Performance Analysis of Diesel Engine Inlet Port with Variable Angles

K. Sri Harsha[1], V.CVS.Phaneendra[2]

[1]M.Tech student, Thermal Engineering, Department of Mechanical Engineering, Guntur Engineering College, Yanamadala, Guntur (D.T), A.P
[2]Associate Professor, Thermal Engineering, Department of Mechanical Engineering, Guntur Engineering College, Yanamadala, Guntur (D.T), A.P

---***---

Abstract - Remarkable developments of engines have been esteemed as alternative technologies in the area of road transportation for decades. The technology of engine has been improved from 2 valve systems with gravity-fed combustion to modern fuel injection system. Usually engines with fuel consumption and high duty efficiency are widely used. Many attempts have been done to improve efficiency and to minimize the consumption for achieving an ideal engine.

It is well known that the intake manifold is the ultimate part of engine's air induction system which serves a chief role in managing the air supply to cylinder. These intake ports can be spotted in all engines as they have influence on the information of air/fuel mixture. Various researches have been performed on the ports as its design is important for motion of charge and it impacts engine's breathing capacity. In large cylinder engines the trouble is to obtain on actual blend of air fuel mixture and its uniform spread from head to block through the port. The project is predominantly focused on enhancing this spread into block which is determined by the helix angle at inlet port.

The Port Simulation in project involves CFD analysis on ports with different helix angle to boost the spread. The efficiency of 6 cylinder engine was improved by Redesign of port which is called helical port. Because of this, swirl motion of air occurs inside the cylinder that results in the improvement of both relative velocity between air and fuel and efficiency with the help of abundant mixture from combustion. NX-CAD software is used for the modeling of the port. CFD analysis was performed in ANSYS software by making use of Flotran 142 element in a module to understand the flow characteristics.

Key words: Inlet Port, Helix Angle, NX-CAD, ANSYS

1. INTRODUCTION

Intake ports are the final part of an engine's air induction system. They connect the intake manifold with the combustion chamber and are opened and closed with the intake valves. While intake ports are found in all types of engines, they have an especially pronounced influence on the air/fuel mixture formation in gasoline (SI) engines. In Diesel engines, the piston bowl also helps with that task. Furthermore, the port shape is responsible for the charge motion, where favorably shaped vortices reduce energy dissipation, and it influences the amount of air that gets into the combustion chamber, where an increase leads to higher engine performance.

In the present work Six Cylinder Hino Engine was considered. The combustion where take place at the inside of the cylinder. Water is used as a coolant. This Six Cylinder Hino engine is mainly used in Trucks / Trailers for moving Cargo containers from shipyard to the customer place. This Six Cylinder engine is used in CHEETAH-231 a truck manufactured by M/s. ASHOK LEYLAND, Ennore.

The efficiency of the engine is dependent upon the complete mixing of air and fuel inside the cylinder. This complete mixing can be achieved by creating relative velocity between the fuel and air inside the cylinder. For that we have to give turbulence motion to the air inside the cylinder.

1.1 Introduction to Turbulence

In order that the atomized fuel injected into the combustion chamber of a compression ignition engine may be burnt efficiently there should be a high relative velocity between the air and fuel so that a thorough mixing takes place. This is achieved by turbulence which implies the violent eddying motion or swirling of air. Since the fuel and oxygen molecules are brought into intimate contact, it is possible to burn weak mixtures more satisfactorily than if there were no turbulence. It produces a quick propagation of flame and thus accelerates the combustion process. The cylinder head, inlet ports and the top of the piston head can be so designed as to give a certain amount of turbulence. In compression ignition engines care should be taken to inject the fuel across and not in the same direction as the air stream so that fresh oxygen is brought into contact with the particles of fuel. The turbulence induced during compression and during charging of fuel due to the caution of the spray itself is called the primary turbulence while the term secondary turbulence is taken to mean the further increase turbulence by the burning or exploding nature of the process after the combustion has been once initiated. Turbulence also decreases the delay period provided that temperature of air is maintained. Too much turbulence should be avoided firstly because it results in greater heat loss to the cylinder walls and thus lowers the efficiency and secondly it produces harsh running and prevents complete combustion. All over that port redesign gives the excellent way of achieving the swirled motion of air inside the cylinder. This will give the thorough mixing of air & fuel and increase efficiency. The Port wall thickness get an important role for reducing the weight of the engine corresponding to that the cooling flow rare can be modulated.

1.2 Swirl Ratio

Controlling the swirl ratio according to the engine operating parameters is necessary to reduce exhaust emissions and improve the power output of small, high-speed direct-injection diesel engines that operate over a wide range of conditions. Additionally, a new combustion system called the Modulated Kinetics (MK) concept, which dramatically improves the emission performance of direct-injection diesel engines, requires an intake swirl ratio of 10 or greater in the part-load region. To meet these requirements, an attempt was made in this research to develop a variable swirl intake port system capable of controlling the swirl ratio over a wide range. This system combines two mutually independent intake ports, one of which incorporates a swirl control valve that controls the swirl ratio by varying the flow rate. To investigate the performance of the intake port system, steady state flow tests were conducted in parallel with three-dimensional computations. Using these applications, the swirl flow characteristics of this intake port system were analyzed over a wide range of operating conditions.

1.3 Configuration of variable swirl intake port

Following figure shows the basic configuration of the variable swirl intake port system examined in this research for application to small, high-speed, 4 valves/cylinder direct injection (HSDI) diesel engines. The system consists of two independent intake ports, one for generating a high swirl ratio and the other for generating a low swirl ratio. A flow control valve is installed at the inlet to the low-swirl port. Closing this valve results in a high swirl ratio and opening it produces a low swirl ratio. This means that the maximum controllable swirl ratio is dependent on the High-swirl port, and the minimum swirl ratio is the combined swirl obtained when the two ports are used together.

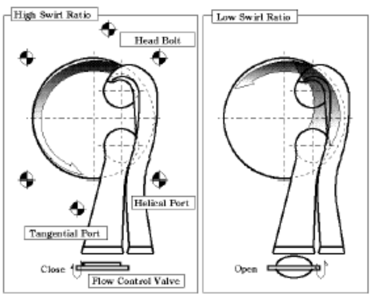

Fig 1.1: Configuration of Variable Intake Port System.

2. LITERATURE REVIEW

Dr.Stanley K. Widener (1995) has done the design of helical intake ports for swirl generation is a process that has been developed over a number of years through primarily empirical methods. A number of design rules have been established that enable designers to develop ports that approach the state-of-the-art for maximum swirl generation with minimum pressure loss. More recently, computer-aided design (CAD) tools have been introduced that permit geometry and features to be accurately defined by mathematical surface descriptions, and to be parameterized such that derived geometry is updated automatically along with parent features.

A set of experiments has been performed by **William Church and P.V.Farrell (1998)** which were aimed at quantifying the effects of specific intake port geometry changes on large scale in-cylinder motions. Using a modular engine with replaceable intake port blocks, 3 different intake ports were used and results obtained at three crank angles: BDC, 90% before TDC of compression, and TDC of compression. For each port, in-cylinder flows were quantified using a pulsed laser and high-speed imager. The resulting images were analyzed using a particle –tracking scheme.

A variable swirl intake port system for four valve–cylinder direct injection diesel engines were developed by **Jun-ichi kawashima, Hiroshi Ogawa and yoshiyuki Tsuruhave (1998**). This system combines two mutually independent ports, one of which is a helical port for generating a ultra – high swirl ratio and the other is a tangential port for generating a low swirl ratio. The tangential port incorporates a swirl control valve that controls the swirl ratio by varying the flow rate.

Multi-dimensional modeling of the flow and combustion promises to become a useful optimization tool for IC engine design. Currently the total simulation time for an engine cycle is measured in weeks to months, thus preventing the routine use of CFD in the design process. **H.Jasak, J.Y.Luo, B.KaZludercic & A.D.Gosman (1999)** described three tools aimed at reducing the simulation time to less than a week. The rapid template based mesher produces the computational mesh within one to two days. The parallel flow solver STAR – CD performs the flow simulation on a similar time-scale. The package is completed with COVICE MP, a parallel post-processor which allows real – time interaction with the data.

We are also going to perform a similar simulation using FLOTRAN CFD. Here the time taken for simulation is drastically reduced due to high end solvers and high speed processor.

3. MODELING AND ANALYSIS OF INTAKE PORT

The main objective of the project is to improve the efficiency of the engine by re-design of the inlet port. Actually the engine has a straight port. It is located at the cylinder head. The air coming out of the straight port to cylinder is laminar. The vector velocity of a air particle is straight downward direction.

During the fuel injection the fuel particle also pass straight & downward direction. Because of that there is no relative velocity between the air & fuel inside the cylinder. For that creating turbulence (or) swirl of air the port will be modified into helical port.

Initially the existing straight port will be modeled by using the software some CAD package like NX-CAD. Because modeling such complex profile a very high end complex surface generating software will be required. Hence the CAD system used will be NX-CAD. The flow through the inlet port will be performed by CFD flow analysis using Flotran 142 element it is the module in ANSYS Software.

Similarly the modified inlet port or helical port can be modeled in NX-CAD and the flow can be analyzed by using CFD analysis.

3.1 Modeling of Intake Port

Initially the inlet straight port will be modeled in using CAD Package. In that advance NX-CAD software is more suitable for creating port than other software. During Modeling of the port in NX-CAD before that I have clearly understand the geometric nature of the port diagram. The port will be having different cross section along its path

The Path will be taken as a Trajectory. It is a 3D curve drawn by two projection method using curve option. In the two projection method the front view of the 3D curve can be drawn on the front plane and top view of the 3D curve can be drawn on the Top plane finally getting the 3D curve. Along the 3D curve the points will be created wherever we need the different cross section using offset coordinate system in datum points option from the datum features.

The modeling and analysis of intake port will be done in two stages

3.1.1 Phases of Modeling

Phase - One

- The existing straight inlet port will be modeled in NX-CAD.
- CFD Analysis will be performed on the existing port to simulate the flow in to the block.

Phase-Two

- Redesign of existing straight port to helical port.
- CFD Analysis will be performed on helical port to find spread into the Engine Block.
- Port wall will be modeling in NX-CAD

3.2 Analysis of intake port

Already the port will be modeled in NX-CAD. This port will be analyzed by importing this model in to the analyzed software such as ANSYS. Before transmission the model in NX-CAD is getting exported to the **IGES** file in the form of solid and then imported in to the ANSYS Software. After importing the following steps are be taken:

1. Imported solid may not be completely able to import as solid. Hence we will be doing geometric repair to rectify the model.
2. Define element type as a CFD Flotran 142 (3D Elements)
3. In CFD Flotran setup [all the material properties is defined].
4. Meshing the port model using automatic meshing tool which is in the pre processor.
5. Applying boundary conditions which are available in the main menu.
6. Normally for the laminar flow the velocity at the solid will be zero and the negative pressure will be applied at the exit of port both as a boundary condition.
7. The boundary condition will be applied to the inlet & outlet surface.
8. The Flow analysis is a non linear analysis and the problem is solved in cumulative iteration steps.

4. RESULTS OF INTAKE PORTS

4.1 Results of Straight Intake Port

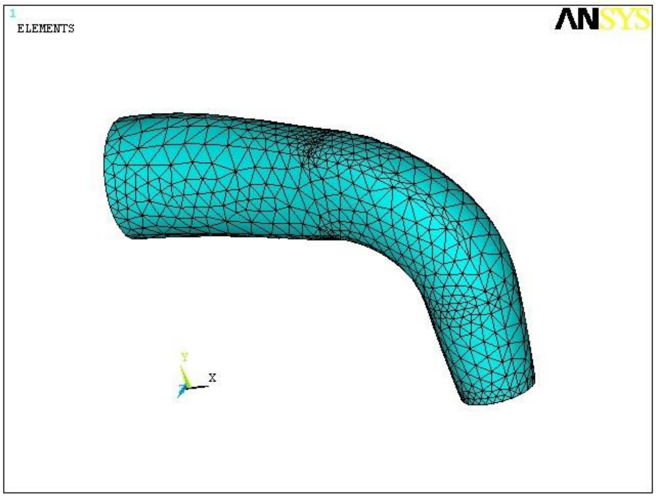

Fig 4.1: Meshed model of straight intake port

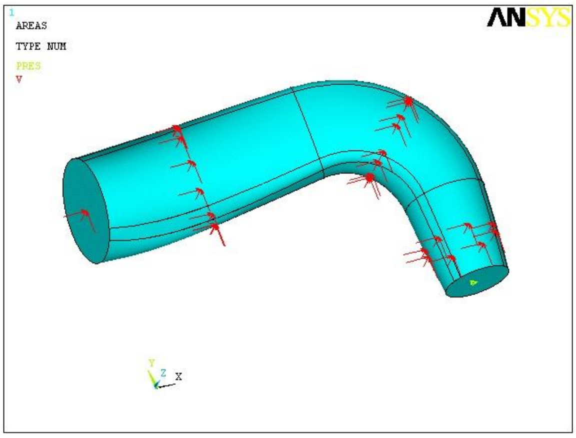

Fig 4.2: Intake port with boundary conditions

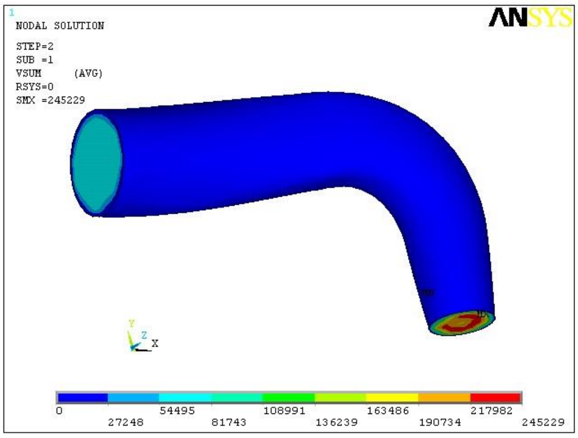

Fig 4.3: velocity distribution

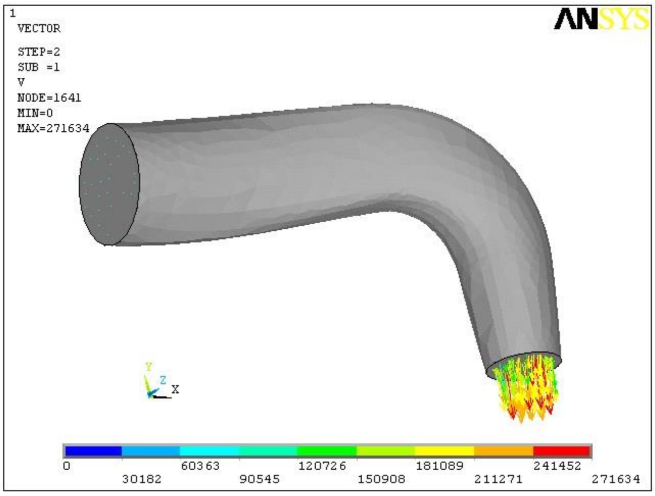

Fig 4.4: Velocity distribution in the port

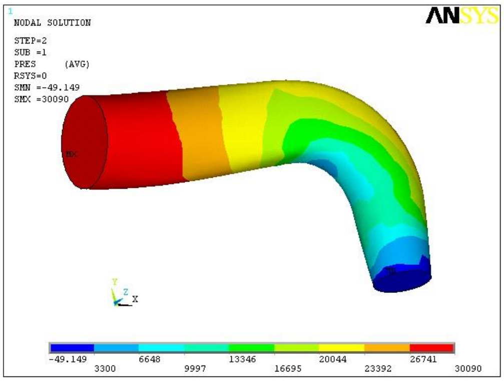

Fig 4.5: pressure distribution in straight intake port

4.2 Results of intake port with 160^0 helix angle

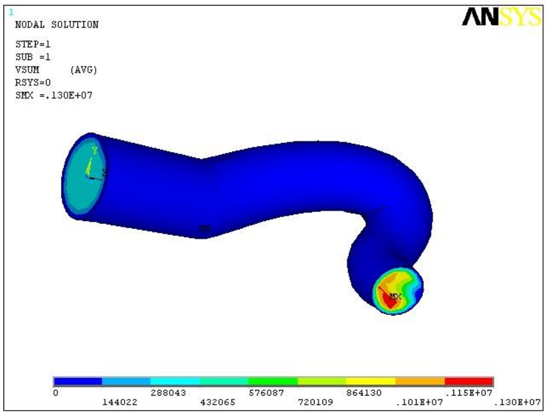

Fig 4.6: velocity distribution in intake port with 160^0 angle

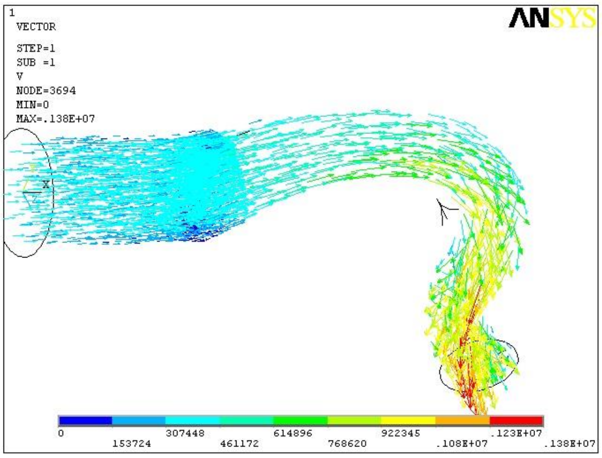

Fig 4.7: velocity stream line in intake port with 160^0 angle

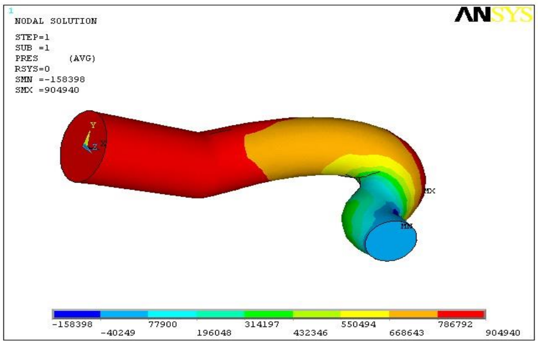

Fig 4.8: pressure distribution in intake port with 160^0 angle

4.3 Results of intake port with 220^0 helix angle

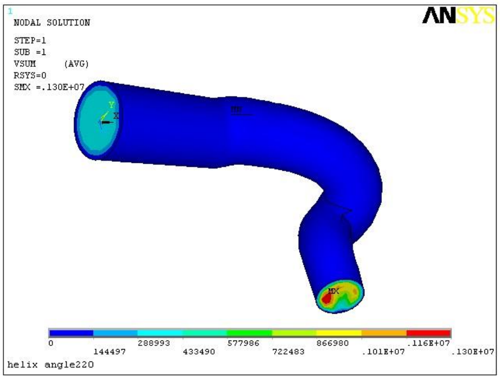

Fig 4.9: velocity distribution in intake port with 220^0 angle

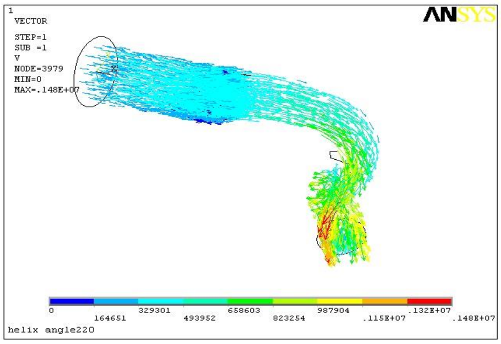

Fig 4.10: velocity stream line in intake port with 220^0 angle

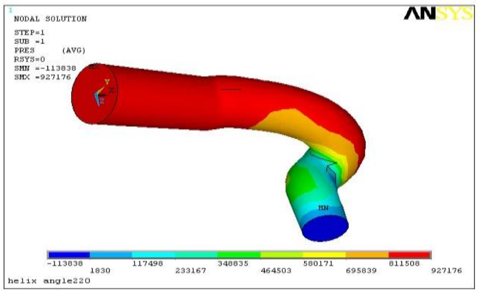

Fig 4.11: pressure distribution in intake port with 220^0 helix angle

4.4 Results of intake port with 300^0 helix angle

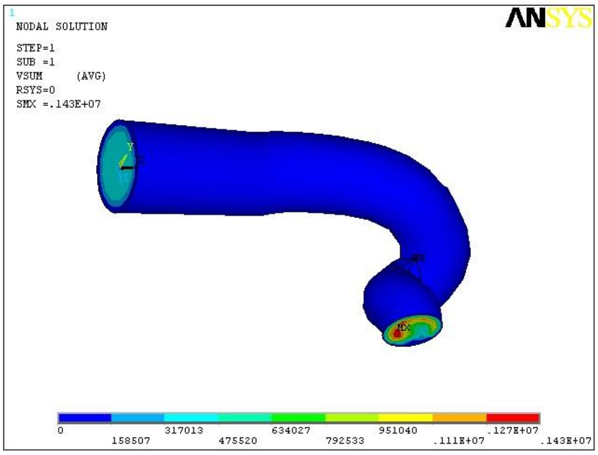

Fig 4.12: velocity distribution in intake port with 300^0 helix angle

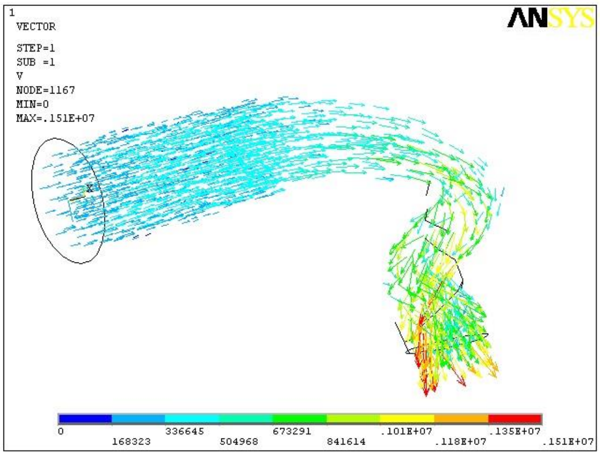

Fig 4.13: velocity stream line in intake port with 300^0 helix angle

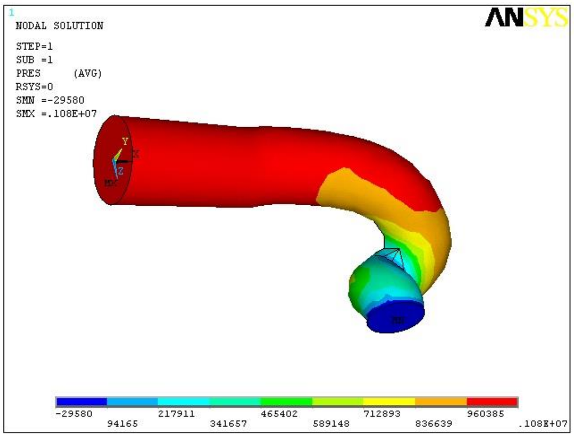

Fig 4.14: pressure distribution in intake port with 300^0 helix angle

4.5 Swirl ratio calculations

For straight port,
Vx = 5608.48
Vy = 1.941e5
V2 – 4.1217e4

Swirl ratio, SR = square root of $\left\{\dfrac{Vx^2 + Vy^2}{Vz^2}\right\}$

∴ Swirl ration = 0.047

Similarly form the above results swirl ratio for remaining ports was calculated.

Intake Ports	Velocity (outlet)	Pressure (inlet)	Swirl ratio
Straight port	245229	30090	0.047
Port with 160^0 helix angle	0.130E+07	904940	0.30512
Port with 220^0 helix angle	0.130E+07	927176	0.03568
Port with 300^0 helix angle	0.143E+07	0.108E+07	0.34770

Table 1: comparative results of intake ports

5. CONCLUSIONS

CFD Analysis of intake port was done to find out the flow characteristics for different intake port models and suggest the best one among them. This work is majorly focused to increase the Engine Efficiency & to reduce Fuel Consumption. In the process of analysis the intake first the existing straight port is modeled in NX cad and was analyzed in ANSYS Flortran. Analysis was done to determine velocity at the outlet, pressure at inlet and swirl ratio of intake port.

After doing the initial iteration the port was modeled by changing the helical angle. Swirl Ratio calculations the best flow is the Helical Angle 300. Its Swirl Ratio is 0.34470 and the Swirl Ratio for Straight Port is 0.047. Based on the obtained results 300 degrees intake port give better performance.

REFERENCES

[1] Chen & J. C. Dent (1994) An Investigation Of Steady Flow Through A Curved Inlet Port. SAE Paper Volume Ii 950818.
[2] Dr.Stanley K. Widener (1995) Parametric Design of Helical Intake Port. SAE Paper 950818
[3] H.Jasak, J.Y.Luo, B.Kaludercic & A.D.Gosman (1999) Rapid CFD Simulation of Internal Combustion Engines. SAE Paper 1999-01-1185.
[4] Jun-Ichi Kawashima, Hiroshi Ogawa and Yoshiyuki Tsuruhave (1998). Research on A Variable Swirl Intake Port For 4-Valve I-Speed D1 Diesel Engines. SAE Paper 982680.
[5] William Church and P.V.Farrell (1998) Effect of Intake Port Geometry on Large Scale In – Cylinder Flows. SAE Paper 980484.