facing the challenge from greenhouse emissions and high fuel consumption. Today there is need to reduce worldwide fuel consumption, greenhouse gas emissions, and air emissions. Carburetor plays a crucial role in SI engine performance which supplies correct mixture of fuel air at the right time. One of the important factors that affect the fuel consumption is the design of the carburetor. The throat of the carburetor provides required pressure drop in the venturi tube of the carburetor device. Currently, alternative fuels like LPG, CNG, etc. are gaining attention all over the world because of their eco-friendly nature. So the design of the carburetor is important. To get a better economy and uniform distribution of fuel air mixture, it required to design the carburetor with an effective analytical tool or software. In this work, CFD (FLUENT 15) analysis was carried out on a simple carburetor to find pressure drop and velocity profile for different throttle valve angle (30,45,60) and fuel discharge nozzle angle (34&38).*

For 38 deg Fuel discharge nozzle angle (FDNA) it is observed that more negative pressure gets generated at the outlet as compared to FDNA 34 deg. So it will suck more air fuel mixture into the engine resulting in proper and complete combustion of charge thereby generating more power.

In an Internal Combustion Engine the performance, efficiency and emission formation depends on the formation of air-fuel mixture inside the engine cylinder. The fluid flow dynamics plays an important role for air-fuel mixture preparation to obtain the better engine combustion, performance and efficiency. In this paper analysis of various Carburettors is done by using ANSYS FLUENT.The intensification of the swirl is done by providing masking on inlet valves. The Modeling of this carburettor is done by using Creo parametric 4.0.. The purpose is to create a more atomized mist of the air fuel mixture entering into the combustion chamber giving a more and complete burn of the fuel.

Keywords-Carburetion, CFD, ANSYS FLUENT, creo parametric.

I. INTRODUCTION

SI engines generally use volatile liquids. The preparation of the air fuel mixture is done outside the engine cylinder. The fuel droplets that remain in suspension also continue to evaporate and mix with air during suction and compression processes also. So carburetion is required to provide a combustible mixture of fuel and air in required quantity and quality [1]. The process of forming a combustible fuel air mixture by mixing the right amount of fuel with air before admission to the cylinder of the engine is called carburetion and the device is known as a carburetor. The flow through these internal passages may be quite complex and passages is short length.

All carburetors work on the Bernoulli's Principle which states that the velocity of a fluid increases, when the pressure drops. Within a certain range of velocity and pressure, the velocity increases with the drop in pressure. However, this linear relationship only holds within a certain range. Carburetor has to accelerate from rest, to some speed. It depends upon the air flow demanded by the engine speed and the throttle butterfly valve setting. According to Bernoulli's theorem, air flowing through the throat of the carburetor will be at a pressure less than atmospheric pressure, and related to the velocity.

The aim of present work is

1. To Study effect of throttle valve angle and fuel discharge nozzle angle on outlet pressure of carburetor.
2. To get Optimum Fuel economy and uniform distribution of Air-Fuel mixture
3. To reduce effect of green house gases and emissions.
4. Proper mixing of air and fuel mixture.

II. LITERATURE REVIEW

To get a better economy and uniform distribution of fuel air mixture, it required to design the carburetor with an effective analytical tool or software. In this work, CFD (FLUENT 14) analysis was carried out on a simple carburetor to find pressure drop and velocity profile for different throttle valve angle (30, 40, 50, 60, 70,80and 90 degree) and fuel discharge nozzle angle (33,36 and 39 degree). The results obtained from the analysis are analyzed for optimum design of a carburetor. Maximum pressure drop was observed at the throttle angle of 90 degree. Pressure distribution was observed more uniform at fuel discharge nozzle angle of 33degree.[1]

Diego Alejandro Arias [2] studied and conducted an experiment to validate the steady state model of a carburetor by measuring the fuel and air flows in a commercial (Nikki) carburetor. He used a flow-amplifier to create a low pressure zone downstream the carburetor. He compared the results obtained from the experiment and prediction of the steady state model. The uncertainty in the measurement was found to be ±2 cm3/min. These results indicated that the model was successful in showing the effects of the pressure drop and the metering elements in the emulsion tube. He also studied the quasi steady state and dynamic model.

1. Both the steady and dynamic models were used to study the effect of different geometry and physical properties of fuel and air flow.
2. He also used the models to calculate the gravitational and frictional pressure drop across the carburetor.
3. He developed an experimental set up to access the validity of the two phase flow models for both horizontal and vertical pipes.
4. He studied the effect of various parameters on the discharge coefficient. The parameters include the mesh sizes in case of small orifices and chamfered inlet and outlet etc.
5. He studied the effect of mesh size on the velocity profile of the square edged orifices.
6. He studied the effect of inlet and outlet chamfers on the static pressure.

The results obtained from his studies are

1. For the square edged orifices the result was within 5% agreement with the experimental results. The shortest orifice gave an agreement of 1% whereas the larger orifice gave 4.6% agreement.
2. He derived the expressions for prediction of the discharge coefficient by the information obtained from the velocity and pressure fields.
3. The outlet chamfer does not seem to affect the discharge coefficient.
4. The inlet chamfer favored the attachment of velocity profile to the wall and allowed for a development of the velocity profile.
5. The comparison with the FLUENT result showed the derived expressions were simple and effective.

He also studied the CFD analysis of the compressible flow across the carburetor venturi. The steps involved in the analysis process were

1. He developed a C program with 2 scripts. First script was to create the geometry of the carburetor in GAMBIT and the second script was to instruct the analysis of the model carburetor in FLUENT.
2. GAMBIT was used to create the geometry of the carburetor, to mesh the carburetor and to define the boundary conditions.
3. He used condor to run the different geometries and flow cases.
4. Finally he analyzed the solutions obtained in FLUENT.

The results of the above analysis are

1. When he considered different obstacles in the flow path, there was a larger decrease in flow pressure after the fuel tube and throttle plate.
2. In the absence of the fuel tube the inlet obstacles reduce the discharge coefficient. But with the presence of the fuel tube, suppose it is 3 mm long, all the different geometries show the same value of discharge coefficient.

Arias A Diego and Shedd A. Timothy [4] together worked to present a mathematical model of network of complex flow which contained short metering orifices, compressible flow and two-phase flow in pipes of small diameter. They have done a detail review of pressure drop, effect of fuel well and dynamic flow in the previously developed models. The homogeneous two-phase flow model were found to be very poor in agreement with the empirical correlation derived from the experiments on small pipes. They solved the instantaneous one-dimensional Navier-Stoke equation in single phase pipes to access the dynamic flow model. This was proved successful in explaining the mixture enrichment seen under pulsating flow conditions. They also used to model the model to derive a sensitivity analysis of geometries and physical properties of air and fuel.

III. EXPERIMENTATION AND ANALYSIS

In this study two different cases are studied for carburetors, depending on which the conclusions are drawn. The carburetors considered in this study are 100cc, 125cc & 150cc carburetors.

Case 1) air-fuel mixture flowing through 100cc.
Case 2) air-fuel mixture flowing through 125cc.
Case 3) air-fuel mixture flowing through 150cc.

In all the cases mentioned above, flow rate of air-fuel mixture in the form of pressure and velocity contours on inlet and outlet are studied.

3.1 Carburetor & its specifications

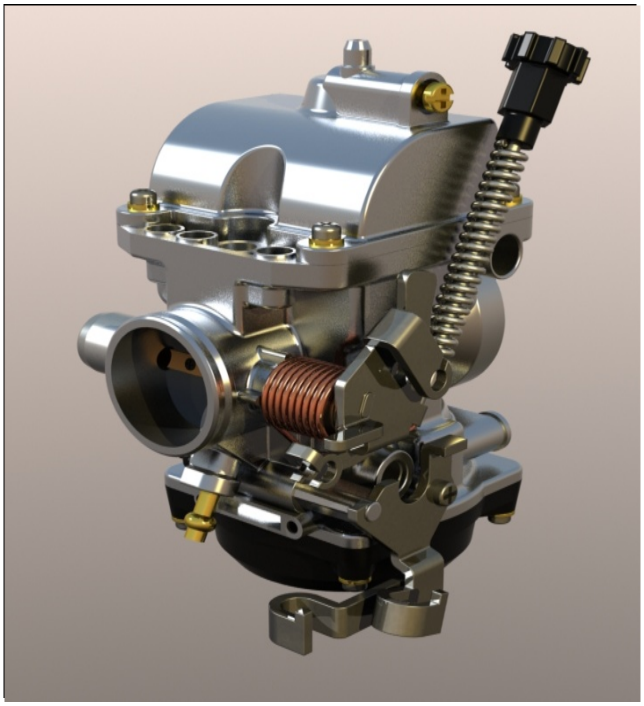


Fig.3.1 .General carburetor.

Table 3.1. Dimensions of carburetor

Sr. No.	Name of the parts All Dimensions are in mm	100cc	125cc	150cc
1	Inlet diameter	40	42	44
2	Outlet diameter	31	37	36
3	Total length	75	80	85
4	Throat diameter	21	27	28
5	Length of throat	5	5	7
6	Radius of venturi curve	9.5	8.5	7
7	Throttle plate position from outlet (both side)	20	22	24
8	Throttle plate thickness	1	1	1
9	Throttle plate diameter	21	25	28
10	Throttle plate angle	(30, 45, 60) degree.		
11	Fuel discharge nozzle angle	(34, 38) degree.		

3.2 MESHING

3.2.1 Mesh

There are mainly two aspects of grid generation for any CFD problem.
1) To achieve an optimal solution
2) To capture flow physics of given geometry, for which the grid must be sufficiently fine.

The quality of grid has an effect on convergence and stability of particular solution. While generating a grid for any geometry, initially a coarser grid is generated that gives an idea of initial solution. Depending on whether the given grid gives converging or diverging solution, element sizes are changed and grid is regenerated. This procedure is implemented number of times till an optimum solution is obtained.

3.2.2 Rate of convergence

The rate of convergence depends on the quality of mesh and the CFD solver. The computational time required to obtain the solution can be reduced with higher rate of convergence. Poor quality mesh may leave some important phenomena like boundary layer.

3.2.3 Solution accuracy

Accuracy of solution depends on quality of mesh. Better the quality of mesh, higher will be the accuracy of the solution. Fine mesh should be generated near walls, edges, curves to capture the exact geometry, where high gradients are possible. Refining the mesh in such areas increases the reliability of the solution.

3.2.4 Importing the geometry and pre-mesh setup

ANSYS module ICEM-CFD is used for meshing purpose. In order to mesh geometry, it needs to be imported into a meshing tool in IGES or STEP format. Once the geometry is imported, CAD clean-up operation is performed where; any open surfaces, missing surfaces are produced. Then all the surfaces are assigned with their respective names. In the solver we need to assign materials which will have influence on the solution.

3.2.5 Mesh Setup

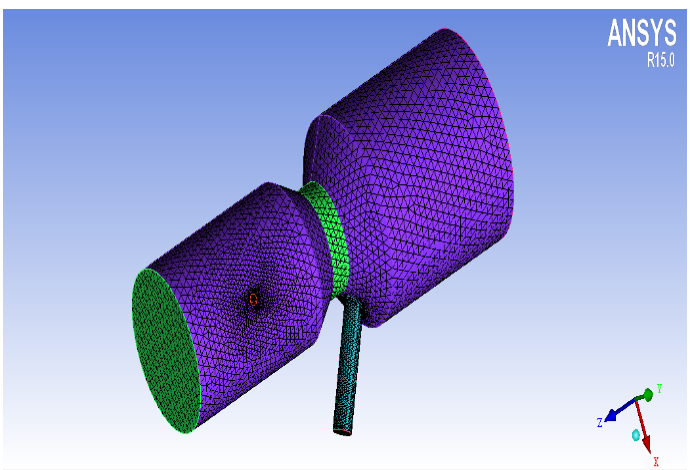
In order to discretise the control volume of the carburetor into number of small control volumes, different mesh sizes are tested. Mesh sizes giving quality metrics of 0.3 for the surface mesh and that of 0.2 for volume mesh and prism mesh are selected. Details of the same are given below.

Table.3.2 Mesh setup

1. Global Mesh Setup	
1.1 Global Mesh Size	
1.1.1 Scale factor	1
1.1.2 Max element size	5
1.2 Shell mesh Parameters	
1.2.1 Mesh type	All Tri
1.2.2 Mesh method	Patch Dependant
1.3 Volume meshing parameters	
1.3.1 Mesh type	Tetra/mixed
1.3.2 Mesh method	Quick (Delaunay)

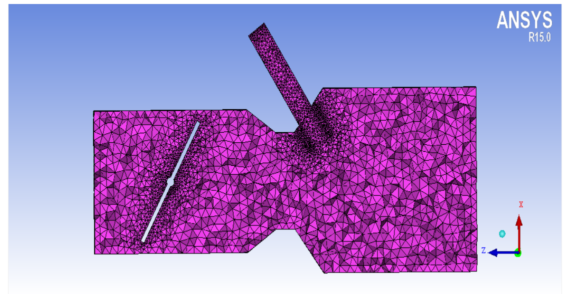
3.2.6 Surface mesh

Very first step in meshing process is generating a surface mesh which acts as an input to the volume mesh. For surface meshing, mesh type used is all triangles. Patch dependant algorithm is employed which follows curves on the surface and discretises the domain patch wise.



3.2.7 Volume mesh

A carburetor is a 3-D geometry. Hence generating a volume mesh is necessary. In the case of unstructured meshing, surface mesh acts as an input to the volume mesh. Element sizes in volume mesh will depend on sizes specified for surface mesh i.e. if the surface mesh is fine in nature then volume mesh will also be fine. Here mesh type of Tetra elements and quick (Delaunay) algorithm is used for volume meshing



3.3. SOLVER SETUP

3.3.1. Solver settings

The output file generated in ICEM is imported in ANSYS solver FLUENT for solution, where basic setup needs to be done. Basic set up includes switching energy equation ON/OFF, selecting desired turbulence model, defining and assigning materials to their respective cell zones, setting up boundary conditions, applying monitors to the surfaces for monitoring results in fluid properties like temperature, pressure, velocity etc.

3.3.2. Models

(A) Energy equation

As the carburetor involves thermal calculations, energy equation needs to be selected. The form of energy equation used by FLUENT is given below,

$$\delta/\delta t(\rho E)+\nabla\cdot(\vec{v}(\rho E+p))=\nabla\cdot(k_{eff}\nabla T) \qquad (4.1)$$

Where $k_{eff}=k+kt$ is effective thermal conductivity.
kt= Turbulent thermal conductivity

(B) Turbulence modeling

To capture the effects of turbulence, a two equation SST $k-\omega$ turbulence model is used, where k stands for turbulent kinetic energy and ω stands for specific dissipation rate. The main difference between standard and SST $k-\omega$ model is the way in which model calculates the turbulent viscosity in account of the transport of the principal turbulent shear stress. SST $k-\omega$ model is developed using transformed $k-\varepsilon$ and standard $k-\omega$ model. ω equation contains a cross diffusion term and a blending function which triggers $k-\omega$ model for near wall treatment and triggers $k-\varepsilon$ model in the region away from wall. The equations for k and ω (14) are as follows,

1) Turbulent kinetic energy (k)

$$\delta k/dt+U_j\delta k/\delta x_j=P_k-\beta^{*}k\omega+\delta/\delta x_j[(\nu+\sigma_k\nu_T)\delta k/\delta x_j] \qquad (4.2)$$

2) Specific dissipation rate (ω)

$\delta\omega/\delta t+U_j\delta\omega/\delta x_j=\alpha S^2-\beta\omega^2+\delta/\delta x_j[(\nu+\sigma_\omega\nu_T)\delta\omega/\delta x_j]+2(1-F_1)\sigma\omega_2 1/\omega\delta k/\delta x_i\delta y/\delta x_i$ (4.3)

Where F_1–blending function $F_1=\tanh\{\{min[max(\sqrt{k}\beta*\omega y,500\nu y2\omega),4\sigma\omega 2kCDk\omega y2]\}4\}$

$CD_{k\omega}=max(2\rho\sigma\omega^2 1/\omega\delta k/\delta x_i\delta\omega/\delta x_i,10^{-10})$

ν_T – Kinematic eddy viscosity $\nu_T=a1kmax(a_1\omega,SF_2)$

F_2 – Second blending function $F_2=\tanh[[max(2\sqrt{k}/\beta^*\omega y,500\nu/y^2\omega)]^2]$

3.3.3. Cell zone conditions

In cell zone conditions window, every solid or fluid body is assigned with desired material. So that the solver understands which fluid is flowing through the shell and which fluid is flowing through tubes.

3.3.4. Boundary conditions

Boundary conditions are known at some particular surfaces i.e. air inlet, fuel-inlet and mix-outlet. The mass flow rate is assigned at the inlet, pressure outlet condition is assigned at outlet.

3.3.5. Solution methods

Simple scheme is used for solution. In SIMPLE scheme, a pressure field is guessed which is used to solve momentum equations. From continuity equation a pressure correction equation is constructed. This pressure correction equation is then solved to obtain pressure correction field, which in turn is used to update pressure and velocity fields. These guessed fields are iteratively improved until the convergence is obtained for pressure and velocity fields. For pressure, momentum, turbulent kinetic energy, specific dissipation rate and energy, first order upwind scheme is used in the beginning, then it is switched to second order upwind scheme for more accurate results.

3.3.6. Monitors

Monitors are applied at surfaces such as:-
1) air-inlet,
2) fuel- inlet,
3) mixture-outlet, and
4) Venturi

For monitoring field variables such as pressure, velocity etc. Pressure and velocity monitors are applied as mass weighted average.

3.3.7. Post-processing

Post-processing is the final step in CFD analysis. It involves organization, interpretation and presentation of the results. The following steps are involved:

5. Production of CFD images and animations showing the flow field and other relevant variables
6. Calculation of integral parameters
7. Analysis and interpretation of the results
8. Reporting

It is not unusual that the insight gained from the first round of CFD analysis prompts another round aimed at making improvements to the model. Depending on the nature of changes, the whole process, or at least most of its steps, has to be repeated for each round.

The post-processing program is used to make evaluation of the data generated by the CFD analysis. When the model has been solved, the results can be analyzed both numerically and graphically. Post-processing tools of the powerful CFD software can create visualization ranging from simple 2-D graphs to 3-D representations. Typical graphs obtained with the post-processor might contain a section of the mesh together with vector plots of the velocity field or contour plots of scalar variables such as pressure. In such graphs, colors are used to differentiate between the different sizes of the values. When some results have been obtained, they must be analyzed, first to check that the solution is satisfactory and then to determine the actual flow data.

IV. CFD ANALYSIS

In this study two different cases are studied for carburetors, depending on which the conclusions are drawn. The carburetors considered in this study are 100cc, 125cc & 150cc carburetors.

Case 1) air-fuel mixture flowing through 100cc.
Case 2) air-fuel mixture flowing through 125cc.
Case 3) air-fuel mixture flowing through 150cc.

In all the cases mentioned above, flow rate of air-fuel mixture in the form of pressure and velocity contours on inlet and outlet are studied.

4.1. Governing equations

4.1.1. Continuity equation

$\delta u/\delta x+\delta v/\delta y+\delta w/\delta z=0$

4.1.2. Momentum equations

1) x-momentum equation

$$u\delta u/\delta x+v\delta u/\delta y+w\delta u/\delta z=-1\rho\delta P/\delta x+(\delta^2u/\delta x^2+\delta^2u/\delta y^2+\delta^2u/\delta z^2) \quad (5.1)$$

2) y-momentum equation

$$u\delta v/\delta x+v\delta v/\delta y+w\delta v/\delta z=-1\rho\delta P/\delta y+(\delta^2v/\delta x^2+\delta^2v/\delta y^2+\delta^2v/\delta z^2) \quad (5.2)$$

3) z-momentum equation

$$u\delta w/\delta x+v\delta w/\delta y+w\delta w/\delta z=-1\rho\delta P/\delta z+(\delta^2w/\delta x^2+\delta^2w/\delta y^2+\delta^2w/\delta z^2) \quad (5.3)$$

4.1.3. Energy equation

$$u\delta T/\delta x+v\delta T/\delta y+w\delta T/\delta z=\alpha(\delta^2T/\delta x^2+\delta^2T/\delta y^2+\delta^2T/\delta z^2) \quad (5.4)$$

4.2. Boundary conditions

In this study of carburetor, three different boundary conditions are studied.
1. Air inlet,
2. Fuel inlet,
3. Mixture outlet.

4.3. Effect of throttle valve position on air pressure at the throat section

In this work, air is assumed to enter at atmospheric temperature and pressure. Results were obtained for pressure variation when air is flowing through the carburetor for different throttle valve angles. The analysis was done for 30º, 45º and 60ºrespectively.

4.3.1.100CC CARBURETOR

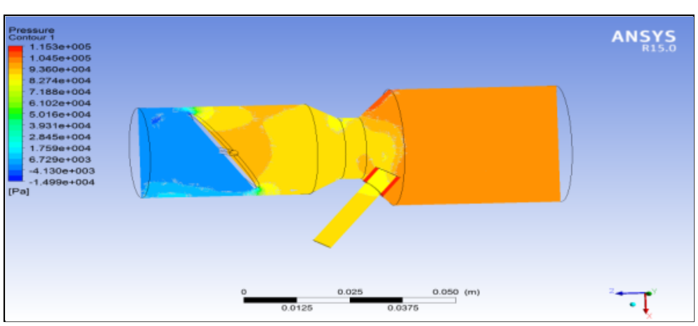


Fig.4.3.1.1.(a) Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 30

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

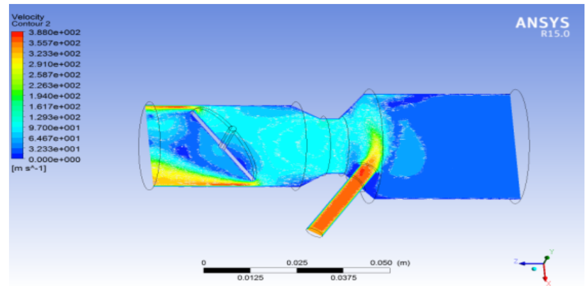


Fig.4.3.1.1.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 30

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

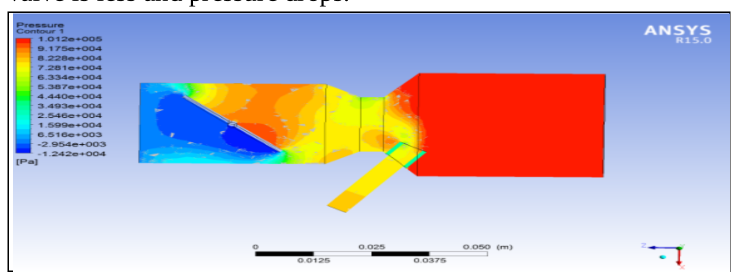


Fig.4.3.1.2.(a) Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 45

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

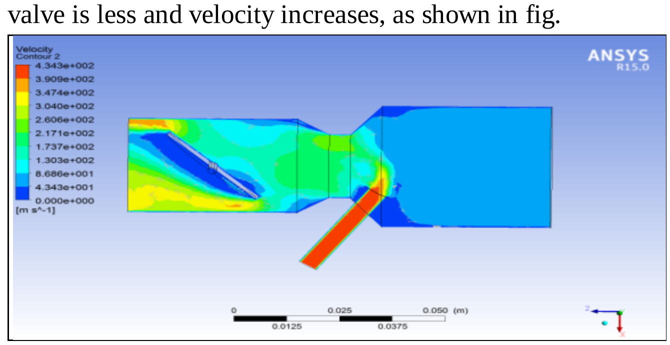


Fig.4.3.1.2.(b) Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 45

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

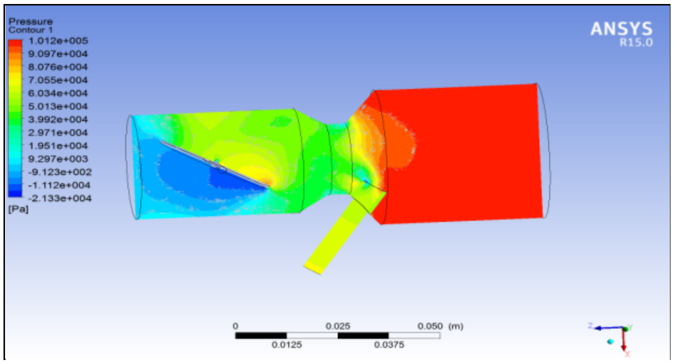


Fig.4.3.1.3.(a). Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 60

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

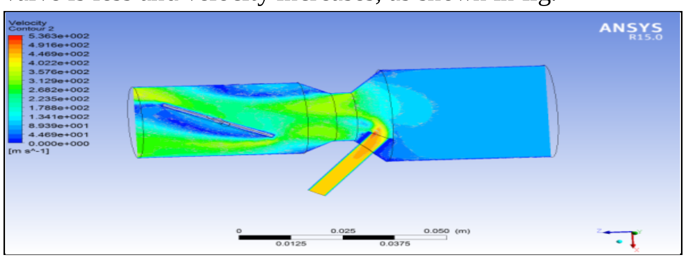


Fig.4.3.1.3.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 60

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

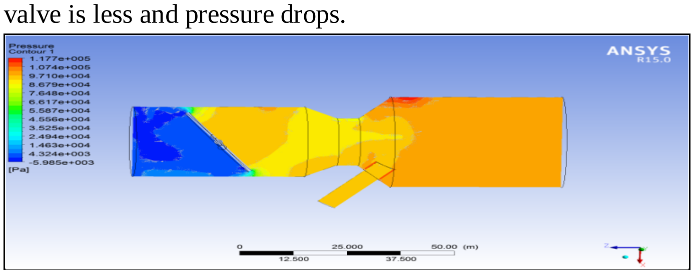


Fig.4.3.1.4.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 30.

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

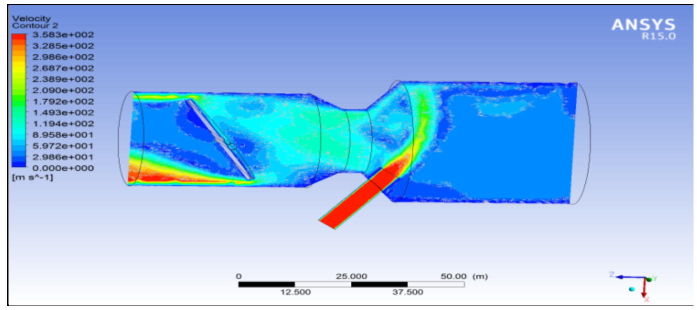


Fig.4.3.1.4.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 30.

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

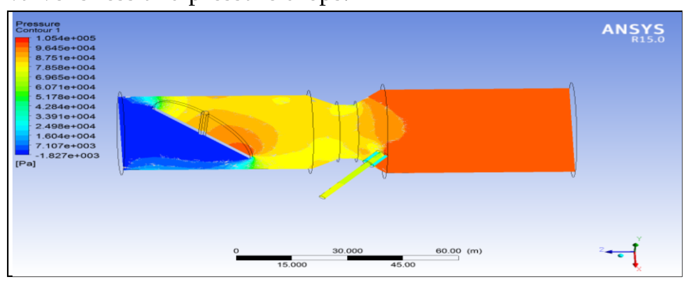


Fig.4.3.1.5.(a) Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 45.

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

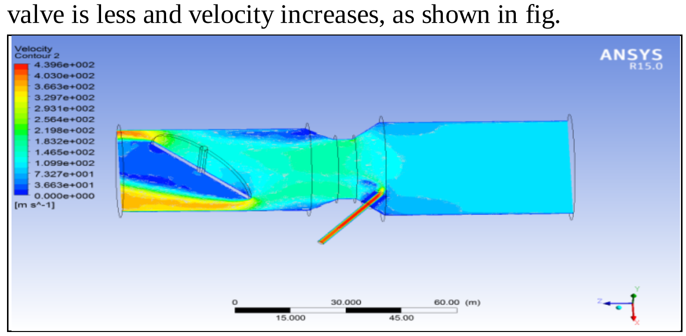


Fig.4.3.1.5.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 45.

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

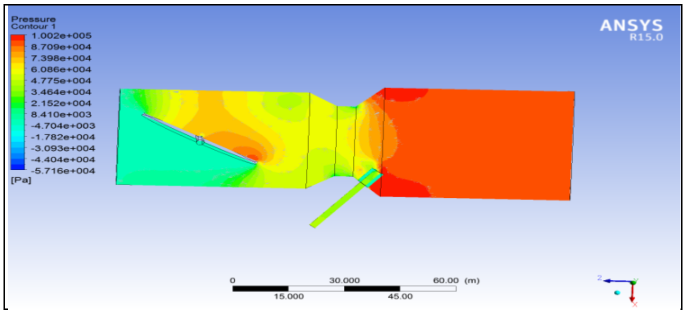


Fig.4.3.1.6.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 60.

At air inlet, pressure is maximum. Fuel after passing through air inlet the pressure drops near throat. As area near throat is minimum, velocity increases. Then fuel near the throttle valve also drops the pressure, as area near throttle valve is less and velocity increases, as shown in fig.

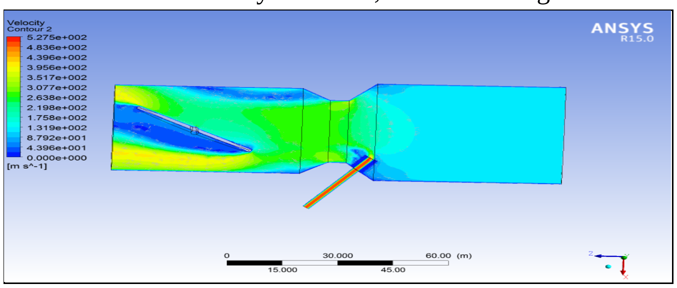


Fig.4.3.1.6.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 60.

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

4.3.2. 125CC CARBURETOR

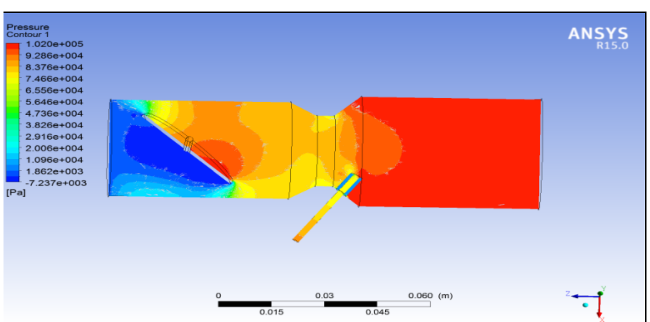


Fig.4.3.2.1.(a) Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 30.

As compared to 100cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, Also power will be maximum, as shown in fig.

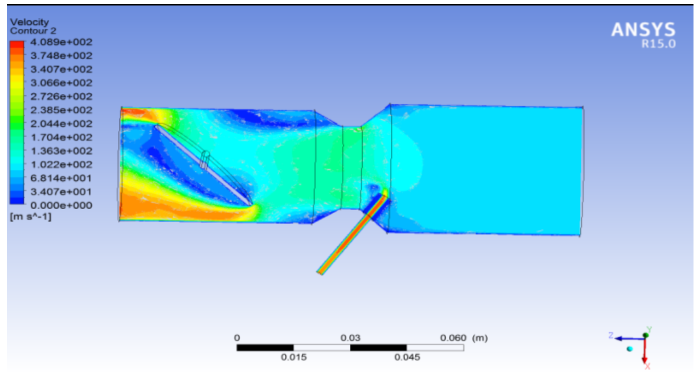


Fig.4.3.2.1.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 30.

As compared to 100cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

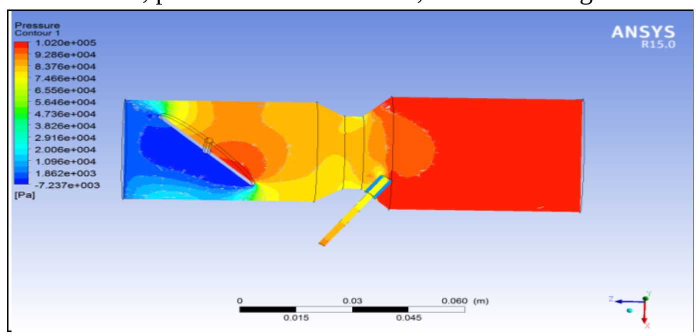


Fig.4.3.2.2.(a) Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 45.

As compared to 100cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

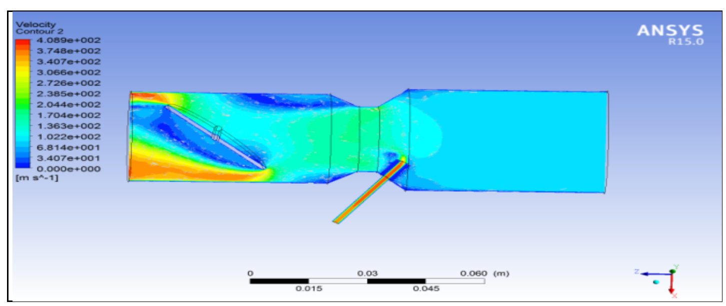


Fig.4.3.2.2.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 45.

As compared to 100cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

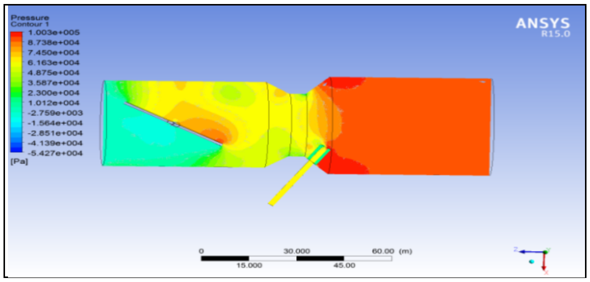


Fig.4.3.2.3.(a). Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 60.

As compared to 100cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

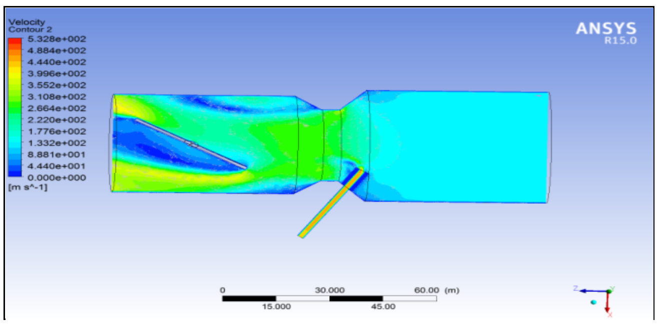


Fig.4.3.2.3.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 60.

As compared to 100cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

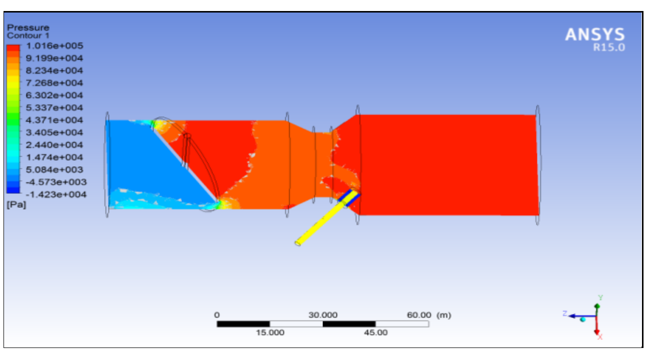


Fig.4.3.2.4.(a) Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 30.

As compared to 100cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

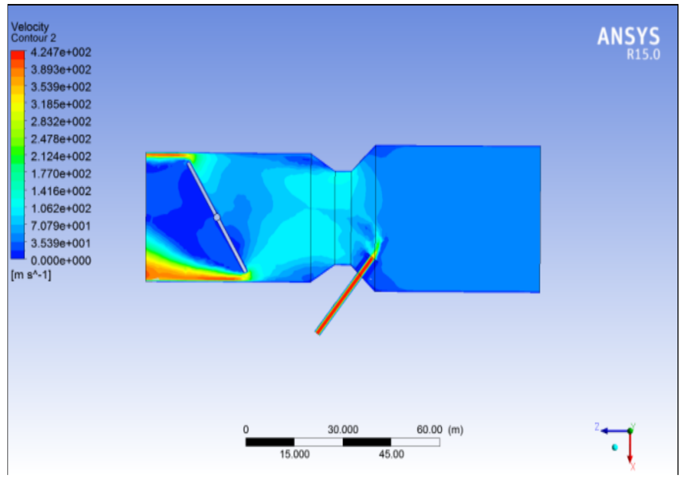


Fig.4.3.2.4.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 30.

As compared to 100 cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

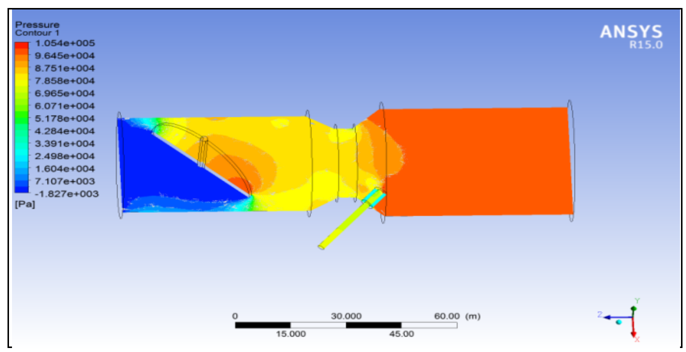


Fig.4.3.2.5.(a) Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 45.

As compared to 100 cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

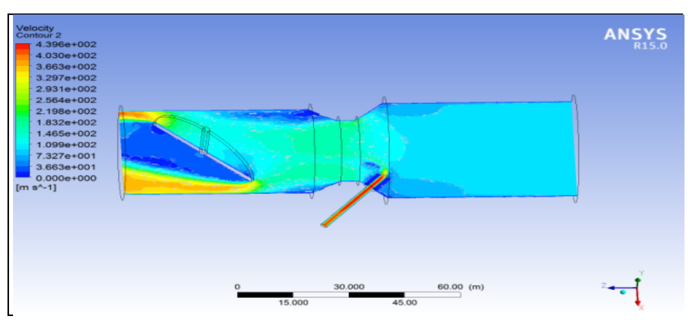


Fig.4.3.2.5.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 45.

As compared to 100 cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

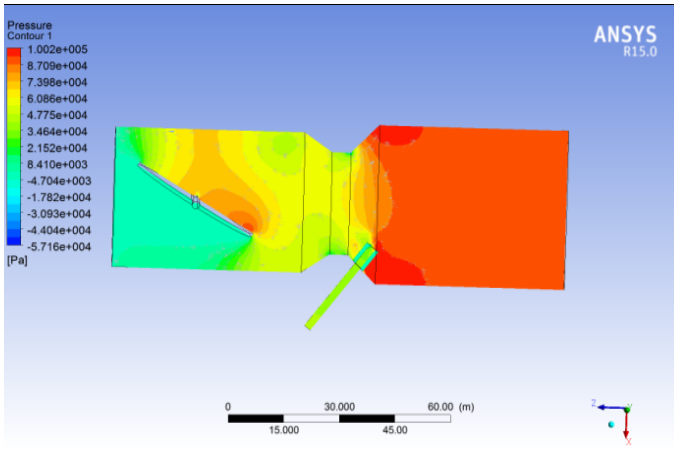


Fig.4.3.2.6.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 60.

As compared to 100 cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

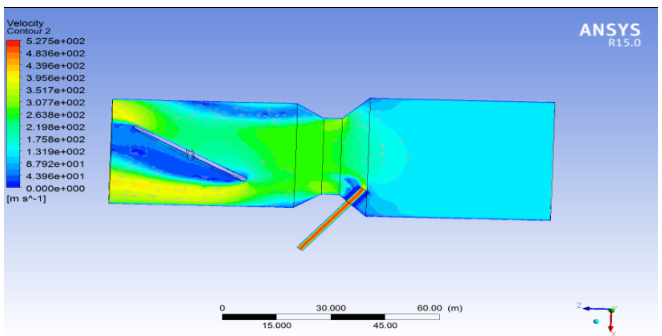


Fig.4.3.2.6.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 60

As compared to 100 cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

6.3.3. 150CC CARBURETOR

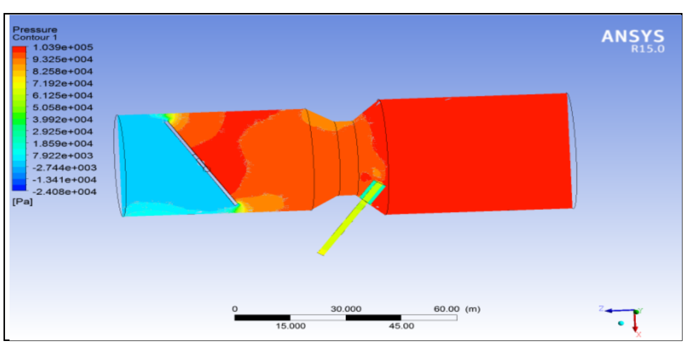


Fig.4.3.3.1.(a) Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 30.

As compared to 100 cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

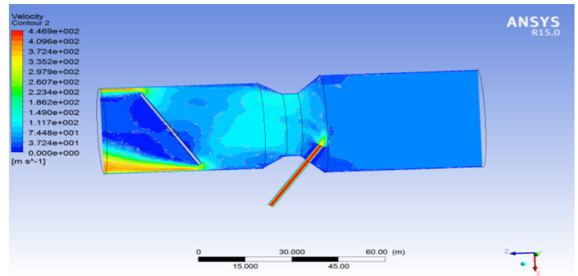


Fig.4.3.3.1.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 30

At air inlet, velocity is minimum. Fuel after passing through air inlet the velocity increases near throat. As area near throat is minimum, pressure decreases. Then fuel near the throttle valve also increases the velocity, as area near throttle valve is less and pressure drops.

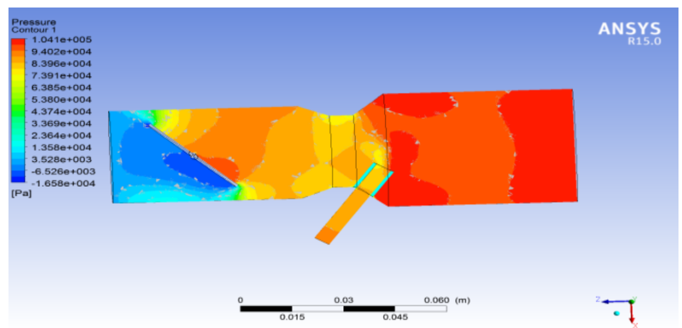


Fig.4.3.3.2.(a). Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 45.

As compared to 100 cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

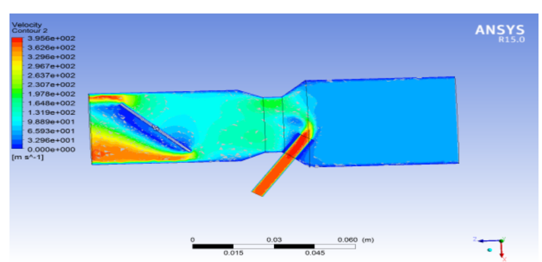


Fig.4.3.3.2.(b). Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 45.

As compared to 100 cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

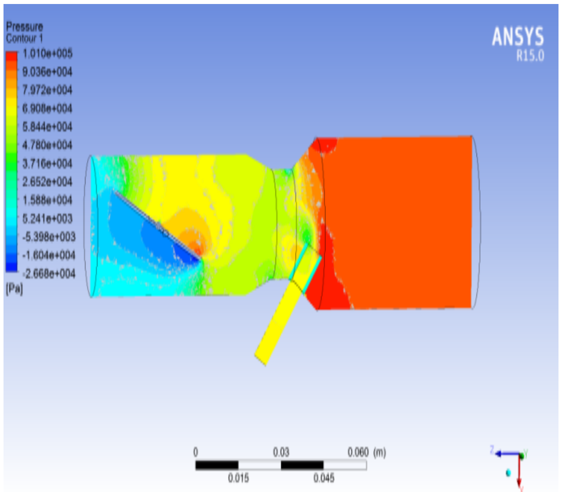


Fig.4.3.3.3.(a). Pressure contour at fuel discharge nozzle angle 34 & throttle valve angle 60.

As compared to 100 cc carburetor maximum velocity will get near throttle plate and more negative outlet pressure is generated, power will be maximum, as shown in fig.

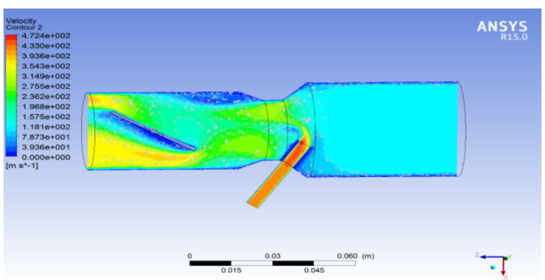


Fig.6.3.3.3.(b) Velocity contour at fuel discharge nozzle angle 34 & throttle valve angle 60.

As compared to 100 cc carburetor more negative outlet pressure will get near throttle plate and more Velocity will be more, power will be maximum, as shown in fig.

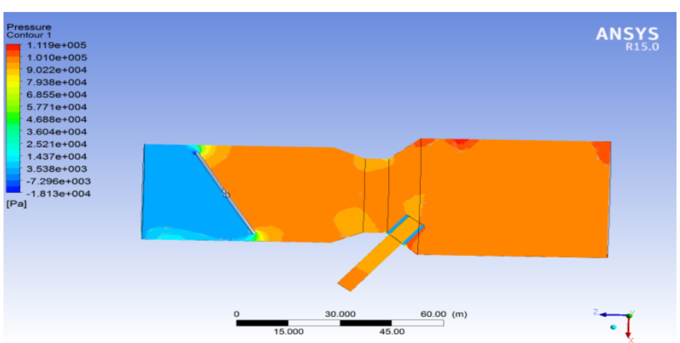


Fig.4.3.3.4.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 30.

As area at inlet is maximum, pressure is maximum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. But here more negative pressure is generated, more fuel will suck in the carburetor.

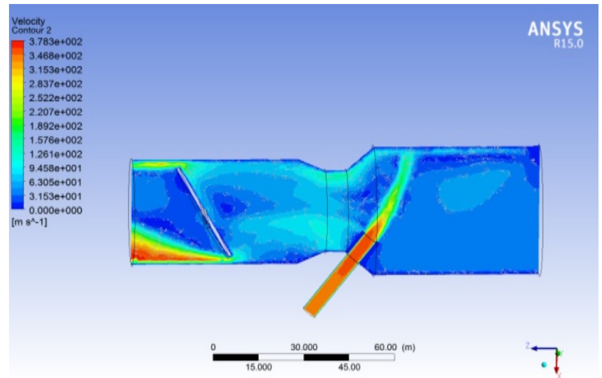


Fig.4.3.3.4.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle30.

As area at inlet is maximum, velocity is minimum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. From above discussion, as a more fuel will be suck in the carburetor, power will be maximum.

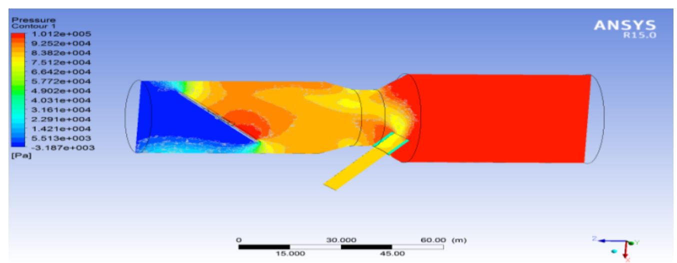


Fig.4.3.3.5.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 45.

As area at inlet is maximum, pressure is maximum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. But here more negative pressure is generated, more fuel will suck in the carburetor

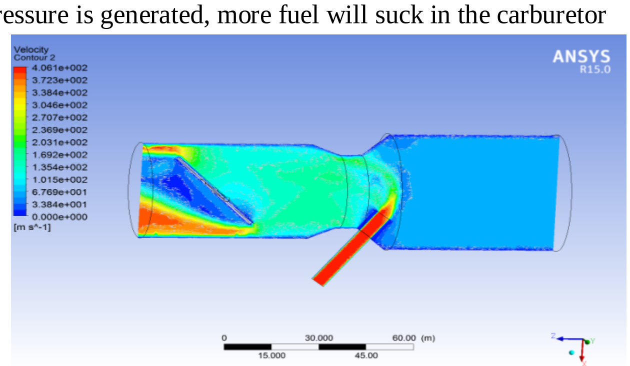


Fig.4.3.3.5.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 45

As area at inlet is maximum, velocity is minimum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. From above discussion, as a more fuel will be suck in the carburetor, power will be maximum.

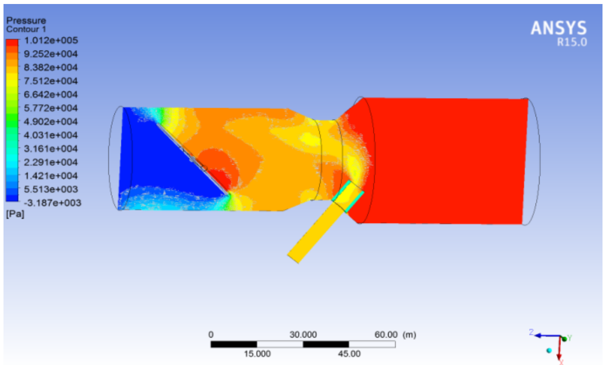


Fig.4.3.3.6.(a). Pressure contour at fuel discharge nozzle angle 38 & throttle valve angle 60.

As area at inlet is maximum, pressure is maximum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. But here more negative pressure is generated, more fuel will suck in the carburettor

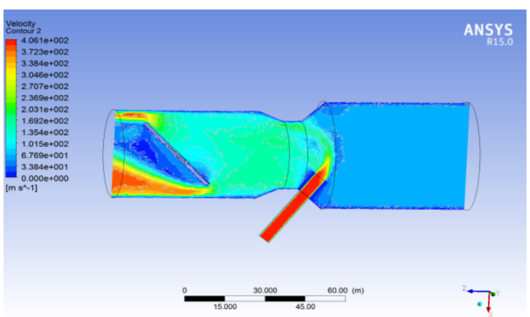
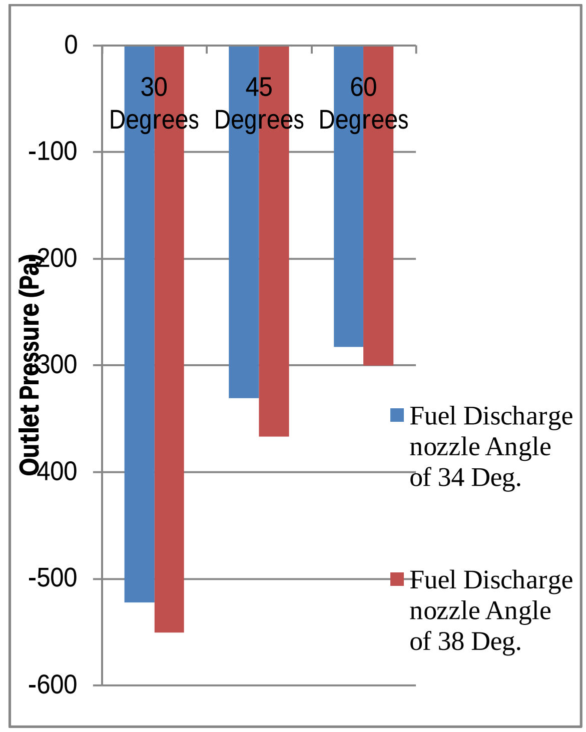


Fig.4.3.3.6.(b). Velocity contour at fuel discharge nozzle angle 38 & throttle valve angle 60.

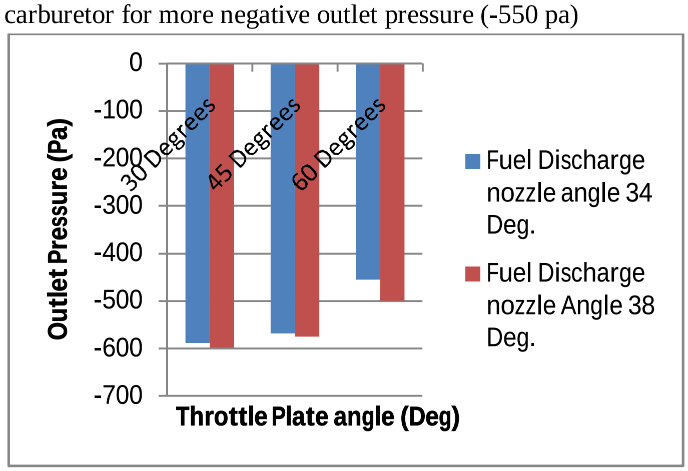
As area at inlet is maximum, velocity is minimum and lower velocity. Area near throttle plate is less, pressure will reduce and velocity is increased. From above discussion, as a more fuel will be suck in the carburetor, power will be maximum.

IV. RESULT AND DISCUSSION



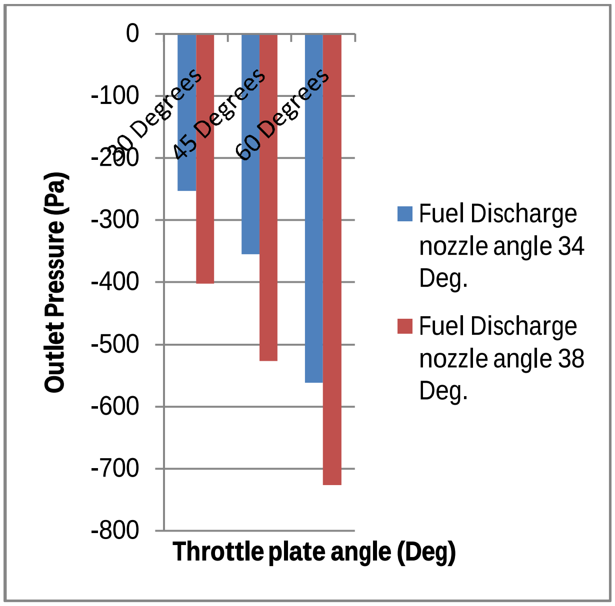
Graph 4.1.Throttle plate angle Vs Outlet pressure of 100cc bike carburetor

In 100cc bike carburetor, the fuel discharge nozzle angle of 34& 38 degree is maximum for 30 degree throttle plate angle than 45& 60 degree throttle plate angle. Also outlet pressure is more negative for 38 fuel discharge nozzle angle as compared to 34 degree fuel discharge nozzle angle in 30 degree throttle plate angle. More fuel will suck inside the carburetor for more negative outlet pressure (-550 pa)



Graph 4.2.Throttle plate angle Vs Outlet pressure of 125cc bike carburetor

In 125cc bike carburetor, the fuel discharge nozzle angle of 34& 38 degree is maximum for 30 degree throttle plate angle than 45& 60 degree throttle plate angle. Also outlet pressure is more negative for 38 fuel discharge nozzle angle as compared to 34 degree fuel discharge nozzle angle in 30 degree throttle plate angle. More fuel will suck inside the carburetor for more negative outlet pressure (-600 pa).



Graph.4.3.Throttle plate angle Vs Outlet pressure of 150cc bike carburetor

In 150cc bike carburetor, the fuel discharge nozzle angle of 34& 38 degree is maximum for 60 degree throttle plate angle than 30 & 45degree throttle plate angle. Also outlet pressure is more negative for 38 fuel discharge nozzle angle as compared to 34 degree fuel discharge nozzle angle in 60 degree throttle plate angle. More fuel will suck inside the carburetor for more negative outlet pressure (-720 pa).

V.CONCLUSION

1. Fuel discharge nozzle angle (FDNA) has great influence on Atomized Vaporization of Air Fuel Mixture in Carburetor Whereas Throttle Valve Angle has comparatively less impact on carburetor Functioning.

2. For 38 deg FDNA it is observed that more negative pressure gets generated at the outlet as compared to FDNA 34 deg. So it will suck more air fuel mixture into the engine resulting in proper and complete combustion of charge thereby generating more power

REFERENCES

[1] Ganesha T.CFD analysis on a simple carburetor for different positions of throttle valve and fuel nozzle valve” VTU, PG-Center, Mysore(India), Issue No. 07, July 2014

[2] Diego A. Arias. Numerical and experimental study of air and fuel flow in small engine carburetors. University of Wisconsin-Madison, 2005

[3] Heywood John B. Internal combustion engines fundamentals. Mc Graw Hill, Inc, 1988.

[4] Diego A. Arias, Timothy A. Shedd, Steady and non-steady flow in a simple carburetor, Inst Mech Eng (Lond) Proc , Volume 188, Issue 53, 1974, Pages 537-548.

[5] Fluent. FLUENT 5 User Guide, 1999.

[6] Gambit. GAMBIT 5 User Guide, 1999.

[7] Versteerg H.K. and Malalasekera W.“an introduction of computational fluid dynamics.

[8] J.B. Heywood, Internal Combustion Engine Fundamentals. McGraw-Hill, Singapore, 1988.

[9] Sayma Abdulnaser, Computational Fluid Dynamics, ISBN- 978-87-7681-938-4, Abdulnaser Sayma and Ventus Publishers, 2009

[10] D.L. Harrington. Analysis and digital simulation of carburetor metering. PhD thesis, University of Michigan, 1968.