NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 2, ISSUE 9, SEP.-2015

PORT FLOW SIMULATION OF AN IC ENGINE

Himanth Kumar H Y,
PG Student
Department of Mechanical Engineering,
Vidyavardhaka College of Engineering, Mysuru, India

Jayashankar N
Associate professor,
Mechanical department,
Vidyavardhaka College of Engineering, Mysuru, India

ABSTRACT

This paper deals with the numerical analysis of 3d model which has inlet port diameter 46mm, valve diameter 43mm and the length and diameter of the cylinder is 562mm and 93.65mm respectively which is developed to study the effect of valve lift on the flow of fluid inside the cylinder. For different valve lifts velocity will change inside the cylinder. Results of CFD simulation indicated that valve lift affects velocity flow field inside the cylinder. It also proved that CFD is a convenient tool for designing and optimizing the flow field in the engine.

KEYWORDS-Computational fluid dynamics; intake manifold; inlet port; Numerical simulation;

INTRODUCTION

The internal combustion engine usually consists of four cycles such as intake, compression, expansion (including combustion) and exhaust. Out of these four processes intake and compression strokes are most important as these strokes influences the pattern of air flow coming inside cylinder during intake stroke and it generates the condition needed for the fuel injection during compression stroke. For maximizing the air passing into the cylinder manifold design has to be optimized. Hence design of intake manifold is more important in engine design. Experimental methods would cost time and money to achieve this. Using CFD tool without undergoing any time consuming experiment engine efficiency can be estimated.

Short discussions of the previous studies of authors are presented here. Chen et.al [1] gives result of fluid flow through an IC engine inlet port both in experimental and computational mode. The port was straight generic inlet port. They used CFD tool to analyze the velocity contours near port region. They have used LDA and LSFV technique to analyze fluid flow. Both experimental and prediction results shows that the valve lift that affect the flow Robert et.al [2] studied the two common motions in an automobile engine they are tumble and swirl motion using visualization techniques. Benny et.al [3] investigates the effect of helical, spiral manifold configuration on air motion in a diesel engine. Using CFD tool flow patterns of these different manifold configurations are studied. The CFD results of swirl velocity inside combustion chamber are compared with experimental results.

Prasad.et.al [4] investigates the swirl air created due to injecting air into the manifold. To direct air flow into the inlet manifold they have grooved the inlet manifold with helical groove to create turbulence in the manifold. The tests are conducted by varying dimensions of groove and speed was maintained constant.

The main objective of this project is to analyze the velocity characteristics of fluid flow through the intake manifold, inlet port and valve to get optimized swirl and tumble motion, so that one can enhances the performance of the engine by improving combustion rate.

II MATHEMATICAL FORMULATION

2.1 The physical model and coordinate system

A 3D model of a inlet port and valve with cylinder is shown in fig is created in ANSYS FLUENT (Ver. 14.5).

Table 2.1 Dimensions of various parts of model as shown in Fig 2.1

Diameter of inlet and outlet port	46 & 39.5 mm
Diameter of the valve	43 mm
Diameter of the cylinder	93.65 mm
Length of the cylinder	562 mm
Valve lift	10 mm

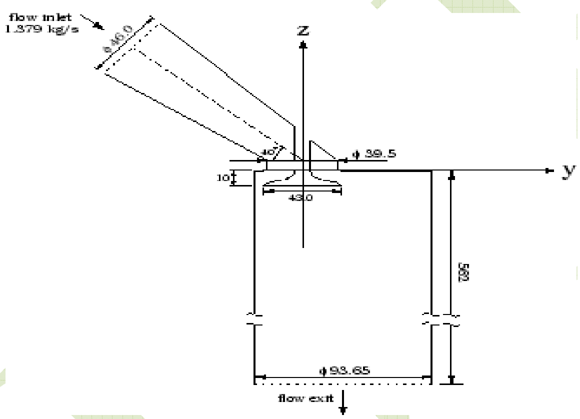
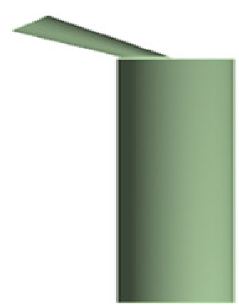


Fig 2.1 Theoretical model



Geometry model created in fluent

MESHING

- ✓ The model is created using ANSYS FLUENT software.
- ✓ In-cylinder problems solved in Fluent consist of three stages. The first stage is to decompose the geometry into different zones and mesh them properly.
- ✓ The second stage is to set up the engine case inside Fluent with the help of a setup journal.
- ✓ The third stage is to perform a transient in-cylinder simulation.

- ✓ First, the surface is meshed with triangular element. In order to resolve the turbulent boundary layer on the solid surfaces, it is best to have growing prismatic cells from the Valve surfaces. Finally the remaining region in the domain is filled with tetrahedral cells. No of elements is used for mesh generation is 11 to 12 lakhs approximately.

- ✓ The domain has been subdivided into growing boxes to make it easier to generate the grid. The choice for the elements has been both tetrahedral and hexahedral mesh volumes.

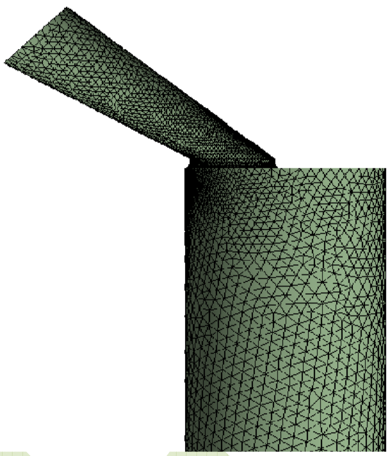


Fig 3.1 Meshed model in FLUENT

GOVERNING EQUATIONS

There are mainly three equations we solve in computational fluid dynamics problem. They are Continuity equation, Momentum equation (Navier Stokes equation) and Energy equation.

4.1.1 Continuity Equation

A continuity equation expresses a conservation law by *"Equating* a net flux over a surface with a loss or gain of material within the surface". Continuity equations often can be expressed in either integral or differential form as shown below.

$$\int_{cs} \rho V\, dA + \frac{\partial}{\partial t}\int_{cv} \rho dA = 0$$

This is a statement of the principle of mass conservation for a steady, one-dimensional flow, with one inlet and one outlet.

$$\nabla(\rho V) + \frac{\partial \rho}{\partial t} = 0$$

$$\text{Where, } \nabla = \frac{\partial}{\partial x}\hat{i} + \frac{\partial}{\partial y}\hat{j} + \frac{\partial}{\partial z}\hat{k}$$

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho.u)}{\partial x} + \frac{\partial(\rho.v)}{\partial y} + \frac{\partial(\rho.w)}{\partial z} = 0$$

4.1.2 Momentum (Navier Stokes) Equations

The momentum equation is a statement of Newton's Second Law and relates the sum of the forces acting on an element of fluid to its acceleration or rate of change of momentum. The Newton's second law of motion F = ma forms the basis of the momentum equation.

$$\rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} + \rho w\frac{\partial u}{\partial z} = \rho g_x - \frac{\partial p}{\partial x} + \mu\frac{\partial^2 u}{\partial x^2} + \mu\frac{\partial^2 u}{\partial y^2} + \mu\frac{\partial^2 u}{\partial z^2}$$

$$\rho\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} + \rho w\frac{\partial v}{\partial z} = \rho g_y - \frac{\partial p}{\partial y} + \mu\frac{\partial^2 v}{\partial x^2} + \mu\frac{\partial^2 v}{\partial y^2} + \mu\frac{\partial^2 v}{\partial z^2}$$

$$\rho\frac{\partial w}{\partial t} + \rho u\frac{\partial w}{\partial x} + \rho v\frac{\partial w}{\partial y} + \rho w\frac{\partial w}{\partial z} = \rho g_z - \frac{\partial p}{\partial z} + \mu\frac{\partial^2 w}{\partial x^2} + \mu\frac{\partial^2 w}{\partial y^2} + \mu\frac{\partial^2 w}{\partial z^2}$$

4.1.3 Energy Equation

This equation demonstrates that, per unit volume, the change in energy of the fluid moving through a control volume is equal to the rate of heat transferred into the control volume plus the rate of work done by surface forces plus the rate of work done by gravity

$$\frac{\partial}{\partial t}\left(\rho e + \frac{1}{2}\rho v^2\right) + \frac{\partial}{\partial x}\left(\rho u e + \frac{1}{2}\rho u v^2\right) + \frac{\partial}{\partial y}\left(\rho v e + \frac{1}{2}\rho v v^2\right) + \frac{\partial}{\partial z}\left(\rho w e + \frac{1}{2}\rho w v^2\right) =$$

$$k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial xz^2}\right) - \left(u\frac{\partial p}{\partial x} + v\frac{\partial p}{\partial y} + w\frac{\partial p}{\partial z}\right) +$$

$$\mu\left[u\frac{\partial^2 u}{\partial x^2} + \frac{\partial}{\partial x}\left(v\frac{\partial v}{\partial x} + w\frac{\partial w}{\partial x}\right) + v\frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial y}\left(u\frac{\partial u}{\partial y} + w\frac{\partial w}{\partial y}\right) + w\frac{\partial^2 u}{\partial z^2} + \frac{\partial}{\partial z}\left(u\frac{\partial u}{\partial z} + v\frac{\partial v}{\partial z}\right)\right]$$

$$+2\mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial u}{\partial y}\frac{\partial v}{\partial x} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial v}{\partial z}\frac{\partial w}{\partial y} + \frac{\partial^2 w}{\partial z^2} + \frac{\partial w}{\partial x}\frac{\partial u}{\partial z}\right] + \rho u g_x + \rho v g_y + \rho w g_z$$

BOUNDARY CONDITIONS

In this simulation Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known.

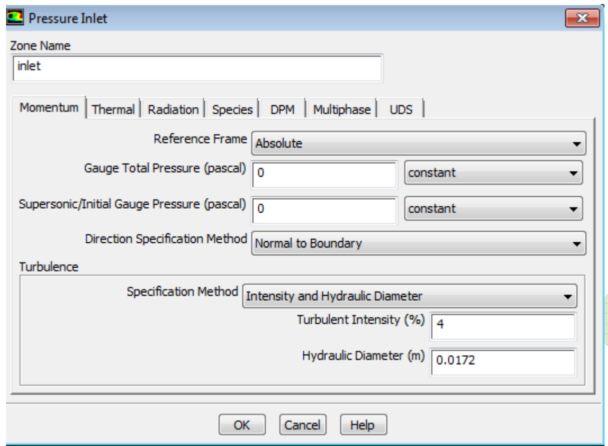


Figure.5. 1 Pressure inlet boundary conditions windows

VALIDATION

The problem is solved using ANSYS FLUENT and the code is validated with the results of a research paper [1], it is found that it gives good agreement with the experimental results of research paper [1] as shown in following graphs.

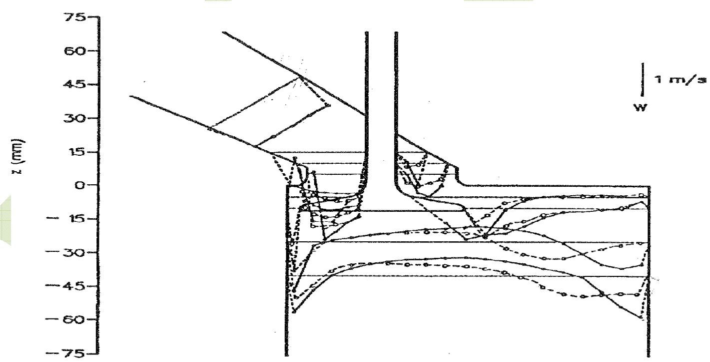


Fig.6.1 Comparison of predicted and experimental results for valve lift 10mm performed in previous study

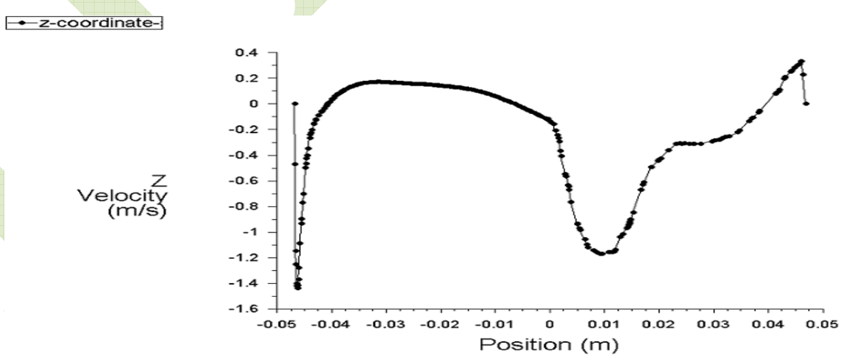


Fig 6.2.1 Prediction of velocity in z=-15mm for valve lift 10mm in present study

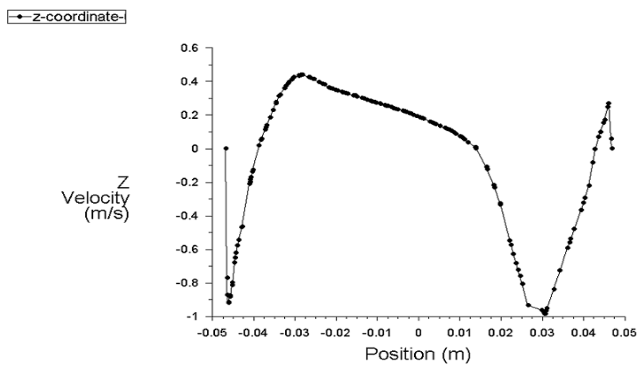


Fig.6.2.2 Prediction of velocity in z=-30mm for valve lift 10mm in present study

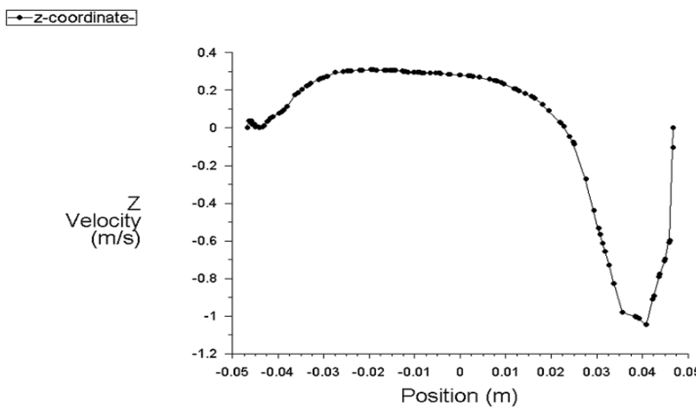
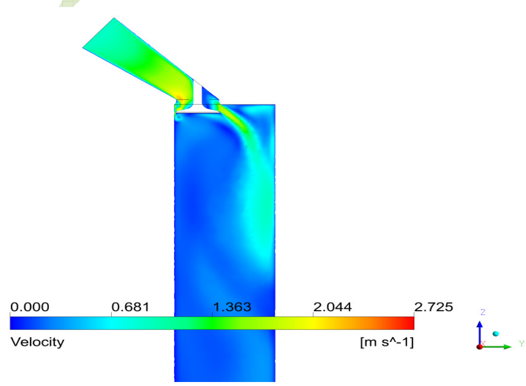


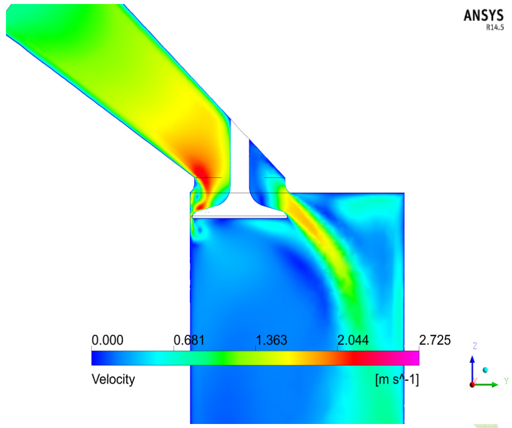
Fig.6.2.3 Prediction of velocity in z=-45mm for valve lift 10mm in present study

It may see from the figures results obtained from the FLUENT shows good agreement with previous study except in the critical region of valve.

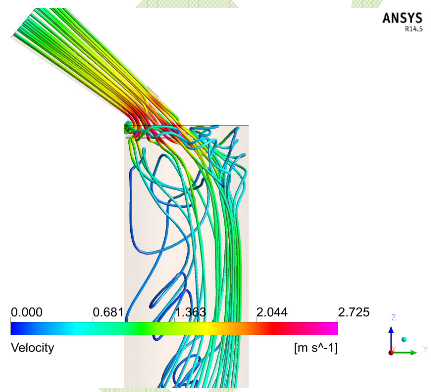
Animated results of FLUENT for 10mm valve lift obtained in present study



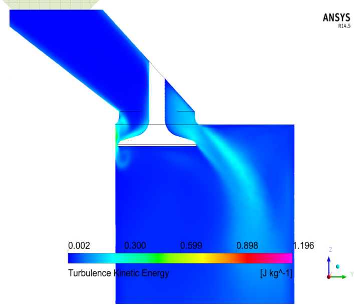
Velocity profile for 10mm valve lift



Velocity profile for 10mm valve lift

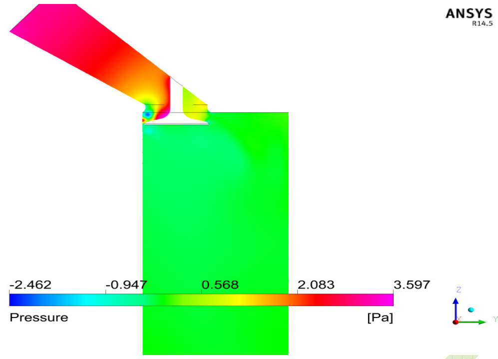


Velocity streamline for 10mm valve lift

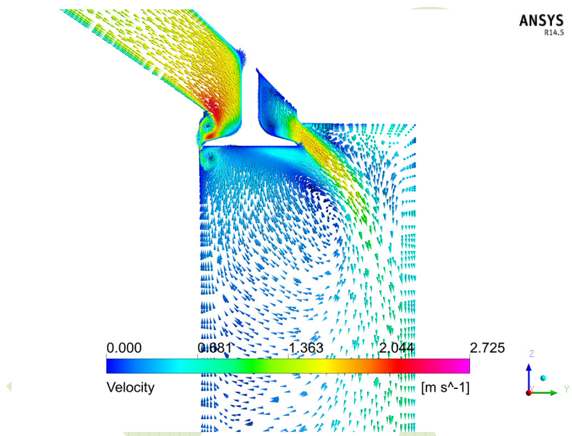


Turbulent kinetic energy profile for 10mm valve lift

NOVATEUR PUBLICATIONS
INTERNATIONAL JOURNAL OF INNOVATIONS IN ENGINEERING RESEARCH AND TECHNOLOGY [IJIERT]
ISSN: 2394-3696
VOLUME 2, ISSUE 9, SEP.-2015



Pressure profile for 10mm valve lift



Velocity contour profile for 10mm valve lift

CONCLUSIONS

To make combustion to happen air flow inside intake port plays a vital role. To get good engine efficiency with lower emission we need to study the flow inside the intake port, manifold through the valve. In this thesis we understand the flow characteristics through port inside the engine with the help of CFD tool FLUENT.

CFD helps to understand the flow field inside the cylinder and also gives depth knowledge of flow in graphical mode where measurements are not easy to make.

1. Three-dimensional modeling of intake manifold, port and valve are key factors to predict the velocity contours inside the engine cylinder. From this thesis we can conclude that valve lift affect the flow except region near the port bend upstream.
2. As the valve lift increases flow separation becomes critical because as valve lift increases losses near the valve increases.

3. The results obtained from prediction shows good agreement with the experimental results except some critical region near the valve.

REFERENCES

1. Chen, A., Lee, K.C., Yianneskis, M., and Ganti, G., *Velocity Charac- teristics of Steady Flow Through a Straight Generic Inlet Port, Interna- tional Journal for Numerical Methods in Fluids, 21:571,590, 1995*

2. Robert S. Laramee , Daniel Weiskopf , Jürgen Schneider , Avl Graz , Helwig Hauser, *Investigating Swirl and Tumble Flow with a Comparison of Visualization Techniques, In Proceedings IEEE Visualization, 2004.*

3. S. L. V. Prasad, V. Pandurangadu, V. V. Prathibha bharathi and V. V. Naga deepthi*, Experimental study of the effect of air swirl in Intake manifold on diesel engine performance*

4. N.Kumar.et.al "*Studies on variable swirl intake system for DI diesel engine using CFD", ISSN 0973-4562 volume 2, number 3 (2007)*

5. Soonseong Hong, Byungkeunoh and Yongtae Kim, "*Optimization of intake manifold design using design for six sigma, SAE, 2007-01-3534*

6. Mike Dong, Grant Chen, Min Xu and Chao Daniels, "*P*