RESEARCH ARTICLE OPEN ACCESS

Design and Analysis of the Effect of a Modified Valve with Helical Guideways on Combustion Performance of an I.C Engine

Pavan Chandra P V*, Siva Chaitanya A**
*(Department of Mechanical Engineering, Andhra University College of Engineering (A), Visakhapatnam-03)
**(Department of Mechanical Engineering, Andhra University College of Engineering (A), Visakhapatnam-03)

ABSTRACT
The in-cylinder flow of an Internal Combustion Engine(ICE) has drawn much attention of the automotive researchers and scientists in the present time. A good swirl promotes fast combustion and improves the efficiency. Based upon this concept, this paper describes the results of a study conducted to investigate the effects of a “modified valve with helical guide ways” on the performance of combustion. Small internal combustion engine is designed to be part of a very efficient vehicle to enter a consumption marathon. The engine should run at low speeds, in order to have low mechanical losses but combustion should be fast, enabling good combustion efficiency. Therefore, high turbulence is required prior to combustion within the cylinder, hence the concept of swirl is introduced and its effect on the combustion within the cylinder has been ascertained in the present work. Assessment of the effect of swirl on combustion performance within the cylinder requires excessive experimentation by modifying the design of various components of combustion chamber. Therefore, in the present work using the concept of CFD the simulation of combustion phenomenon has been carried out and the output parameters in the form of swirl ratio has been assessed. The modelling of combustion chamber has been carried out using CATIA software and the same is imported to analysis software ANSYS- CFX module. Here, the performance of the model is assessed by the swirl ratio. The swirl ratio of the modified valve (Valve with helical guide ways) is obtained as 1.45 which is much higher than a normal valve with swirl ratio 0.65 as assessed from the present work

Keywords- Swirl ratio, Turbulence, Combustion Performance, Helical Valves, Efficiency.

I. INTRODUCTION

The in-cylinder flows of Internal Combustion Engine (ICE) have drawn much attention to the automotive researchers and scientist in the present times. It is due to the fact that the flow structure generated by intake flows is related closely to the design and performance e of the Internal Combustion Engines. The production of turbulence of higher intensity is one of the most important factors for stabilizing the ignition process, fast propagation of flame, especially in case of lean-burn combustion. In general, two types of vortices are utilized in order to generate and preserve the turbulence flows efficiently. These vortices are usually known as swirl and tumble flows, which are organized rotations in the horizontal and vertical plane of the engine cylinder, respectively. They contribute to the improvement of engine performance. Hence, it is indispensable for the Development of an ICE with high compression ratio to realize high turbulence intensity and lean burn combustion. Many researchers worked in this are avia experimental as well as computation alto explore the phenol men on of the in-cylinder flow of Internal CombustionEngine.Bytakingtheimportanceofproducti onofturbulenceininternal combustion engine in to consideration and also taking the factors influencing creation of turbulence, an idea has been checked by performing flow analysis to find whether the idea is feasible or not .

As we know there are two types of fluid motion in production of turbulence i.e swirl and tumble, we are only concentrating on swirl and how to increase swirl which depends on many factors like by changing inlet port design, changing inlet valve design etc. After study of many research papers we implemented an idea in valve design to improve swirl and for analysing theamount of swirl produced, we considered a parameter called swirl ratio. As swirl ratio increases turbulence increases. The idea here is to introduce a helical guide ways in fluid flow. This helical guide ways comes over valve head attached to valve stem this is intentionally introduced to create turbulence at the very starting stage of fluid entering into chamber resulting in high swirl.

Internal combustion engines are quite different from external combustion engines, such as steam or Stirling engines, in which the energy is delivered to a working fluid not consisting of, mixed with, or contaminated by combustion products. Working fluids can be air, hot water, pressurized water or even liquid sodium, heated in a boiler. ICEs are usually powered by energy- dense fluids such as gasoline or diesel, liquids derived from fossil fluids.

Typically an ICE is fed with fossil fluids like natural gas or petroleum products such as gasoline, diesel fluid or fluid oil. There's a growing usage of renewable fluids like biodiesel for compression ignition engines and bioethanol for spark ignition engines. Hydrogen is sometimes used, and can be madefrom either fossil fluids or renewable energy.

A piston is seated inside each cylinder by several metal piston rings fitted around its outside surface in machined grooves; typically two for compressional sealing and one to seal the oil. The rings make near contact with the cylinder walls (sleeved or sleeveless), riding on a thin layer of lubricating oil; essential to keep the engine from seizing and necessitating a cylinder wall's durable surface

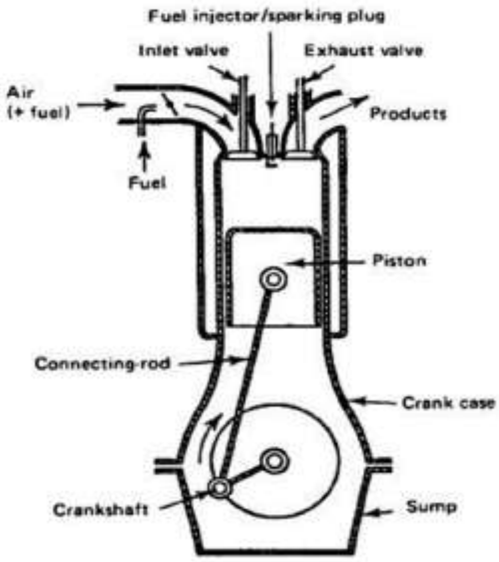
II. FLUID MOTION IN COMBUSTION CHAMBER

2.1 TURBULENCE

Due to the high velocities involved, all flows into, out of, and within engine cylinders are turbulent flows. The exception to this is those flows in the corners and small crevices of the combustion chamber where the close proximity of the walls dampens out turbulence. As a result of turbulence, thermodynamic transfer rates within an engine are increased by an order of magnitude. Heat transfer, evaporation, mixing, Turbulence and combustion rates all increase. As engine speed increases, flow rates increase, with acor responding increase in swirl, squish, and turbulence. This increases the real-time rate of fluid evaporation, mixing of the fluid vapour and air, and combustion. When flow is turbulent, particles experiencer and om fluctuations in motion superimposed on their main bulk velocity. These fluctuations occur in all directions, perpendicular to the flow and in the flow direction. This makes it im possible to predict the exact flow conditions at any given time and position.

Statistical average in gover many engine cycles gives accurate average flow conditions, but cannot predict the exact flow of any one cycle. The result is cyclic variations in operating parameters within an engine (e.g., cylinder pressure, temperature, burn angle, etc.).

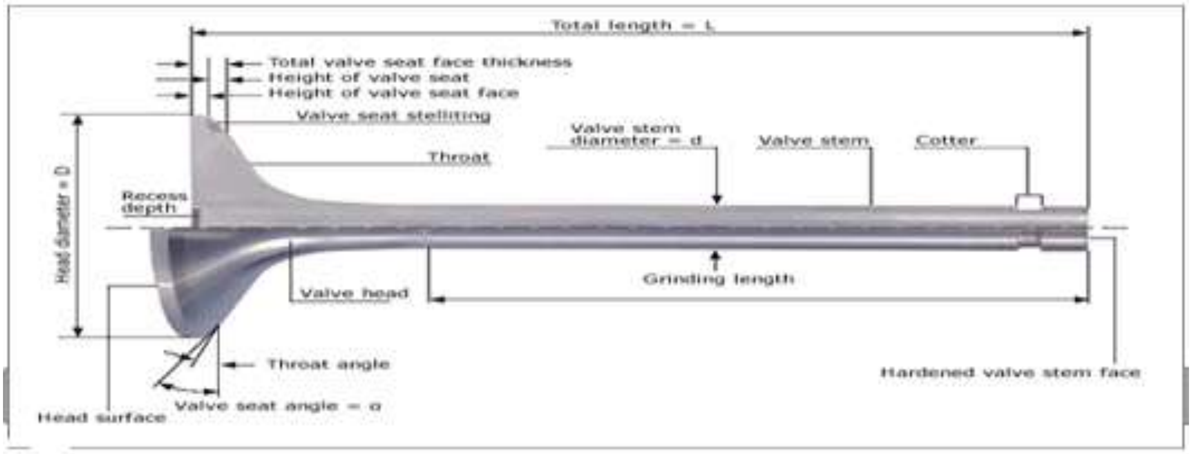
Turbulence in a cylinder is high during intake, but then decreases as the flow rate slows near BDC. It increases again during compression as swirl, squish, and tumble increase near TDC. Swirl makes turbulence more homogeneous throughout the cylinder. The high turbulence near TDC when



ignition occurs is very desirable for combustion. It breaks up and spreads the flame front many times faster than that of a laminar flame. The air-fluid is consumed in a very short time, and self-ignition and knock area voided. Local flame speed depends on the turbulence immediately in front of the flame. This turbulence is enhanced by the expansion of the cylinder gases during the combustion process. The shape of the combustion chamber is extremely important ingenerating maximum turbulence and increasing the desired rapid combustion.

2.2 SWIRL

The main macro mass motion within the cylinder is a rotational motion called swirl. It is generated by constructing the intake system to give a tangential component to the intake flow as it enters the cylinder. This is done by shaping and contouring the intake manifold, valve ports, and sometimes even the piston face. Swirl greatly enhances the mixing of air and fluid to give a homogeneous mixture in the very short time available for this in modern high-speed engines. It is also a main mechanism for very rapid spreading of the flame front during the combustion process.



Combustion chambers of most modern engines are shaped, with most of the clearance volume close to the cylinder centreline. The reason for this is to reduce the flame travel distance for most of the air-fluid mixture as it combusts near TDC. The clearance volume can be in the cylinder head in the crown of the piston, or in a combination of the two. With this kind of combustion chamber, as the piston nears TDC the radius of the rotating cylinder of air-fluid is suddenly greatly reduced.

This results in a large increase in angular velocity due to conservation of angular momentum. It is common to have angular velocity increase by a factor of three to five at TDC, even though viscous drag with the walls is very great at this point. High angular velocity at TDC is very desirable because it spreads the flame front through the combustion chamber very quickly. In some engines, burn time is decreased by positioning the spark plug so that it is offset from centre to take advantage of high swirl. In two-stroke cycle engines with intake ports in the cylinder walls, swirl is generated by shaping the edges of the ports and direction of the intake runners. Swirl greatly reduces dead spots in the scavenging process but also increases mixing of the inlet charge with exhaust residual. The shaping of inlet ports and runners to promote swirl reduces the volumetric efficiency of all engines.

Swirl ratio is a dimensionless parameter used to quantify rotational motion within the cylinder. Average values of either the angular speed or tangential speed should be used in these equations. Angular motion is very non-uniform within the cylinder, being a maximum away from the walls and being much less near the walls due to viscous drag.

The non-uniformity is both in the radial direction, due to drag with the cylinder walls, and in the axial direction, due to drag with the piston face and cylinder head.

During intake it is high, decreasing after BDC in the compression stroke due to viscous drag with the cylinder walls. Combustion expands the gases and increases swirl to another maximum part way into the power stroke. Expansion of the gases and viscous drag quickly reduce this again before blow down occurs. One-fourth to one-third of angular momentum will be lost during the compression stroke. One simple way of modelling cylinder swirl is the paddle wheel model.

III. METHODOLOGY

3.1 DESCRIPTION OF THE PROBLEM

Producing a computer simulation of a flow problem requires the analyst to providealargeamountofdatatothesolverprogram.Itisthe qualityofthisdata, in terms of both suitability and accuracy that may well determine the quality of the results of the simulation. The key to a sound analysis is the production of as pecification of the flow problem. This is a clear exposition of the reasons why the simulation is being carried out and of what the physical flow situation is. Once it has been produced it can be translated in to the set of data that is required by the simulation package.

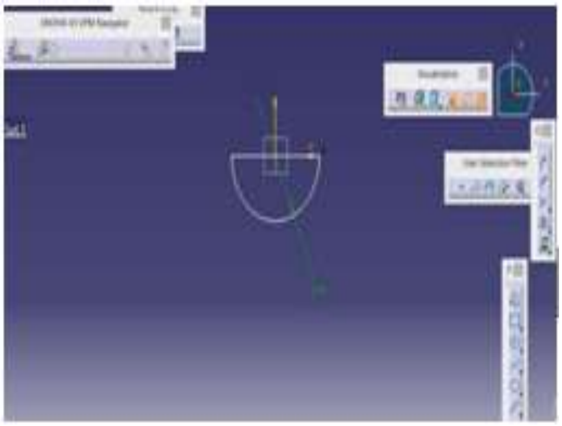
3.2 DESIGN OF THE COMBUSTION CHAMBER

In the present work a standard combustion chamber having a cylinder of diameter 50mm, is modelled using CATIA software.

Cylinders along with inlet and outlet ports, valves are modelled. The modelling is done in such a way that the model can be applied in practical conditions. The model generated using the CATIA software is further exported to the ANSYS software for the meshing and analysis of the model.

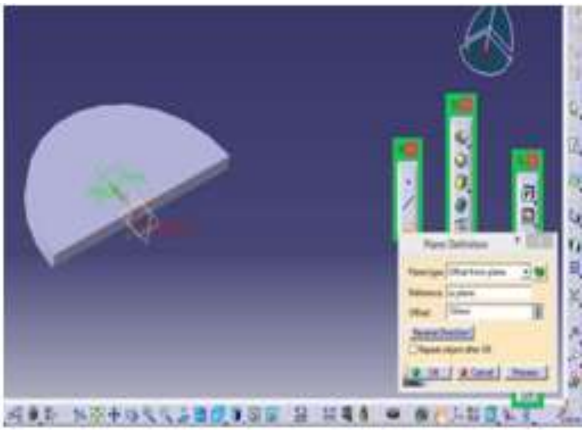
Step-1 Generation of circle

From this part design is taken from anyaxis. Then the cylinder head is modelled with50mm diameter as shown in the figure below.



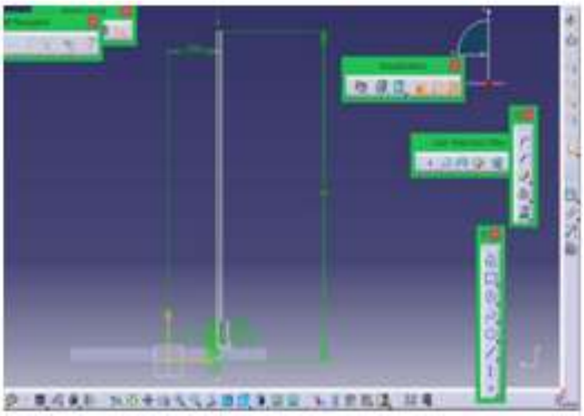
Step-2 Padding

Then pad operation is done on the figure to get the thickness of 3mm. Here padding is giving thickness.



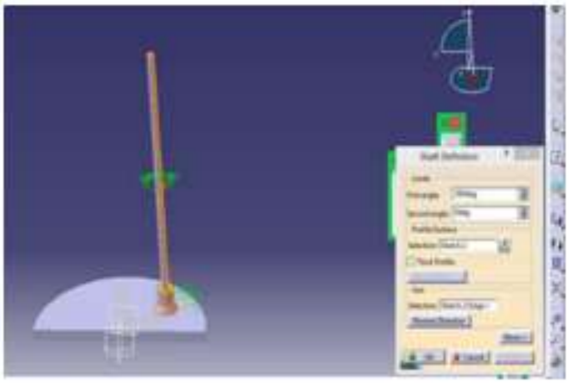
Step-3 Defining plane

Todrawthevalvesonthesurfaceoftheheadareferenc eplaneisselectedandthevalvedesign is continued as shown in the figure. The reference plane is done with a offset of 10mm



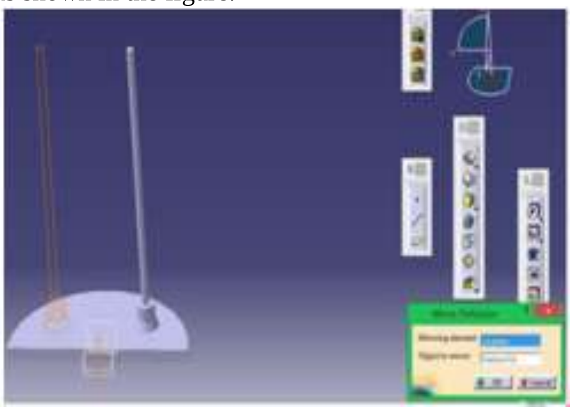
Step-4 Producing valve using poly-line

The reference plane is selected and the valve is drawn on the face of the head as shown in the figure



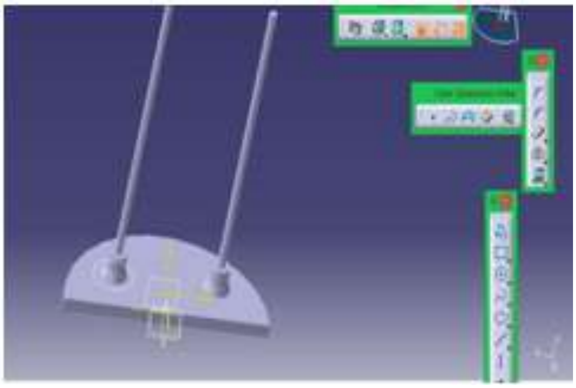
Step-5 Shaft generation

The shaft operation is performed and on the valve and it is revolve by taking the reference plane as shown in the figure.



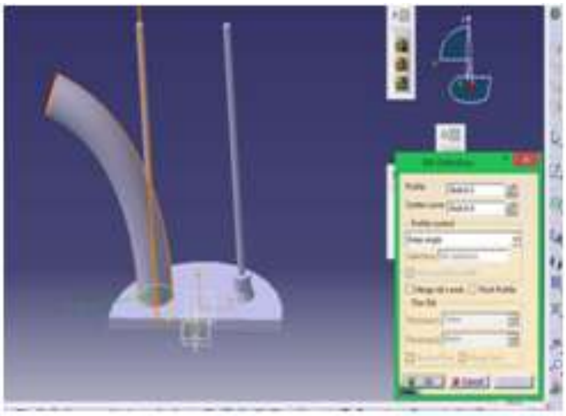
Step-6 Mirroring the shaft

After the shaft operation it is mirrored by selecting the axis. Then we get the two valves on the head.



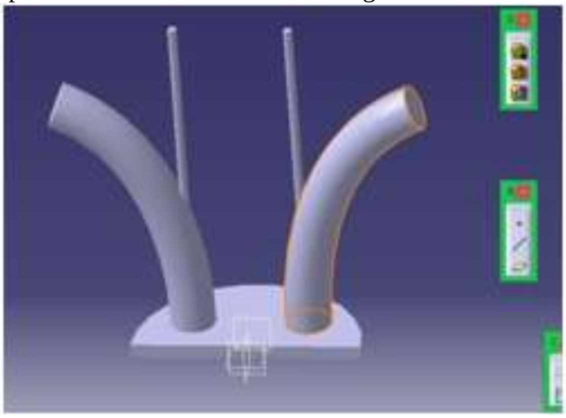
Step-7 Defining plane and generation of inlet port

To create the inlet and outlet ports we have to perform rib operation. By using this operation we can get the solid ports. The reference plane is selected and the inlet port is drawn with a diameter of 10mm.



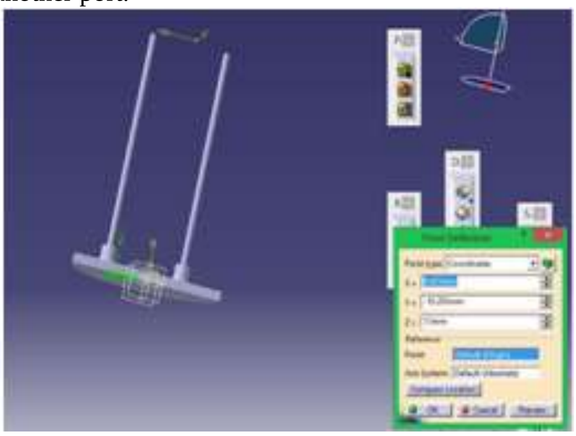
Step-8 Rib operation using spline tool

Here the spline tool is used to perform the rib operation as shown in the below figure



Step-9 Mirroring the port

Again the mirror operation is performed to create another port.

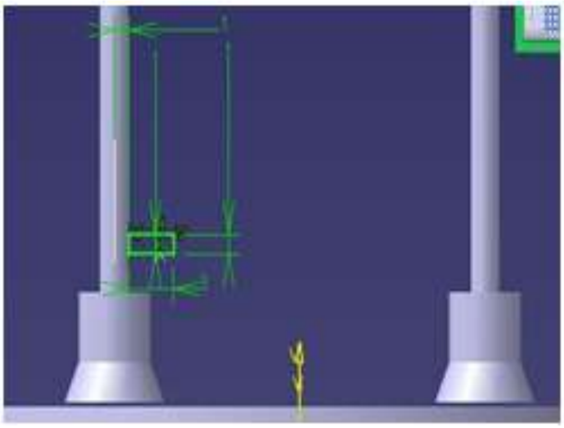


Step-10 Selecting coordinates

Now the helical spring is created by making a reference point and the helical spring is generated. On the inlet port, a helix is generated by using shape design module. To generate the helix wehave to take a start point.

The start point is taken with three coordinates as shown in below figure.

X= 9.423mm
Y= -10.203mm
Z= 11mm

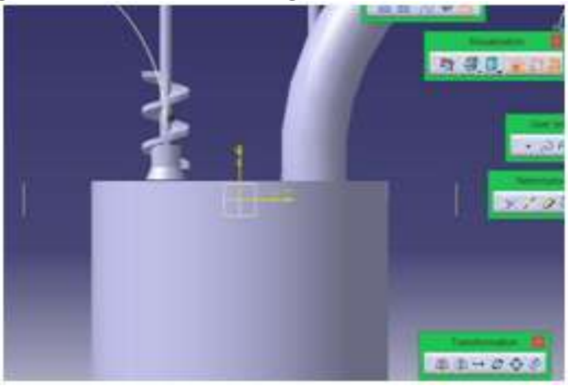


Step-11 Helix operation using generative shape design

By keeping the coordinates a reference line is taken on the inlet valve and the helix operation is done by using generative shape design module.
The pitch is taken as 4.5mm with height 10mm

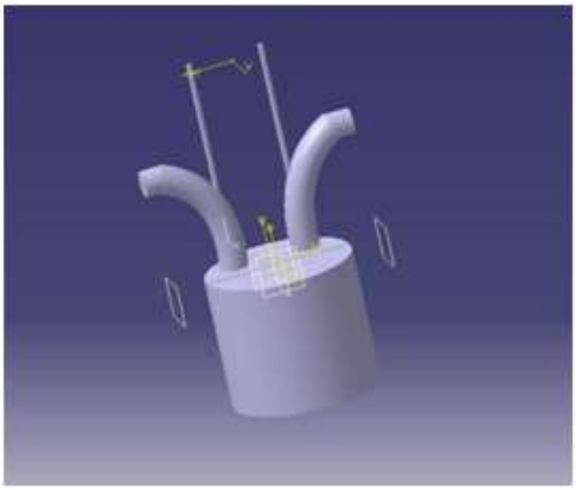
Step-12 Constraints for helix operation

To perform the helix operation the constraints are given as shown in the dialogue box



Step-13 Formation of helical guide ways

Now the helix will be formed on the inlet port as shown in the below figure

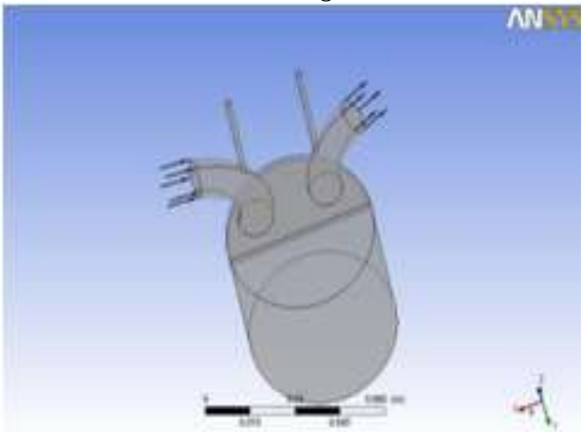


Step-14 Final design

The total part body is

IV. ANALYSIS USING CFD

The model generated using CATIA which is saved in .igs or .stp file is imported into ANSYSCFX module. Now for obtaining better results in analysis, meshing for the file is done. Automatic meshing method is used for mesh generation. The boundary conditions are then selected. The inlet boundary conditions taken are normal speed 40m/s, turbulence with medium intensity. The outlet boundary conditions taken are static pressure which is 0Pa. The swirl ratio expression is given as input for obtaining the swirl ratio. By generating solution for applied boundary condition and giving iteration convergen cenumber, the solution converges.



4.1 CFD ANALYSIS OF NORMAL VALVE

Theboundaryconditions areset fortheinlet byselectingtheface. Theinputis given as normal speed 40m/s,medium intensityturbulence. Nowthe outlet faceis selected and static pressureis taken as 0Pa.Theports, valves andcylinderis consideredas wall, which is no slip condition.

SWIRL RATIO FORMULA:

*AREAINT(DENSITY*SQRT(VELOCITYV*VELOCITYV)*SQRT((VELOCITYU*VELOCITYU)+(VELOCITYW*VELOCITY W))*SQRT((X*X)+(Z*Z)))@PLANE1/ (MAXVAL(SQRT(X*X)+(Z*Z)))@PLANE1*AREAINT(DENSITY*VELOCITYV *VELOCITYV)@PLANE1*

Theswirl ratio expression is given in CEL expression and the swirl ratio is generated usingthis expression. Thesolution is obtained byusingthe solver control at high resolution and convergence control up to 100.

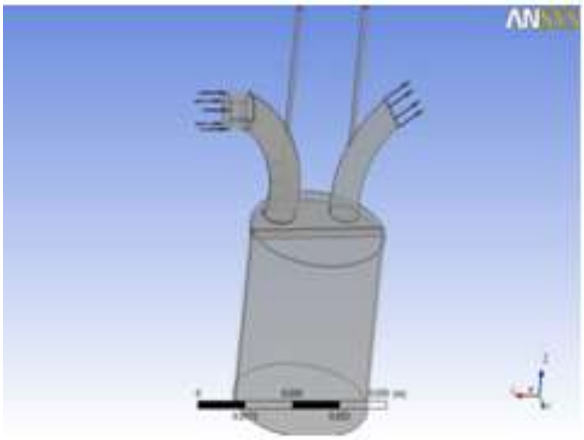
4.2 CFD ANALYSIS OF VALVE WITH HELICAL GUIDEWAYS

The bound ary conditions are set for heinlet by selecting the face. The input is given as normal speed 40m/s, medium in tensity turbulence. Now the outlet face is selected and static pressure is taken as 0Pa.Theports, valves and cylinder is considered as wall, which is no slip condition. The helical guide ways is also considered as wall.

Swirlratioformula:

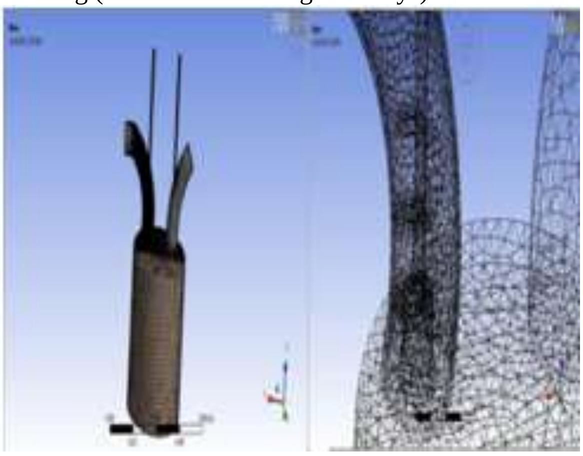
*AREAINT(DENSITY*SQRT(VELOCITYV*VELOCIT*

*YV)*SQRT((VELOCITYU*VELOCITYU)+(VELOCI TYW*VELOCITY W))*SQRT((X*X)+(Z*Z)))@PLANE1/ (MAXVAL(SQRT(X*X)+(Z*Z)))@PLANE1*AREAIN T(DENSITY*VELOCITYV *VELOCITYV)@PLANE1*



Theswirl ratio expression is given in CELexpression and theswirl ratio is generated usingthis expression.

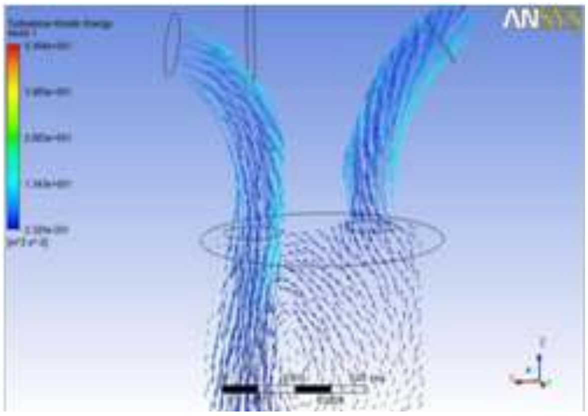
Thesolution is obtained byusingthe solvercontrol at highresolution and convergence control upto 100. Meshing (valve with helical guide ways)



V. RESULTS AND DISCUSSION

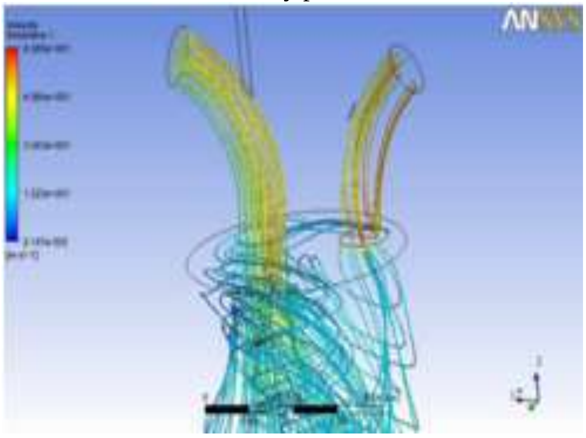
5.1 RESULTS FOR NORMAL VALUE

The model led combustion chamber using CATIA has been solved for various fluid dynamic parameters using CFD and the results obtained are represented as velocity streamlines, pressure stream lines, and variation of fluid velocity in considered plane, variation of fluid pressure in considered plane, variation of fluid turbulence kinetic energy, vector representation of fluid velocity, vector representation offluid pressure.



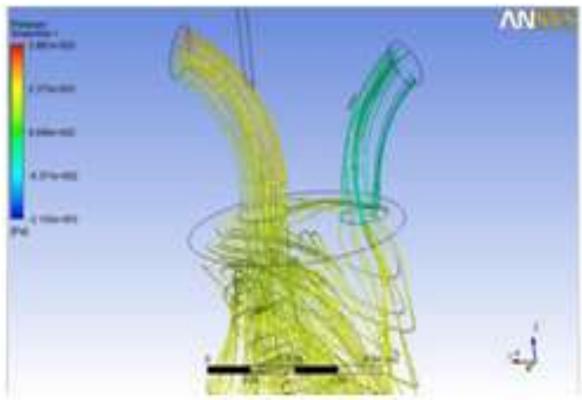
RESULTS FOR NORMAL VALUE

Streamlines are the Geometrical representation of the of the flow velocity. Streamlines are a family of curves that are in stantaneo uslytangent to the velocity vector of the flow. These show the direction amass less fluid element will travel in at any point in time.

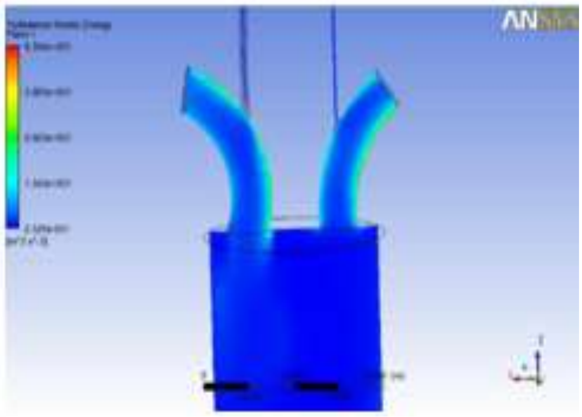


VELOCITY STREAM LINES

PRESSURE STREAM LINES

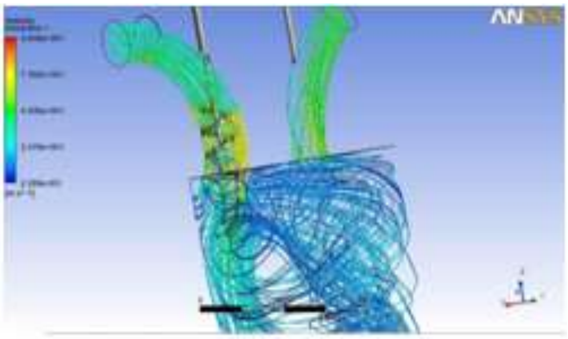


VARIATION OF TURBULENCE KINETIC ENERGY OF FLUID IN CONSIDERED PLANE:



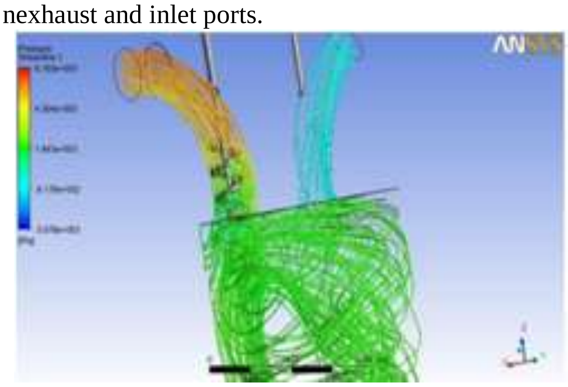
SWIRL RATIO

5.2 RESULTS FOR VALVES WITH HELICAL GUIDEWAYS:

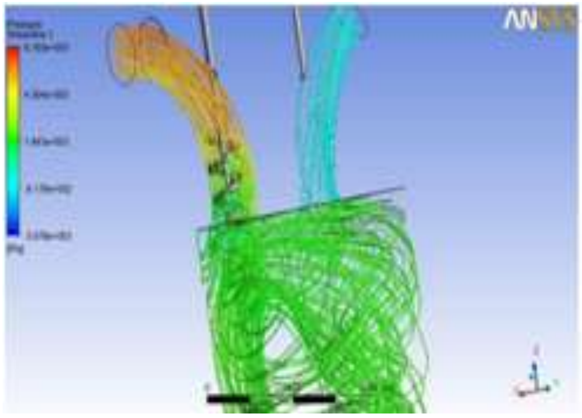


VELOCITY STREAM LINES

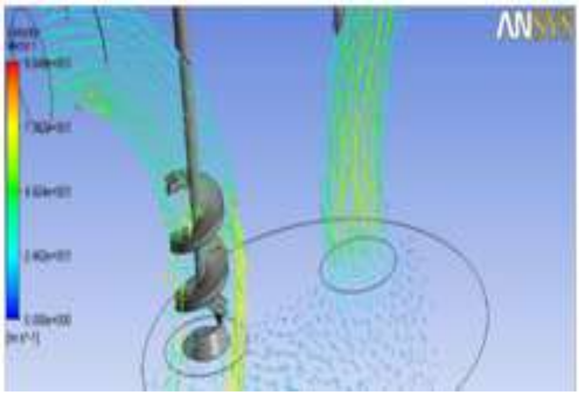
Unlike normal guide ways, the setupwithhelicalguide wayshassuddenincrease inthefluid velocity atthehelicalguidewaysandgoesdecreasingasitentersthe chamberandstabilizes. Velocity offluidsuddenly increasesattheentranceofexhaustportandremainssamei nexhaust and inlet ports.



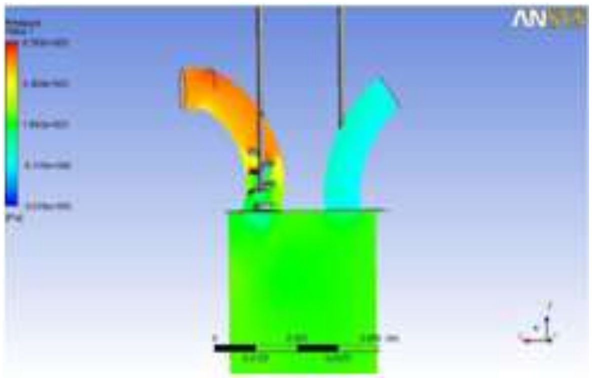
PRESSURE STREAM LINES:



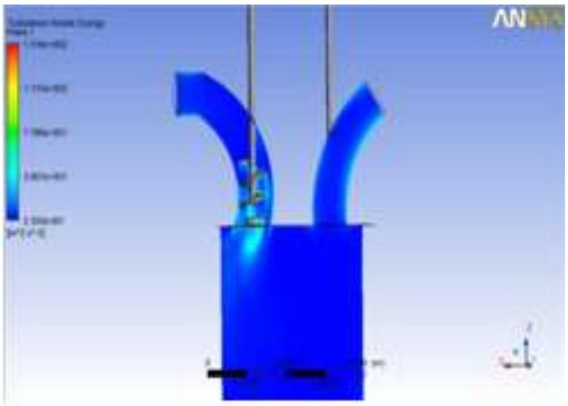
Pressure of fluid is very high in the inletport until the fluid reach zeshelical guide ways. After helical guide way spres sure decreases suddenly and stabilizes in the chamber. Pressure again decreases further at the entrance of exhaust port and remains same in exhaust port.



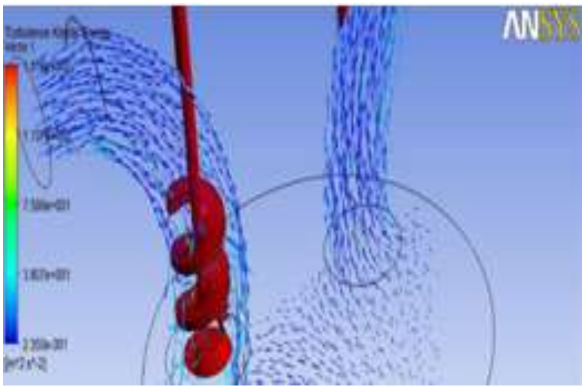
VECTOR REPRESENTATION OF FLUID VELOCITY:



VARIATION OF FLUID PRESSURE IN COSIDERED PLANE



VARIATION OF FLUID KINETIC ENERGY IN CONSIDERED PLANE:

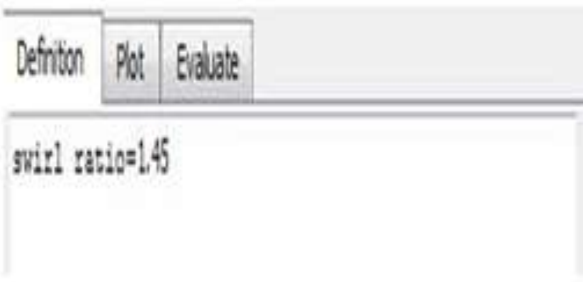


VECTOR REPRESENTATION OF FLUID TURBULENCE KINETIC ENERGY.

Turbulence kinetic energy is highly concentrated at the upper convex part of inlet port at the helical guide ways. It is also highly concentrated in inlet and the exhaust ports and decrease gradually as fluid enters the chamber. TKE is moderately concentrated at the remaining places of the chamber.

SWIRL RATIO

From the given equation.



VI. CONCLUSIONS

Based on the results generated in this study, following conclusions can be derived.

- Redesigning the inlet valve with helical guide ways proves more efficient thanthenormalvalveintermsof turbulence.
- High turbulence in fluid flow can be created by sudden variations in velocity and pressure of the fluid flow. He lical guide ways creates sudden increase and decrease in fluid velocity and sudden drop in its pressure. Due to this high turbulence in fluid flow is created, which leads to high performance.
- Usinghelicalguidewaysontheinletvalveimprovess wirlgenerationinthe cylinder and results in increase in swirl ratio by 123.07%. By increasing the swirl, rate of combustion increases, which results in the increase in efficiency of the engine. As swirl increases in the combustion chamber, more combustion of fluid takes place; this reduces the emissions from the engine.
- It is expected that by reducing the pitch and increaseing the height of the helical guide ways results inmore generation of swirl.
- The analysis of the modified valves with helical guide ways is highly efficient and is a very practical way to reduce incomplete combustion and increase high performance with reduction in harmful gases released to the surroundings.
- There is a lot of future scope for this analysis as the introduction of Bio Diesel as an alternate to the petroleum fuels raises a perspicuous doubt regarding the performance of the engine based upon its swirl ratio.
- Therefore, the conclusion is that by modifying the valves with helical guide ways then the efficiency and performance of engine would increase optimistically allowing a good combustion characteristics. Although the manufacturing of helical guide ways is a little expensive thought, but when compared to its performance it is worth a thought.
- The swirl ratio for the valves with modified helical guide ways is 1.45 and that of a valve without helical guide ways is 0.65.

VII. Acknowledgements

We are grateful to our University Faculty and the Computational Laboratory for guiding us through this great milestone by providing us with valuable suggestions and an original version of CATIA and ANSYS softwares.

We are grateful to our Head Of the Department for allowing us to conduct experiments and analysing the results in the Department Laboratories.

REFERENCES

[1] H. Mohamed Niyaz, Prof. A. S. Dhekane, Twin *Helical Intake Port DesignOptimization And Validation By Using CFD Analysis*, 4,April 2014
[2] Abhilash M Bharadwaj, K Madhu,

Seemanthini J, Vismay K G Anand M Shivapuji&Aravind T6, *Study of Swirl and Tumble Motion using CFD*.

[3] H.Sushma and Jagadeesha.K.B, *CFD modelling of the in-cylinder flow inDirect-injection Diesel engine*, 12, December 2013

[4] C. Ramesh Kumar and G. Nagarajan, *Investigation of flow during intake stroke of a single cylinder internal combustion engine*, February 2012

[5] Pandey K.M. and Roy Bidesh, *Intake valve design for high level swirl induction in carbureted spark ignition engin*e, July-September 2012

[6] Bassemramadan, *A study of swirl generation in DI engines using KIVA-3V*, Kettering university.

[7] K Madhu, Seemanthini J, Vismay K G Anand M Shivapuji&Aravind T6,*Study of Swirl and Tumble Motion using CFD*.

[8] Timoney, David J., and Smith, William J., "*Influences of Fuel Injection and Air Mixing Rates in a DI Diesel Combustion System*". SAE Paper 960035, 1996.

[9] Poulos, Stephen G. And Heywood, John B., "*The Effect Of Chamber Geometry On Spark-Ignition Engine Combustion*". SAE Paper 830334, 1983.

[10] Alkidas, A. C., and Suh, In-Soo*,"The Effects Of Intake Flow Configuration On The Heat Release And Heat Transfer Characteristics Of A Single Cylinder Four Valve Si Engin*e".SAE Paper 910296, 1991.