

ABSTRACT

The flow of air to the combustion chamber plays a vital role in complete and efficient combustion of air-fuel mixture in diesel engines. Thus, the field of air management involves various research tools like computational fluid dynamics (CFD), high resolution cameras to mention a view. The better the air flow structure, the better will be the end result. Thus, to improvise the air flow inside the combustion chamber, air has to be given whirling effect beforehand which can be done by various ways like shrouding the inlet valves' train design, modifying the piston head to mention a few. This not helps in generating the required organized turbulence but also helps in rectifying emissions, performance and fuel economy. Therefore, in this paper, effects of different piston heads will be considered and also will arrive at the best possible shape and design of piston head for single cylinder diesel engine.

KEYWORDS: Air Management, Diesel Engine, Piston Head, CFD

INTRODUCTION

One of the principal innovations in piston head knowhow is the use of different piston "tops" or "crowns". This is the part which is a sub part of the combustion chamber and is subjected to combustion. The diesel engines often use pistons with differently shaped crowns. Nowadays, the petrol engines are also coming with innovative designs of piston heads. The boundaries and overall design of piston head controls the movement of air and fuel as the piston moves in for the compression strike before the mix is ignited.

The basic functions of piston include:

- To receive the thrust force generated by the chemical reaction of fuel in the cylinder and transmit to connecting rod.
- To reciprocate in the cylinder and provide seal in suction, compression, expansion and exhaust stroke.

Generally, the pistons are made of Al alloy and Cast Iron. But, Al alloy is more preferable in comparison to cast iron because of its light weight which suitable for the reciprocating part. There are drawbacks also of Al alloys as Al alloys are less in strength and have lesser wear resistance. The heat conductivity of Al is about thrice of the cast iron which helps in maintaining proper cooling. Al pistons are made thicker which is necessary for strength. Being an important part in diesel engine, piston withstands the recurring fuel-air pressure and the inertial forces. Further, this working condition may cause the fatigue damage of piston eventually, such as piston side wear, piston head/crown cracks and so on. The investigations indicate that the greatest stress appears on the upper end of the piston and stress concentration is one of the mainly reason for fatigue failure. Therefore, my aim will be to optimize the upper end of the piston which is the piston head. Thus, before manufacturing the entire piston, I will simulate the air flow at the intake for different piston heads. It will not only save cost, time and resources but will also help us to arrive at the best possible shape of the piston head. The various designs of piston heads will be considered after referring to various research papers and also by my own imagination. The swirl angle should not exceed five degrees according to the standards. In totality, I will be able to compare the effect on air intake flow because of different shapes of piston heads.

MATERIALS AND METHODS

Objectives

1. To design various shapes of piston heads and understand the role of crowns and depressions.
2. To perform CFD simulations of the designed piston heads of the IC engine.
3. To study the effect of different piston head configurations on the in-cylinder air flow (only intake and compression stroke; combustion stroke is not considered.)
4. To compare effect of different piston head configurations on volumetric efficiency, turbulence, swirl and tumble ratio in the engine.
5. To know whether the flow is achieving uniform, high quality organized flow in the test section
6. To arrive at a shape which will not have any eddy air flow
7. To also arrive at a shape which will have organized strong turbulence and uniform flow

Figure:



Fig. 1: Piston Heads 9pt Times New Roman, Bold, Italic

Engineering Detail

While carrying out CFD analysis, we are concerned with just the depressions and crowns in the piston head presently as the volume of piston combustion stroke will not play role in determining the air flow in the chamber. We are tracking the swirl formed at time-‘t’ and will validate through inspection if the swirl is strong enough to eliminate any eddy formation. Also, we know that volumetric efficiency is directly linked with mass density of the air-fuel mixture drawn, thus a stronger swirl indicating a higher mass density will result in higher volumetric efficiency.

Moreover, we are also going to see if the swirl formed (i.e. the helicity) is utilizing the space effectively or not. By doing this we will be able to ensure that there is no extra or unnecessary amount of material manufactured, thereby decreasing the cost of manufacturing.

Besides swirl, the air has a tendency to tumble which is a characteristic property influenced by the piston head shape. In simple terms, air flips or somersaults over and over.

Thus, these two parameters represents the fluid flow behaviors occurring inside the combustion chamber which influences the air streams during intake stroke and enhances greatly the mixing of air and fuel to give better mixing during compression stroke.

The in-cylinder fluid motion in internal combustion engines is an important factor which controls the combustion process. The better the combustion process, the lesser are the effluents/toxic emissions. Therefore, a better understanding of fluid motion during the induction process is critical for developing engine designs with the most desirable operating and emission characteristics. Therefore, matching the combustion chamber geometry, fuel injection and gas flows are the most crucial factors for attaining a better combustion.

RESULTS AND DISCUSSION

In this study we are concerned on the swirl motion of inducted air during the suction stroke and during compression stroke. What we want to do in CFD is observing the fluid flow prior to combustion in internal combustion engines which is generated during the induction process. The conditions taken are velocity inlet and pressure outlet. Using Bernoulli, we reached to an inlet velocity of 53.87m/s and pressure at outlet of 84000 Pa. We also know that to increase the swirl intensity, the gas velocities must be high at the inlet valve, thereby, requiring a smaller cross-sectional area of the inlet valve. We also know that there is a requirement of smaller inlet valve for better swirl intensity generation.

The observations that were observed are as followings.

CASE 1:

Now, after researching various research papers and reviewing numerous websites we started our design of piston head (which can be scaled according to the requirement) we came to the conclusion that first we will have two triangular depressions in the piston head. (NOTE: It will be difficult to manufacture)

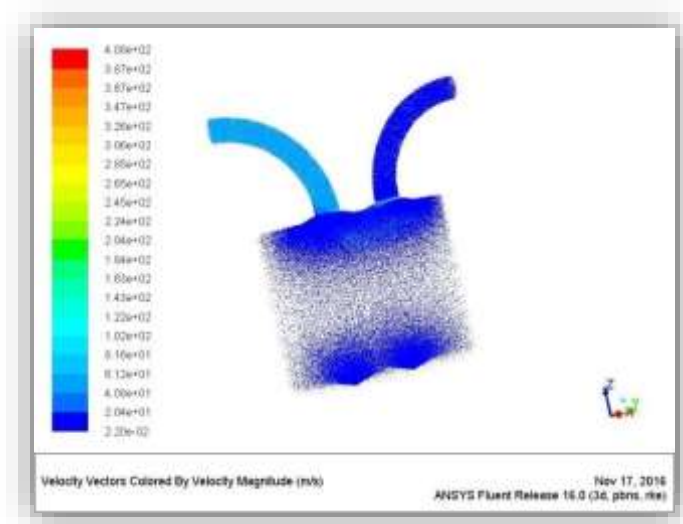
SOLIDWORKS MODEL:

So this is the front view of our geometry. The reason for geometry being like this is that, this is the space where air flow will happen before the injection of fuel. The geometry is modelled in solidworks where two sharp triangular depressions are extruded. The triangular depressions show that there are depressions on the piston head. As, we are just concerned with the air flow above the piston head only, so we've not modelled the entire piston. Also, the full piston head geometry which includes piston rings is not included. This will save our analysis time as numbers of elements in our mesh are reduced drastically. Here, the picture consists of intake and outlet manifold. Now, the inlet and outlet valves are not considered because we are considering the flow during compression at one point of time.

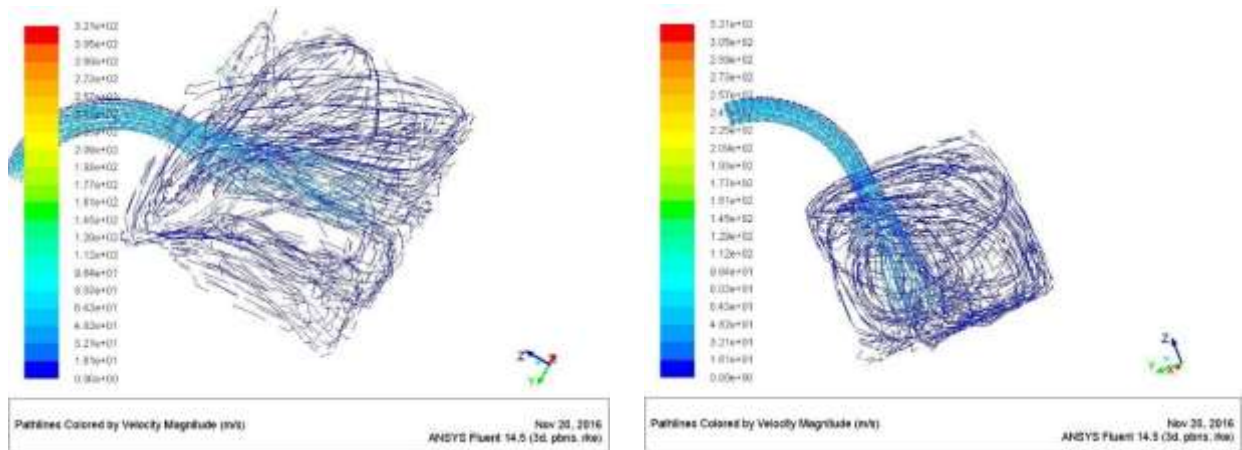


ANSYS SIMULATION RESULTS:

This picture represents the vectors during the intake and compression stroke. The meshing is done before hand which ensures dividing the structure in finite elements. This indicates the air concentration around the region of valves and depressions over piston head.



The path lines which are generated show the flow of air after entering the intake manifold and also the path it will follow, if manufactured. The path lines help us in inspecting the swirl and turbulence in air flow before combustion. The front and back view of the flow has been snapped and attached below.



Here, we can clearly see that the flow is disorganized and turbulence will not be strong. This will result in lesser mass density of air before it gets mixed with fuel. Thus, efficient combustion will not take place.

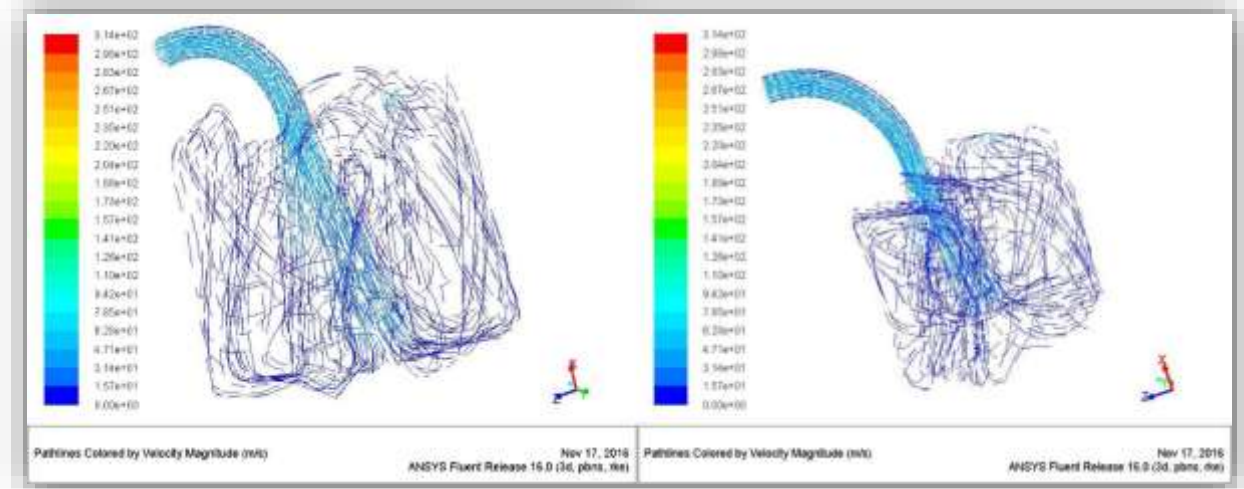
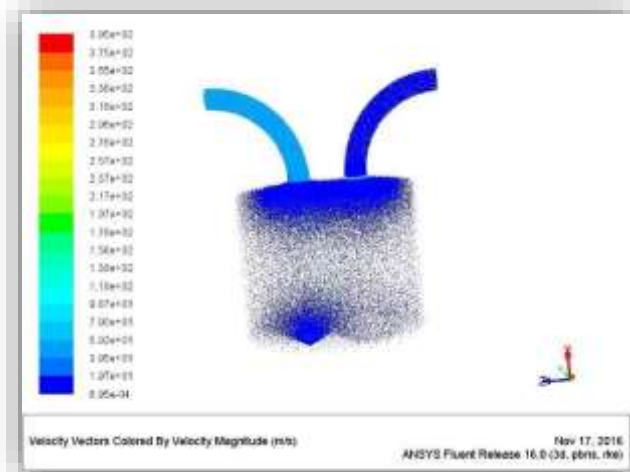
CASE 2:
SOLIDWORKS MODEL:

Now, we thought of a triangular and a circular depression simultaneously placed closed to each other. This means that the piston head of piston will have one triangular and circular depression. The solidworks model is created and imported in Ansys for further analysis. Here, likewise the intake and outlet manifold is defined and inlet and outlet parameters of velocity and pressure are given respectively. (The values of conditions for each of the simulation will remain same).



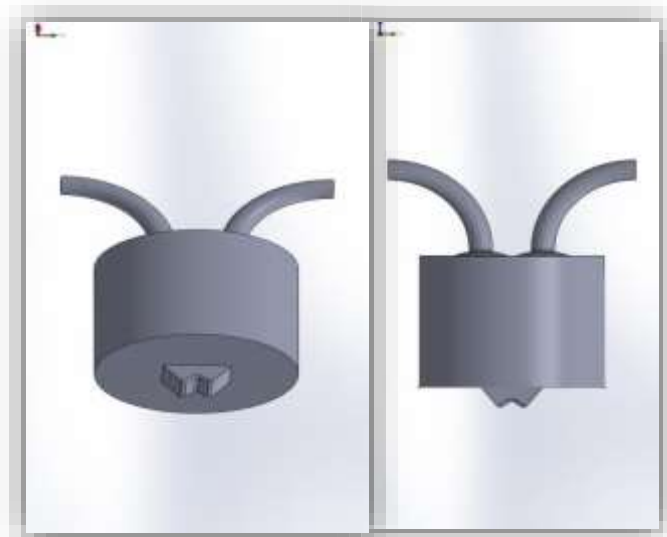
ANSYS SIMULATION RESULTS:

This is the picture of vector generated of piston head with triangular and circular depression. The concentration of vectors at the triangular depression is much more than that at the circular depression. The path lines indicating the flow of air is as follows for the same configuration. The front and back view of the path lines are snapped and attached below. Also, this shows that the flow obtained is scarce and is not dense. This indicates lesser mass density and a weak turbulence.



**CASE 3:
SOLIDWORKS MODEL:**

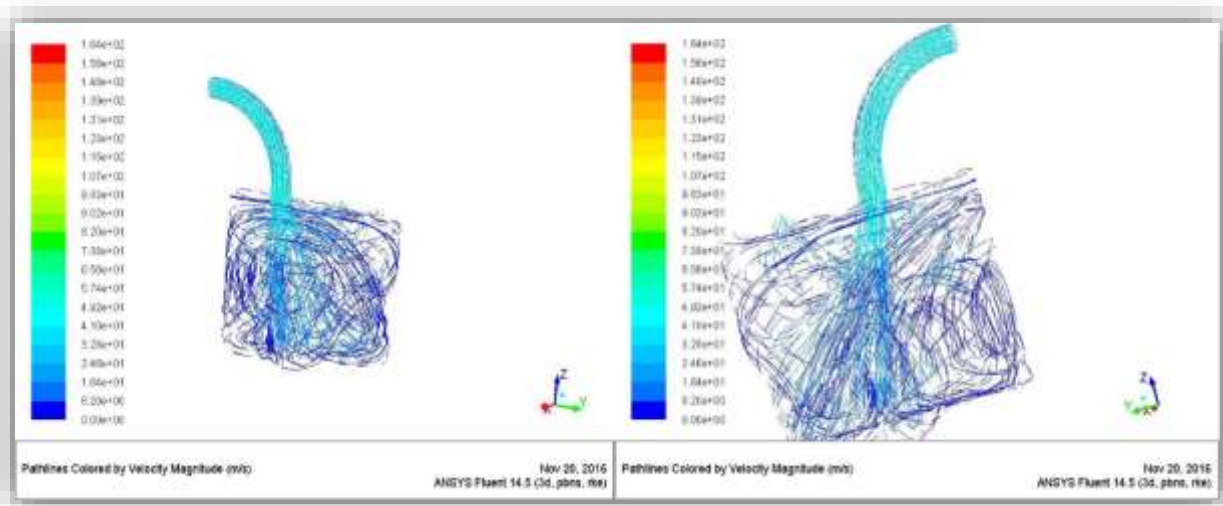
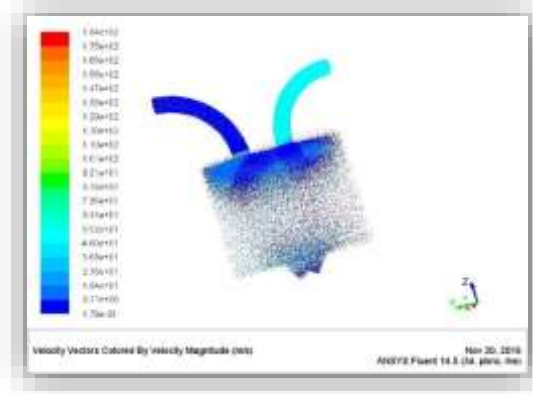
This is the third case where a depression formed is in the shape of 'W'. The location of the depression is kept exactly at the center as the air from inlet will hit approximately at the center of the piston head after intake. The fillets are given at appropriate places for the ease of manufacturing.



ANSYS SIMULATION RESULTS:

The vector formation for this case is hereby attached. The velocity at the inlet is shown in light blue color. The velocities of the vector can be seen by the chart/slide adjacent to it.

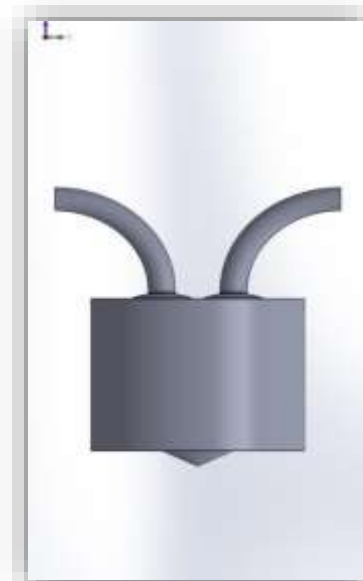
The path lines observed are as follows, the continuous circling of the air can be seen easily from the following pictures. The front view and the back view clearly justifies that there is an eddy formation at some points because the air is circling in some part only and not forming an organized turbulence throughout the compression space. This results in an unstructured flow of air which will not help in correct mixing of air with fuel.



CASE 4:

SOLIDWORKS MODEL

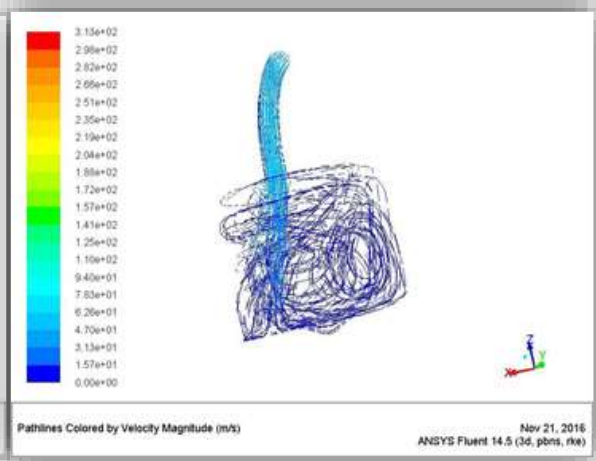
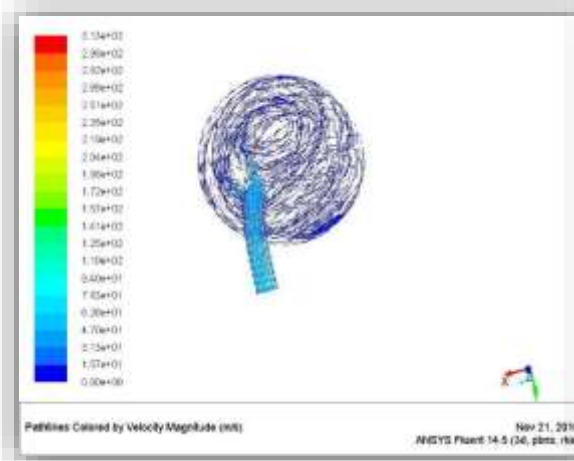
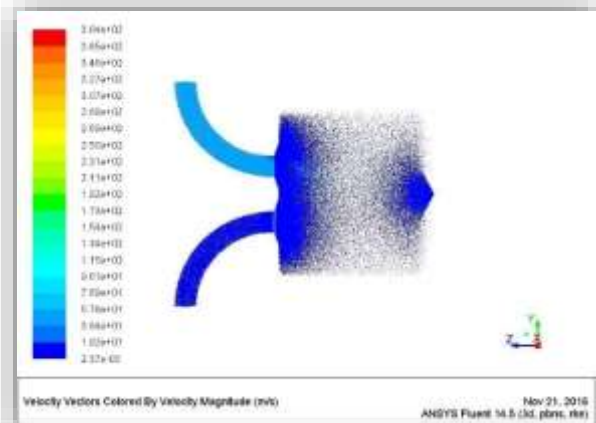
Now, we are having geometry where we will include just a single triangular depression as we also have to ease our manufacturing cost and time, and at the same time, we should be able to generate the required turbulence. And with the same values and conditions, simulations are carried out so that we can observe the path lines and vectors for better understanding.



ANSYS SIMULATION RESULTS:

The vectors are hereby generated with the help of Ansys and attached. The top view and front view of the path lines are also shown below, after the solution has converged following 'n' number of iterations. Thus the air flow can be interpreted by inspecting how the air travels inside the compression space available over the piston head.

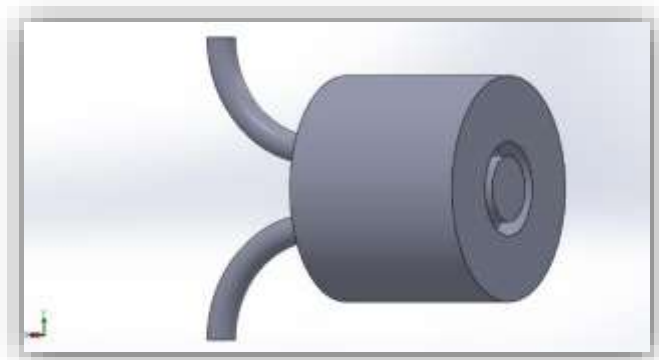
The indicative flow here tells us that the turbulence formed is strong at some patched, but not all over the geometry. Thus, this cannot be taken for consideration.



**CASE 5:
SOLIDWORKS MODEL:**

It can be clearly seen here that a circular crown (as treated by Ansys) is made which will prevent the inlet charge to directly go out of the outlet valve without combustion. Also, crowing of piston head can be studied now.

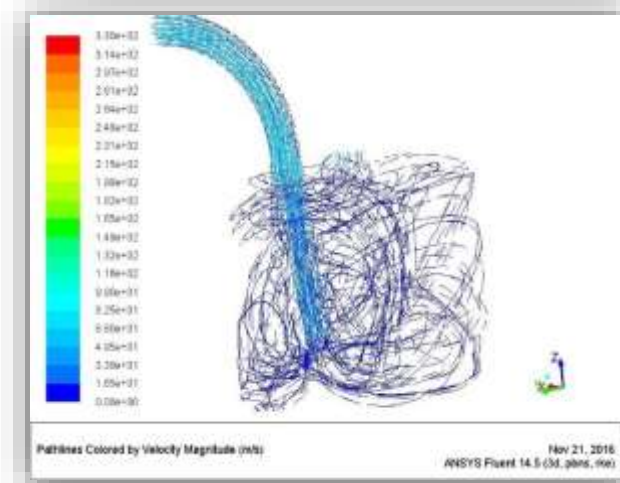
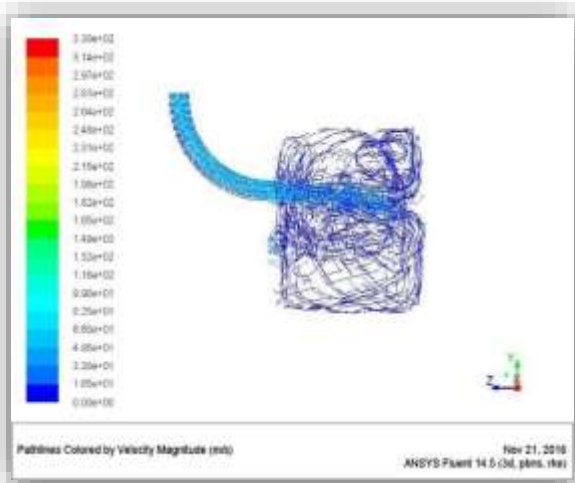
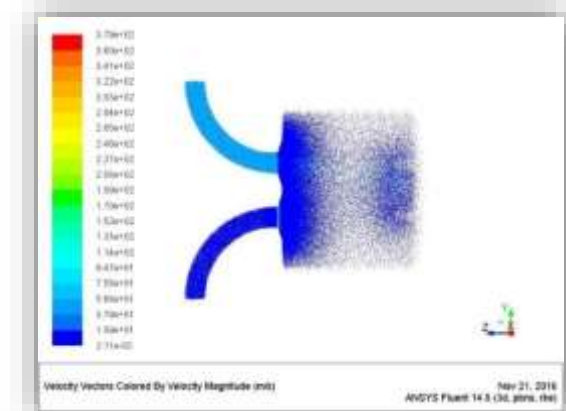
Actually, the piston head will be a hollow cylindrical crown when manufactured. It is designed this way because Ansys treats all the solid body as the shape of piston head.



ANSYS SIMULATION RESULTS:

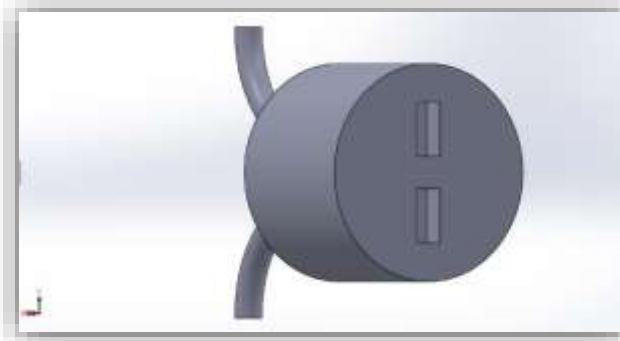
The vector generation showing the intensity of velocity vectors at various points. The vectors are shown to justify that particles reach the crown over piston head and are present in large number.

The path-lines formed are attached as follows and we can see that the swirl is disorganized due to eddy formation and crisscross flow. Hence the geometry will be neglected.



**CASE 6:
SOLIDWORKS MODEL:**

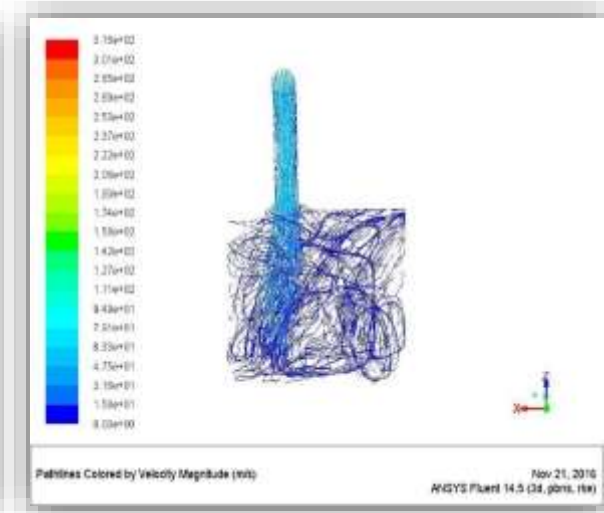
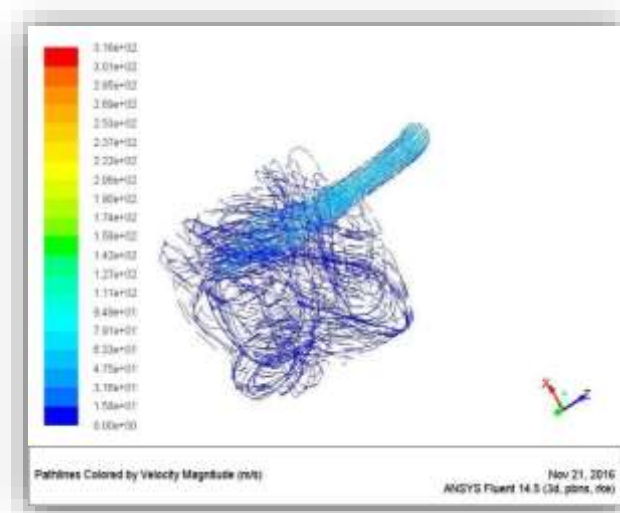
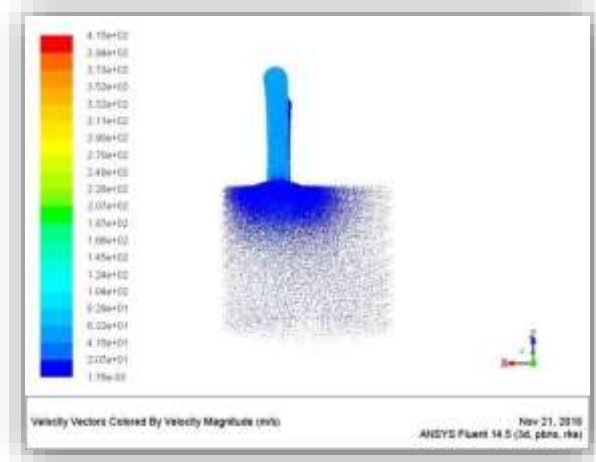
Here, two rectangular crowns are made over the piston head (as treated by Ansys) and the effect is studied later on. We've taken this geometry just to see if crowing is more advantageous over creating depression on the piston head.



ANSYS SIMULATION RESULTS:

The vectors and path lines are shown here. The path which air follows shows waviness and is unsteady. Moreover, it can be seen that concentration of swirl is more at a particular space instead of in the entire section.

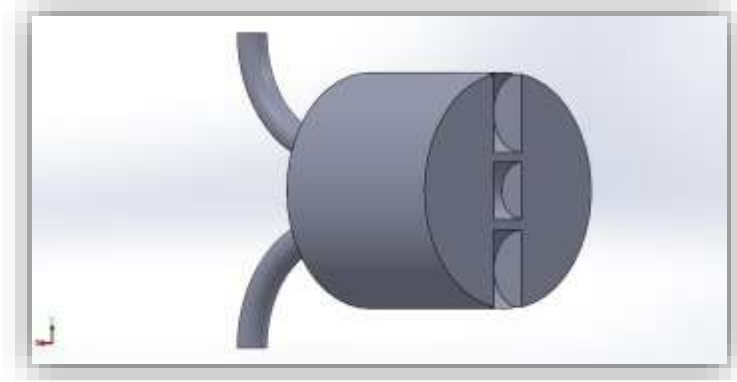
Here, the mass density is considerable but the swirl is not uniform. The different swirls will cut each other's flow and will result in weak turbulence eventually.



CASE 7:

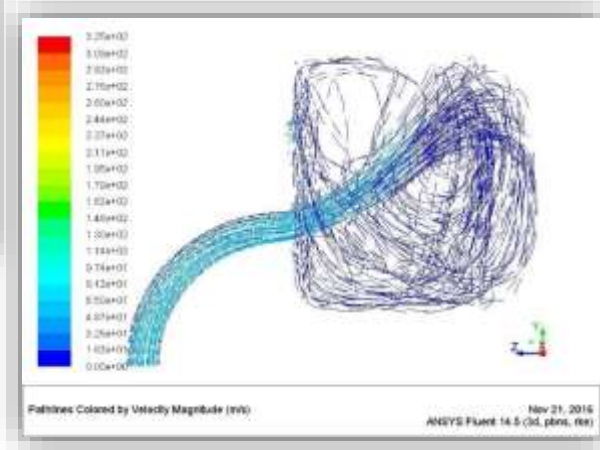
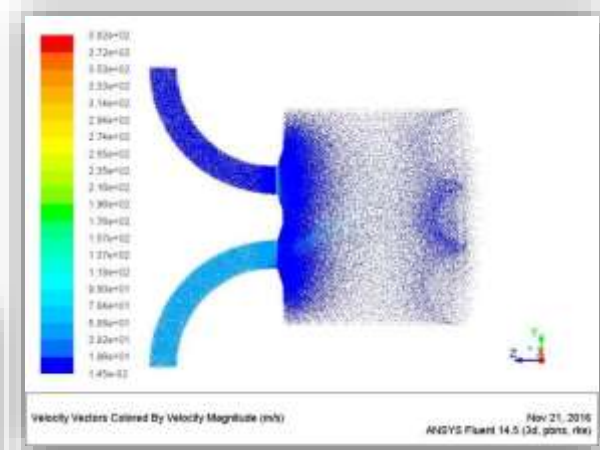
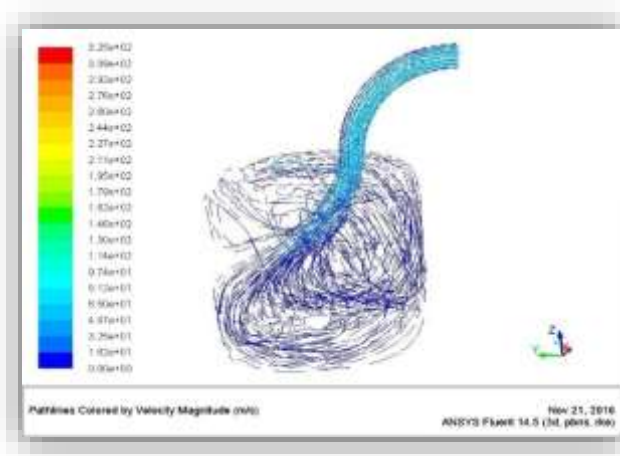
SOLIDWORKS MODEL:

With the use of modelling software we have generated two semicircular and one rectangular crowns over the piston head at appropriate positions. As we know design is also about estimation; we are taking hypothetical cases (knowing the fact that they are hard to manufacture and will incur high costs). We have made cut outs at the periphery of the piston head.



ANSYS SIMULATION RESULTS:

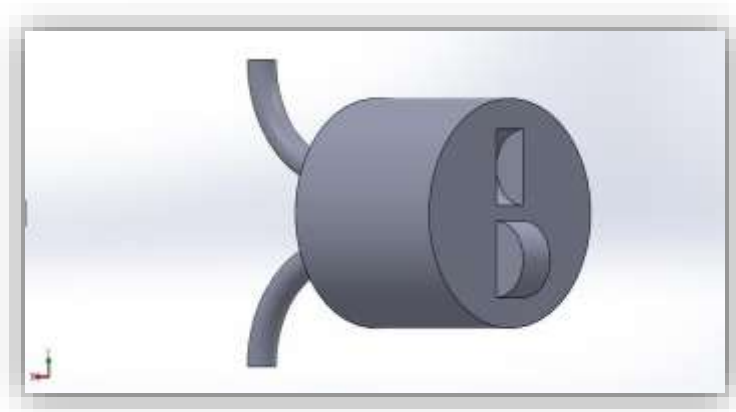
Here, we can clearly see the presence of vectors flowing over the crowns on the piston head. The obtained path lines tell us lot about the flow over the piston head and it is seen that the eddy currents are not being formed. But, we see that entire space is not effectively used. The mass density of air is less due to misaligned flow.



**CASE 8:
SOLIDWORKS MODEL:**

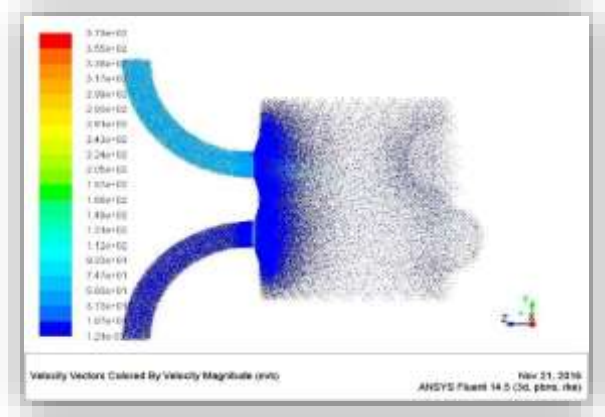
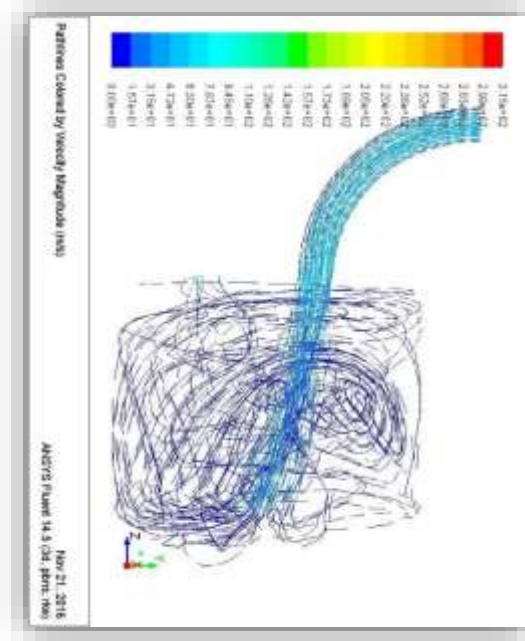
Here, the piston head is having crown and a depression. Now, we have deliberately placed crown nearer to the outlet valve to ensure inlet charge does not go un-burnt. Thus, the model is herewith for a clearer picture.

Again, it is a hypothetical case, as we want to reach to a justifiable conclusion of the best shape. And, in this process we want to innovate a new piston head which will help the intake air to flow in an organized manner.



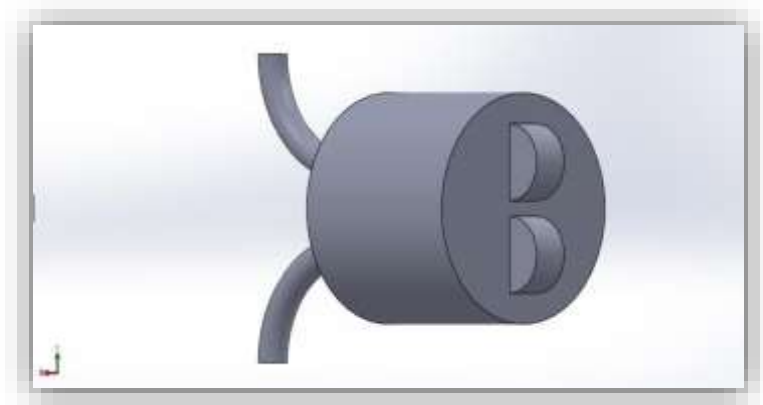
ANSYS SIMULATION RESULTS:

Now what we see are the vectors present all over the piston head geometry at a particular time. The path-lines formed show us that such configuration should be manufactured for use in near future, if at all it satisfies the failure criteria.



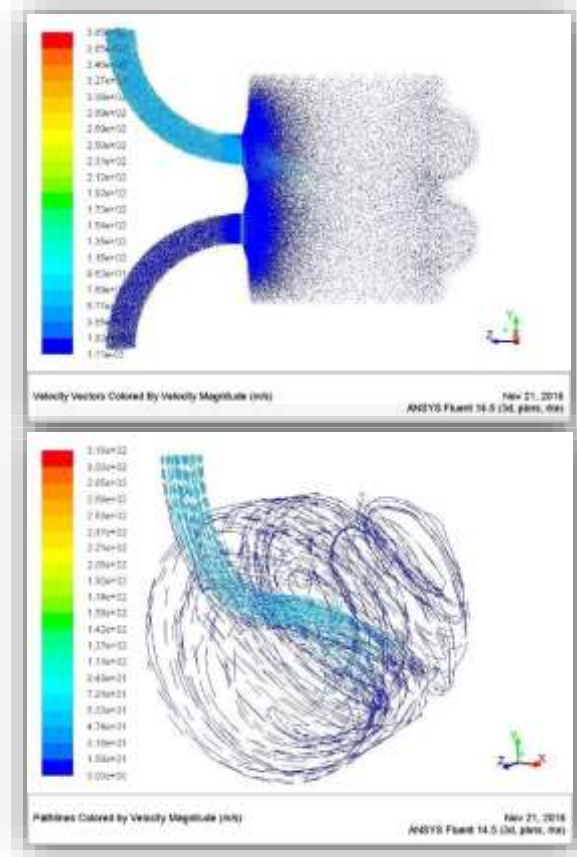
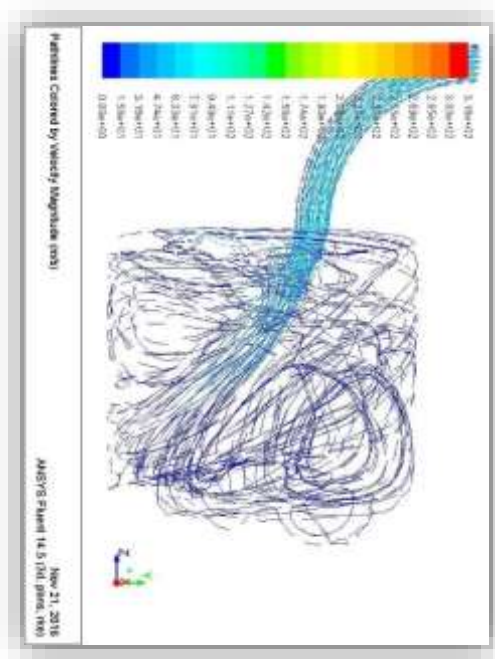
CASE 9:
SOLIDWORKS MODEL:

Here the depressions made are closer to each other and are deeper. Moreover, they are not perfect hemi spherical in shape. The design is attached herewith.



ANSYS SIMULATION RESULTS:

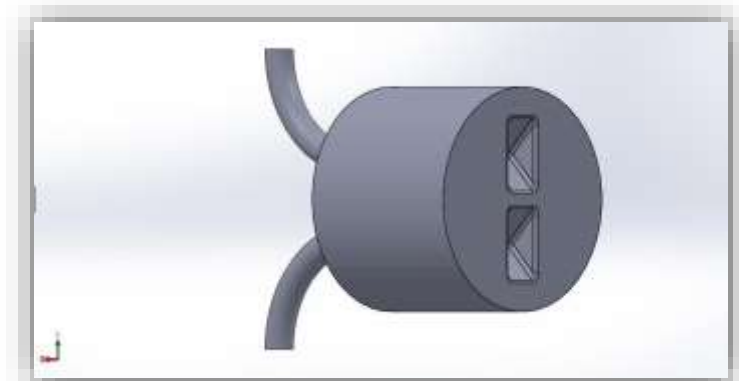
The vector formation alongside. Moreover, the change in velocity can be seen by change in the color (refer to the colored scale in the figure itself). The swirl formed over the piston head is also attached. It shows eddies near the bottom surface.



CASE 10:

SOLIDWORKS MODEL:

Here, our geometry includes two triangular crowns (as interpreted by Ansys). Moreover, they are located at an arbitrary distance and are of arbitrary height. Please note that, all the models are scaled up and size of piston crowns and depressions over the piston head is also scaled up. In reality,

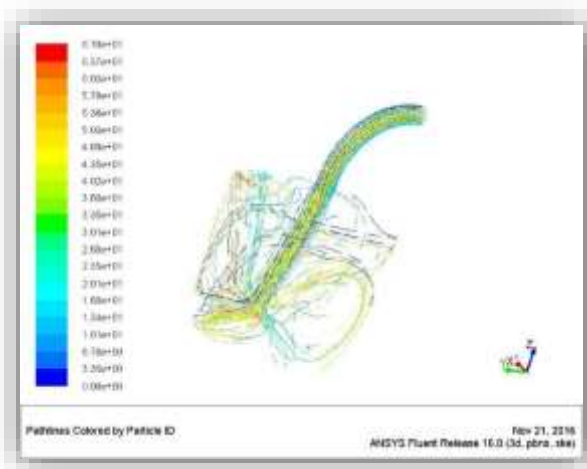
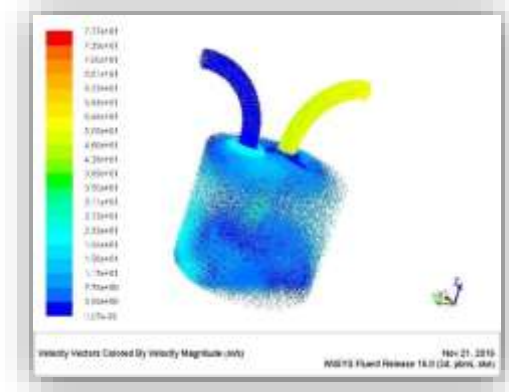


ANSYS SIMULATION RESULTS:

The vectors are shown where in heavy concentration of vectors is visible.

We can also see that the path lines are less dense and this case cannot be taken into consideration as the streamlines are striking each other.

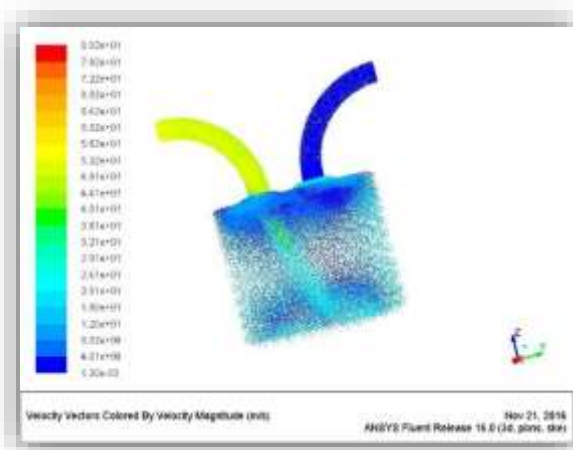
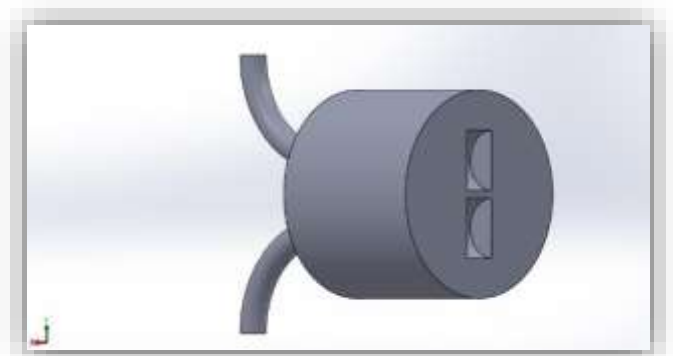
The change in velocity in various places can be seen by change in color. Moreover, highest velocity is also achieved momentarily after striking the first crown.



CASE 11:

SOLIDWORKS MODEL:

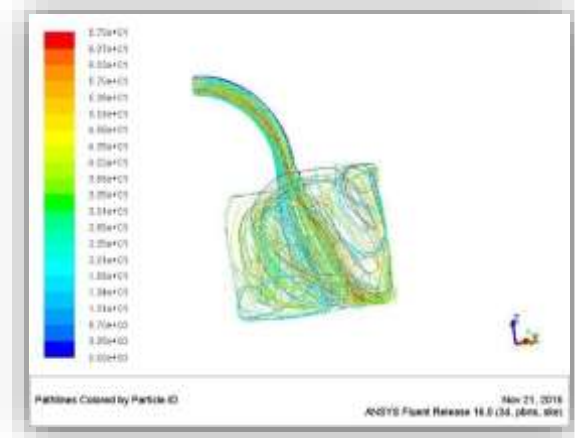
In this case we have made two hemispherical crowns over the piston head. And we are processing it in workbench to see the effect of the projections present over the crown



ANSYS SIMULATION RESULTS:

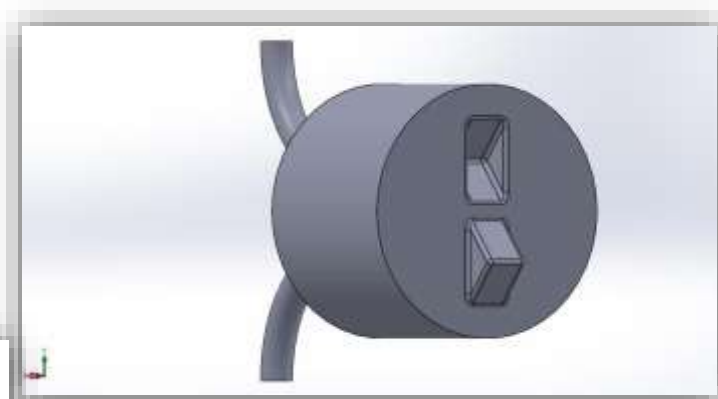
The vector path clearly shows that stream of air first travel around 50 m/s and strikes the piston crown. After striking the piston crown, the velocity is reduced drastically. The concentration of vectors in the entire space is a good indication for greater mass density.

The path lines formed are intermingling and scarce. Moreover, this pre characteristic of air flow will not follow a proper combustion. And, it being a weak induction swirl, will be neglected from our consideration. The intake air after striking the first crown gets deviated from its desired path. Moreover, the crown placed adjacent to it is not used effectively. With the use of pulse feature present in Ansys, one can easily see the transient flow of air from intake till the exit.



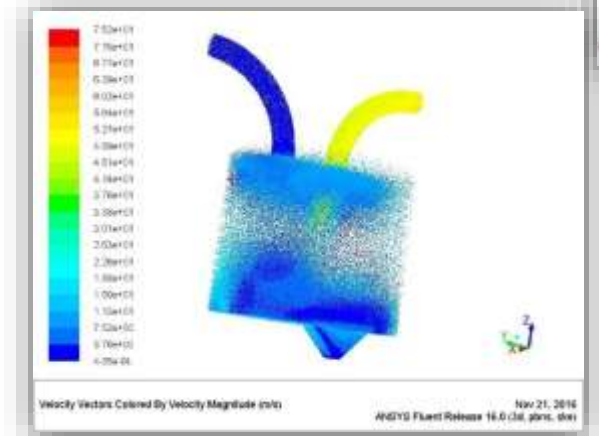
**CASE 12:
SOLIDWORKS MODEL:**

This case deals with a depression in piston head and a crown over piston head. Moreover, the position of depression and crown is done according to the symmetry.

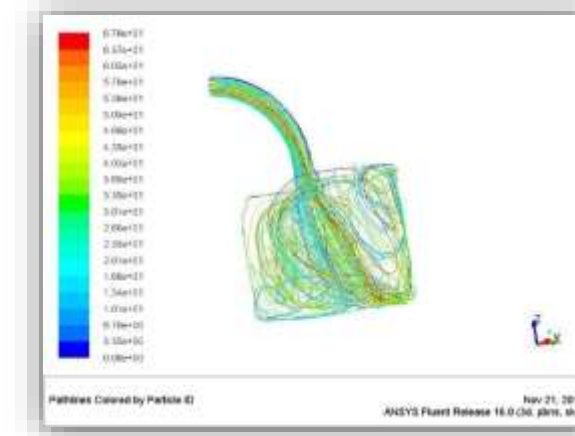


ANSYS SIMULATION RESULTS:

The vector generation is shown in the figure alongside which shows the distribution of vectors throughout the compression space.



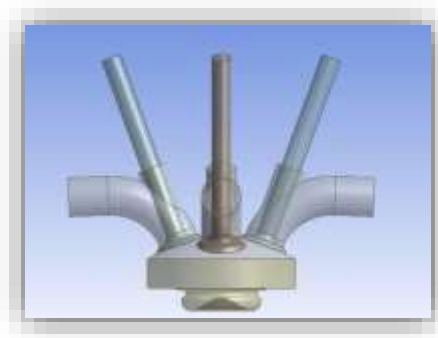
The path lines obtained are scarcely distributed and are not strong enough to produce strong combustion. Thus crowing and depression doesn't work well when manufactured together.



CASE 13:

SOLIDWORKS MODEL:

This is the Solidworks model where in two circular depressions are made which will influence the air motion subsequently. The inlet and outlet valves are removed via Boolean operation in Ansys.

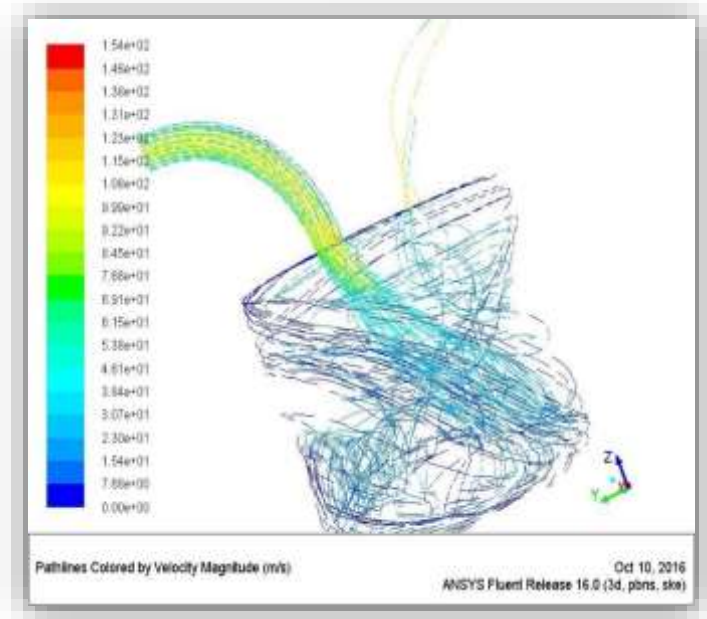


ANSYS SIMULATION RESULTS:

This is by far the best swirl generated where in there is effective utilization of space is observed. Moreover, the eddy formation has been reduced. Also, the mass density of air flow is maintained. A strong thick turbulence can be seen just after the intake manifold.

Also, with reference to other research papers which deals with the shape of piston head, it is the perfect shape of the piston head which involves two hemispherical depressions over the piston head.

Also, the case 9 which is similar to this case can be seen for reference which indicates that presence of strong organized turbulence is observed when two circular depressions are used simultaneously.



CONCLUSION

The cases discussed above have shown what happens when air goes inside the intake manifold. Moreover, CASE 13 is considered to be the best because:

- i. It is achieving organized strong turbulence in the test section.
- ii. The depressions is helping the air flow to form swirl and also help in avoiding unsteadiness in the flow.
- iii. The shape is such that it doesn't form large eddies.
- iv. The shape designed will have uniform flow eventually.

The in-cylinder charge motion often plays a dominant role in preparation and conveyance of fuel mixture in the engine. The production of high turbulence intensity is one of the most important factors for stabilizing the ignition process and fast propagation of flame, especially in the case of lean-burn combustion. This paper outlines the process of design optimization of piston head using Computational Fluid Dynamics (CFD). The parameters that were varied were the location, shape, size and number of the depressions or crowns over the piston head. The basis of optimization was flow uniformity throughout the component, prevention of separation of air flow in the piston head geometry.

ACKNOWLEDGEMENTS

In writing this paper, we had taken the help from some respected persons, who deserve our heartfelt thankfulness. The completion of this paper leaves us self-contented. We would like to extend our heartiest gratitude towards Mr. Praveen Garg & Mrs. Seema Garg, for giving us support in our failures and channelizing us to present this paper in a respectable manner. We would also like to expand our deepest gratitude to all those who have directly and indirectly guided us in writing this assignment. This section should be typed in character size 10pt Times New Roman, Justified.

REFERENCES

- [1] Piston ring tribology A literature survey|| Peter Andersson VTT Industrial Systems Jaana Tamminen & Carl-Erik Sandström Helsinki University of Technology, Internal Combustion Engine Laboratory.
- [2] A Study of Friction and Lubrication Behavior for Gasoline Piston Skirt Profile Concepts Kwang-soo Kim, Paras Shah Federal-Mogul Corporation Masaaki Takiguchi / Shuma Aoki Musashi Institute of Technology.
- [3] H.Sushma, doing master's degree in Thermal power engineering at B.T.L.Institute Of Technology And Management, Bangalore which affiliated to Visvesvaraya Technological University, Belgaum-590014, India.
- [4] https://mechanicalengg.com/uploads/monthly_2017_02/piston.jpg.9f7e7a737d910e6d9e450d4ae17db28c.jpg
- [5] <https://qph.ec.quoracdn.net/main-qimg-35247606ecc947d142a667a07259507f-c>