International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 09 Issue: 06 | June 2022 www.irjet.net p-ISSN: 2395-0072

DESIGN AND ANALYIS OF INLET MANIFOLD WITH VORTEX GENERATOR IN GDI ENGINE

Aswin Rose Thomas[1], Aswin Saju[2], Berin M Biju[3], Eldho Babychan[4]
Asso. Prof. Dr. Rabi Johnson[5]

[1,2,3,4] Btech students, Department of Mechanical Engineering, Mangalam College Of Engineering ,Kerala, India686631
[5,]Faculty, Department of Mechanical Engineering, Mangalam College Of Engineering ,Kerala, India-686631

Abstract -*Gasoline direct injection (GDI) is an increasingly prevalent fuel injection system for passenger cars worldwide, where the growing demand for low fuel consumption and stricter emission limits are cause for innovative engine concepts. Engines with gasoline direct injection create an air fuel mixture right inside the combustion chamber. This results in better combustion characteristics than conventional carburetor engines thus improved efficiency, torque and dynamic driving characteristics, while emission levels are reduced. A Gasoline Direct Injection machine operates on spread mixture to minimize the NOx emigrations due to inordinate heat release in stoichiometric combustion. Lean combustion in a GDI engine requires high turbulence inside the combustion chambers to graese mixing and vaporization of fitted energy .*

Engine requirInlet manifold design plays a major role in the generation of turbulence and vortices at the initial stages of suction stroke. In this project a GDI engine with bore X stroke of 82.5 mm X 84.2 mm will be modelled to generate the flow domain. The model uses recent features of GDI engine involving a pent roof cylinder head design and 4 valve head to enhance better mixing and turbulence inside the cylinder. The present work is aimed to further increase the turbulence inside the combustion chamber by incorporating vortex generators in the inlet manifold to further enhance the turbulence inside the combustion engine. These vortex generators sheds vortex from the tip of the curved vanes, thereby improving mixing characteristics thereby enhancing combustion. Since in GDI engine air-fuel mixture is created inside the combustion chamber, changes in inlet manifold designs doesn't cause fuel or carbon accumulation. Due to stricter emission norms and growing trend of GDI engines, inlet manifold designs could enhance the turbulence in combustion chamber. Three different vane configurations(2 vane, 3 vane and 4 vane configuration) will be designed and numerically analysed using ANSYS FLUENT. The performance of the modified inlet manifold design will be compared against conventional design to evaluate the turbulence enhancement obtained. Steady state analysis of suction stroke is carried out using ANSYS FLUENT

1.INTRODUCTION

Gasoline direct injection(GDI), also known as petrol direct injection(PDI), is a admixture conformation system for internal combustion machines that run on gasoline(petrol), where energy is fitted into the combustion chamber. This is distinct from multifarious energy injection systems, which fit energy into the input manifold. The first GDI machine to reach product was introduced in 1925 for a low-contraction truck machine. Several German Buses used a Bosch mechanical GDI system in the 1950s, . GDI has seen rapid-fire relinquishment by the automotive assiduity in recent times, adding in the United States from2.3 of product for model time 2008 vehicles to roughly 50 for model time 2016.

The inflow field characteristics inside the machine cylinder play an effective part in the combustion process in petrol machines. Turbulence increases the mixing of the air and the energy which improves the combustion effectiveness and reduces the machine emigrations. The inflow characteristics inside the cylinder, which are directly, linked with the machine performance and emigration characteristics. The ideal of this work is to study swirling improvement by modifying the bay manifold with a simple wedge- shaped Whirlpool creators(VGs) attached circumferentially over the inner face of the manifold at the entry position. The inflow in the input manifold is presented, it's one of the central corridor of the machine. The part of the input manifold is to give a slightly distributed air inflow to the cylinders, rational design can reduce the bay pressure losses also adding the air volume introduced in machine. adding the bay multifarious effectiveness is a major challenge to increase the machine overall effectiveness, in this way the emigrations can be reduced. Volumetric effectiveness of the machine is a measure of the effectiveness of the air input system composed by input manifold, input harbourage and cylinder. This means that the haste of air in the bay manifold is adding further air the intake system can deliver to the machine. This will increase the volumetric effectiveness, this effect can increase the necklace and the power of the machine

2. LITERATURE REVIEW

P S Mehta(18 July 2001) *Internal Combustion Machines Laboratory, IndiaInstitute of Technology, Madras, India. The* donation of charge stir in internal com bustionengines towards perfecting their performance and Emigration characteristics is well honored. Tumble stir is a rather lately linked organized rotary charge stir being in an axial aeroplane . Through the product of a well- timed for turbulence improvement through tumble ated turbulence an optimized spill charge has been synthesized and related to these stages. stir can enable better combustion in spark A primary parametric study with input stopcock lifts ignition(SI) Machines, indeed at high situations of charge. It's revealed that aation and also finds wide use in spare burn machine large-angled pentroof retains significant whirlpool structures improvement is studied numerically for a direct injection SI machine with whirlpool creators placed in the bay manifold, ANSYS FluentTM. The three-dimensional figure and mesh are created using pre-processor ANSYS ICEM. Simulations have been carried out to probe the effect of whirlpool creator in bay manifold for a single cylinderengine. It's observed that the modified ba manifold creates invariant mixing inside the machine cylinder which is essential for effective combustion for diesel engine. And landing the inflow patterns inside the cylinder using experimental ways is expensive and it's delicate to carry out the parametric studies in different combination of bay manifolds, piston top surfaces. So the ANSYS is only effective analysis tool to study the inflow geste with colorful manifolds. The approach was used to increase the swirling inside cylinder by changing the piston top face with different coliseum shapes and coliseum positions. An experimental study to enhance swirling by using curve control stopcock(SCV) with different SCV angles and presented Inflow patterns and curve number for different SCV angles. It was that this revision enhances the swirling characteristics inside the cylinder. All the figure shapes and medium tried by colorful researchers have difficulties in manufacturing and it would increase the total vehicle cost.

Analysis of swirl enhancement in diesel engine with vortex generator

*G.sivakumar and S.Semthilkumar*studied numerically for a direct injection diesel machine with whirlpool creators placed in the bay manifold using commercial CFD law, ANSYS Fluent TM. The three- dimensional figure and mesh are created using pre-processor ANSYS ICEM. Simulations have been carried out to investigate the effect of whirlpool creator in bay manifold for a single cylinder creation inside the machine was before studied by colorful experimenters by changing the parameters like multifarious shape, combustion chamber configurations, and piston head shape. There are two ways of enhancing the combustion process thereby adding the machine thermal effectiveness. First one is by adding the contraction rate as high as possible. But the problem is that high contraction rate may produce knock, which should be avoided for good performance. Alternate bone is by enhancing the turbulence in order to have a better mixing of energy and air. They experimentally studied, using fly speck image velocimentry (PIV), the effect of coliseum shape on the top piston face, and set up that coliseum shape onflat piston shows a good enhancement. Jin etal. carried out an experimental study to enhance swirling by using curve control stopcock(SCV) with different SCV angles and presented inflow patterns and curve number for different SCV angle

The Investigation and Application of Variable Tumble Intake System on a GDI Engine

J.Engines(2014), clear energy automotive care The in-cylinder spill intensity of GDI machine is pivotal to combustion stability and thermal effectiveness with a flap valve in intake manifold, the mean velocity and turbulence kinetic energy all were almost twice than those of other Cases when piston close to TDC. With the development and improvement of GDI technology, the various ways of airflow motion in cylinder successively emerged, such as piston top wall guidance, air charging motion of intake port and injector spray guidance, etc.

The basic advantages of higher tumble ratio exist in the following items:

(1) Reduced fuel spray penetration, i.e. wall wetting.

(2)To form well- distributed mixture.

(3) Better combustion stabilization.

(4) Even faster flame propagation velocity. the tumble ratio of GDI engine is desired to be on higher level to produce more uniform mixture, to meet different requirements of engine operation condition.

Therefore ,a new variable tumble system should be applied to GDI engine. The variable tumble system was considered as an effective way to change the in-cylinder tumble intensity.

Influence of swirl, tumble and squish flows on combustion characteristics and emissions in internal combustion engine

Muhamut Kaplan Amasya university(October2019)This Study emission reduction. Characteristics of in-cylinder flows Swirl, tumble and squish flows enhance turbulence intensity during late compression by breaking down these flows to small scale turbulent eddies. This provides increase of turbulent flame speed and so acceleration of burning rate. Swirl is used to speed up to combustion process in SI engines and to increase faster mixing between air and fuel in CI and some stratified charge engines . This flow regarded as a two dimensional solid body rotation is generated by intake system. In spite of some decaying due to friction during the engine cycle, it usually continues through the compression, combustion, and expansion strokes. The swirl flows require energy to produce the vortex during the intake stroke.

Full-Parameter Approachfor the Intake Port Designof a Four-Valve Direct-Injection Gasoline Engine

Lei cui ,Ming jia (November 2015)Compared with the traditional methods, parametric approach attracts increasing attentions by virtue of its high-efficiency, traceability, and flexibility. The evaluation of the flow characteristics of the intake process mainly includes flow capacity and the ability to generate a tumble motion. For a well-designed tangential intake port, it needs a large flow coefficient and an appropriate tumble intensity to ensure a favorable air motion. The reason for the performance change with valve lift can be explained by the flow field feature. The overall flow filed can be divided into two parts: one part moves down the wall in the right side of the cylinder in clockwise direction; the other part moves along the combustion chamber and cylinder wall in the left side in anti-clockwise direction. When a large scale anticlockwise vortex is formed This vortex can contribute to the increased tumble intensity and the decreased flow coefficient to some extent.

Combustion chamber design for a highperformance natural gas engine

Mirko Baratta, Daniela Misul, the design of ultramodern internal combustion(IC) machines represents a grueling task, due to the raising concern for the global warming as well as to the strict constraints set by the current contaminant regulations. Engines, the spill stir is generally generated in order to increase the turbulence position in the combustion chamber, therefore enhancing the combustion stability and the exhaust gas recirculationtolerance.As far as the influence of the chamber design is concerned, for low and intermediate stopcock lift values the presence of the masking wall gives rise to a drop of the stopcock discharge measure, whereas the spill number is increased up to two- three times its original value, due to the rear spill inhibition bandied over. At high lift, the input stopcock results to be displaced beyond the extension of the masking face, accordingly its effect nearly disappears as is witnessed by the similar values of both CD and NT for the birth and the ‘ masked ’design. Overall, the presence of the masking face determined a benefit in the turbulence intensity at spark timing in nearly all the cases at partial Cargo

Numerical Methodology

Numerical simulations under isothermal conditions have been carried out for a single cylinder CI machine with modified bay manifold using whirlpool sphere and mesh structure of the problem respectively. For simplicity, simulations are performed with completely opened stopcock condition at the bay harborage and completely unrestricted stopcock condition at the exhaust harborage. Pressure bay boundary condition is specified for the starting face of the bay manifold. No- slip boundary conditions are specified

at walls of the machine cylinder. - dimensional tetrahedral type mesh is created for the figure of the problem. Pressure haste coupling was done using SIMPLEC pressure correction system. Unsteady calculations are performed by an implicit time discretization within the sphere using incompressible

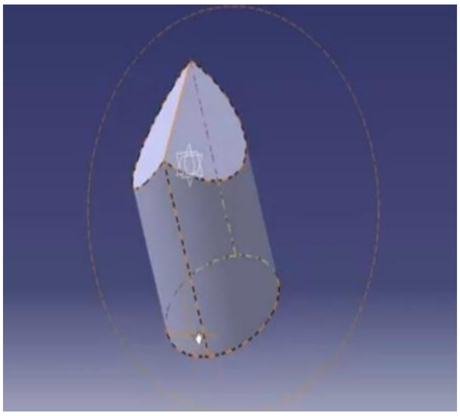
Reynolds — Averaged Navier – Stokes equations with RNG k- e turbulence model with available with marketable software ANSYS FLUENTTM. The Diffusion terms are discretized with alternate- order central scheme and the convective terms are with alternate- order upwind scheme. For all the calculations, residuals of durability, instigation, and turbulence kinetic energy equations are covered, and the

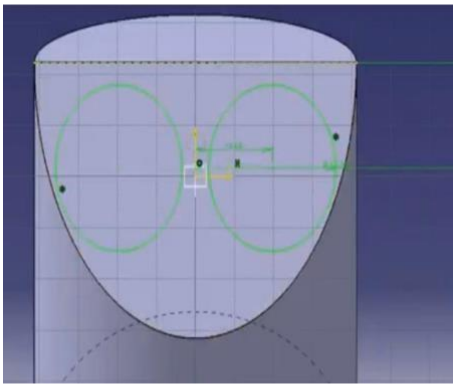
confluence criterion value used equal to 10- 4. In order to quantify the effect of turbulent creation inside the cylinder, it is necessary to calculate the curve number. The is the rate of the angular instigation to the axial instigation. Figure shows the variation of curve number along the stroke length for two bay multifarious configurations, with and without whirlpool creators. It's seen that the curve number for with

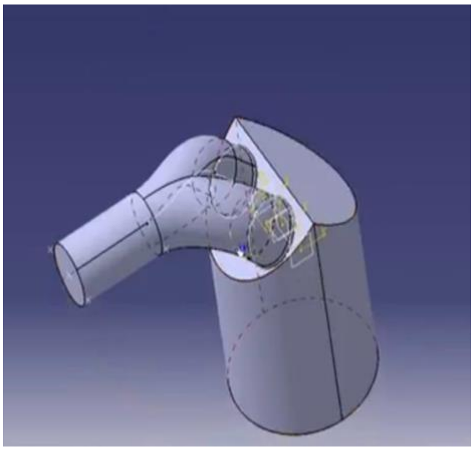
VGs is advanced than that for without VGs, which shows the impact of VGs on swirling improvement at all the positions along the stroke length.

Geometry Modelling

Here We are using CATIA software for designing the model. We have to study the region where the flow is occurring.

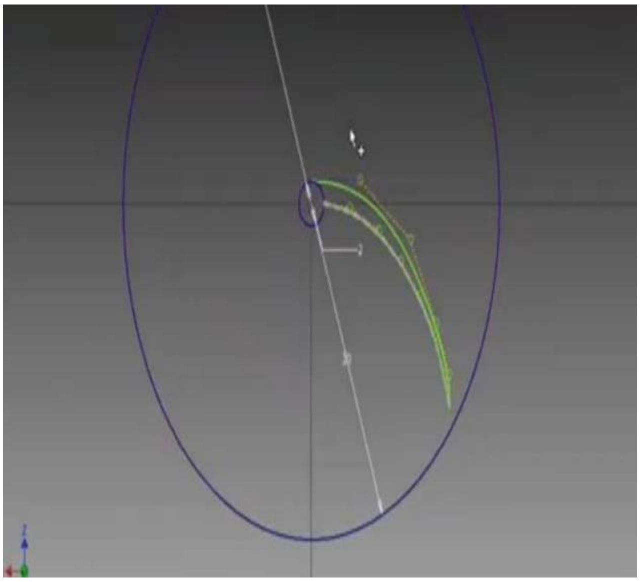


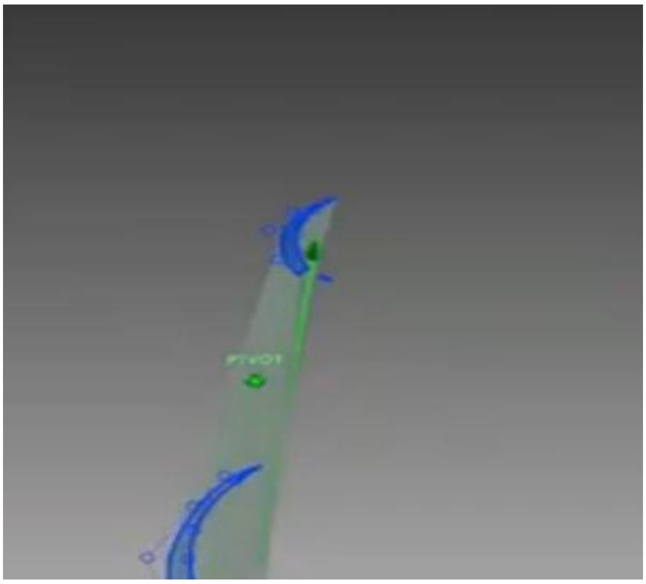




Design of vein

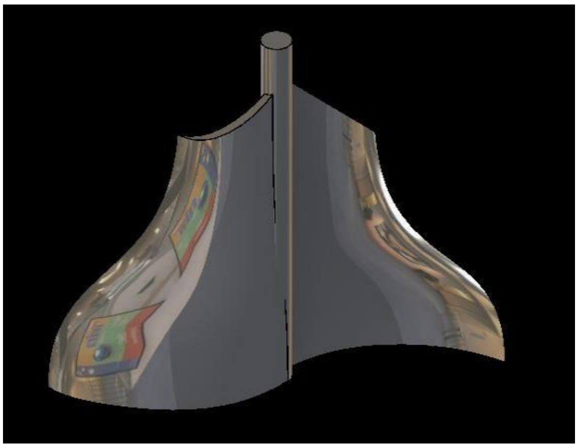
For designing the blade we are using autodesk inventor software.





2 Vane

Here 2 vanes are connected to the shaft and placed in the inlet manifold

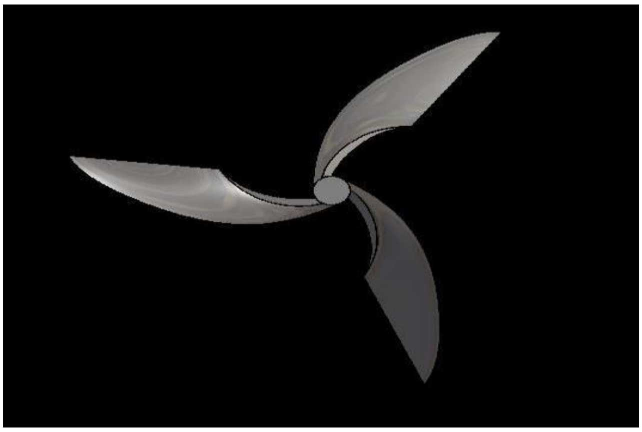


Isometric.view

3 vane

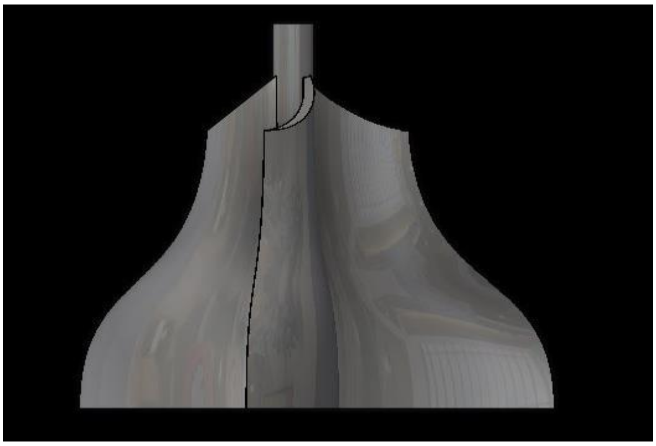
Here 3 vanes are connected to the shaft and placed in the inlet manifold

Isometric.view



Top.view

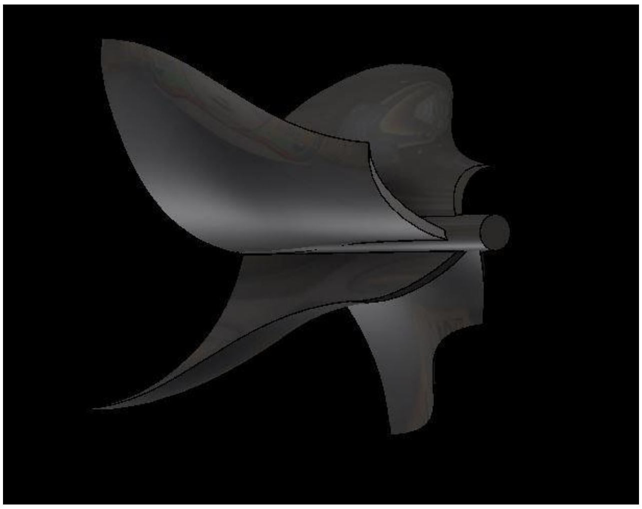
Top.view



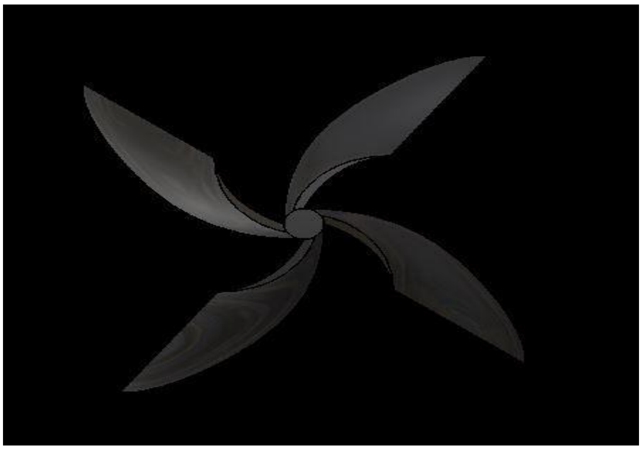
Top.view

4 vane

Here 4 vanes are connected to the shaft and placed in the inlet manifold



Isometric.view



Governing Equations

The following governing equations are used for the analysis;

- Navier stokes equation
- Continuity equation.

Navier stokes equation

In the case when we consider an incompressible , isothermal Newtonian flow (**density =const) viscosity** μ **=const**), with a velocity field *V* = (*u*(*x*,*y*,*z*), *v*(*x*,*y*,*z*), *w* (*x*,*y*,*z*)) we can simplify the **Navier-Stokes equations** to his form:

x component:

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)=-\frac{\partial P}{\partial x}+\rho g_x+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)$$

y- component:

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)=-\frac{\partial P}{\partial y}+\rho g_y+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)$$

z component:

$$\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)=-\frac{\partial P}{\partial z}+\rho g_z+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)$$

Continuity equation

Rate of change of mass within the control volume is equal to the difference of rate of change of mass which enters the control volume and the rate of change of mass which leaves the control volume.

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

K-Omega SST

The the model used for the simulation is K- Omega model. The k- omega(k – ω) turbulence model is one of the most generally used models to capture the effect of turbulent inflow conditions. It belongs to the Reynolds- equaled Navier- Stokes(RANS) family of turbulence models where all the goods of turbulence are modeled. Then SST stands for Shear Stress Transport It's a two- equation model. That means in addition to the conservation equations, it solves two transport equations (PDEs), which regard for the history goods like convection and prolixity of turbulent energy. The two transported variables are turbulent kinetic energy(k), which determines the energy in turbulence, and specific turbulent dispersion rate(ω), which determines the rate of dispersion per unit turbulent kinetic energy. ω is also appertained to as the scale of turbulence.

Meshing

The purpose of meshing is to actually make the problem solvable using finite element. By meshing the domain is breaked into subdomains ,each subdomains representing an element.

Object Name	*Mesh*
State	Solved
Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Element Order	Linear
Sizing	
Size Function	Proximity and Curvature
Relevance Center	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Num Cells Across Gap	Default (3)
Proximity Size Function Sources	Faces and Edges
Min Size	Default (6.7641e-002 mm)
Proximity Min Size	Default (6.7641e-002 mm)
Max Face Size	4.0 mm
Max Tet Size	5.0 mm
Growth Rate	Default (1.20)
Automatic Mesh Based Defeaturing	On
Defeature Size	Default (3.382e-002 mm)
Minimum Edge Length	0.130660 mm
Quality	
Check Mesh Quality	Yes, Errors
Target Skewness	Default (0.900000)
Smoothing	Medium
Mesh Metric	None
Inflation	
Use Automatic Inflation	None

Simulation

For simulation activities we are using ANSYS software. The requirements of the boundary conditions will be asked according to the Name information in meshing.

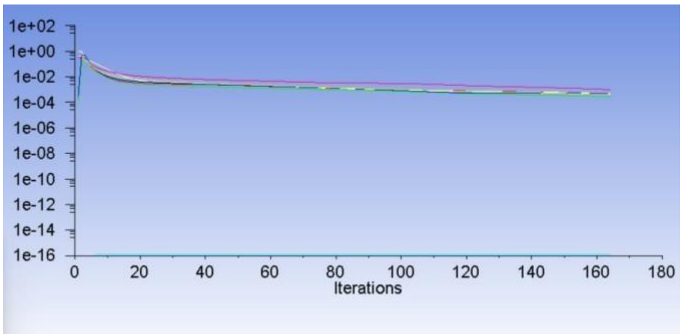
*Cylinder= wall

* inlet velocity=11.26m/s

*temperature= 300k

*outlet = wall

* manifold=wall



. residual plot

- White line = continuity residual
- Red = x velocity
- Green = y velocity
- Blue = z velocity

From this output residual plot we can understand that the simulation is stable.

Result and Analysis

we use CFD Post for result and post processing. Here we compare the results of 2 vane ,3 vane and 4 vane against the baseline

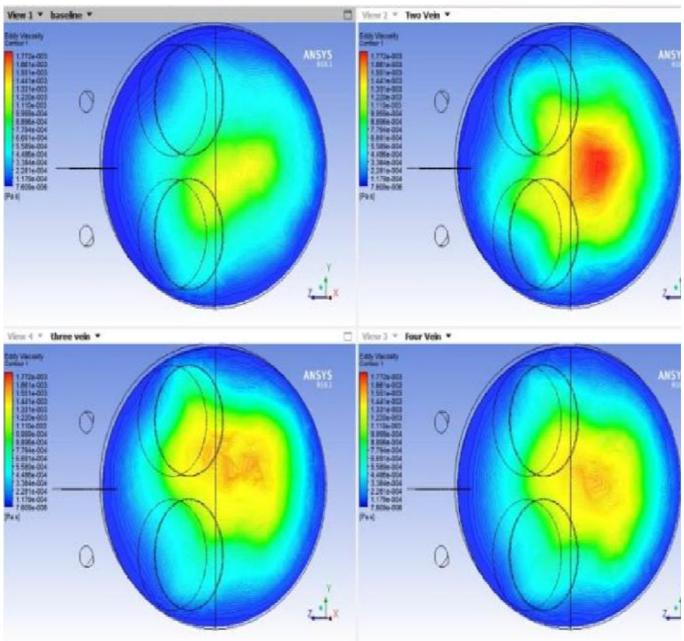
Initially we have constructed a plane near the TDC and analysis is done at that plane, since air fuel mixing starts at TDC.

Contour

Contour is colour map which are used to find change in the properties such as pressure ,kinetic energy etc according to colur variations.

Eddy viscosity

After selecting countor we select the variable as Eddy viscosity,and then select the number of contours as 100.and after loading we will get a result as shown in the figure.

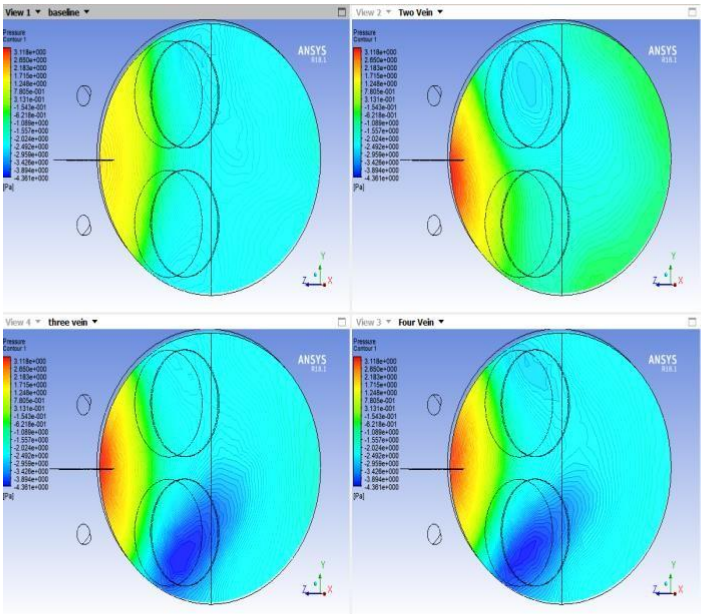


Contour of eddy viscosity

From the figure it is clear that eddy viscosity is maximum for 2 vane . If eddy viscosity is high , it means the there is high turbulence..

pressure

Now we apply pressure as the variable. After running we will get get a result as shown in the figure.



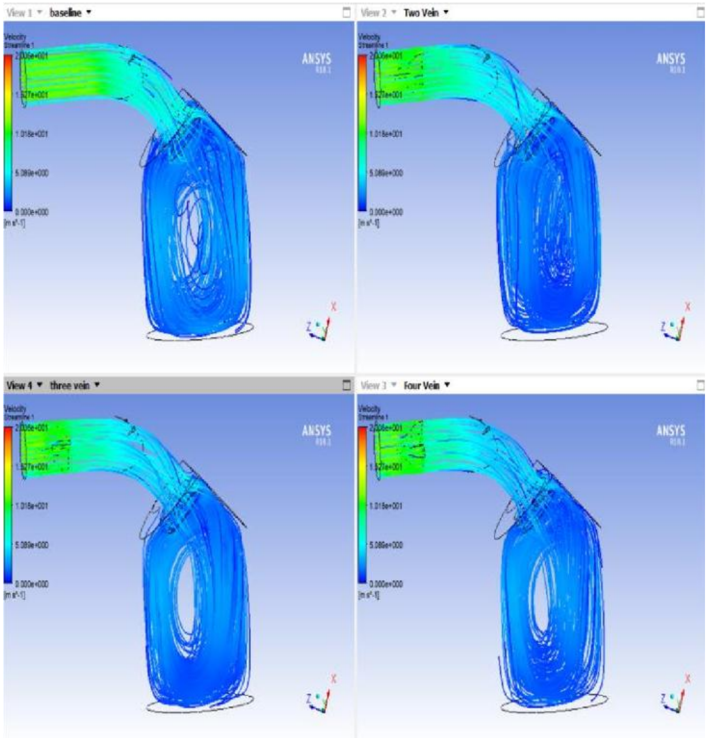
From the figure it is clear that pressure is maximum for 2 vane compared to baseline ,3 vane and 4 vane.

Streamline

Streamline shows the path of flow of air. We have select the starting point as inlet. Number of points is taken as 300 for easy analysing, and the variable is selected as velocity.

The result is obtained as shown int the figure. Velocity streamline

Velocity streamline



we can see that combined rotation in X,Y,Z axis is maximum for 2 vane compared to baseline ,3 vane and 4 vane.so turbulence will be more for 2 vane.

Graphical Result

Here we construct a line in location. then we define initial and final point. then we select the location as li9ne 1. x-axis represent variation from TDC-BDC.

Y axis represent variables.

Eddy viscosity

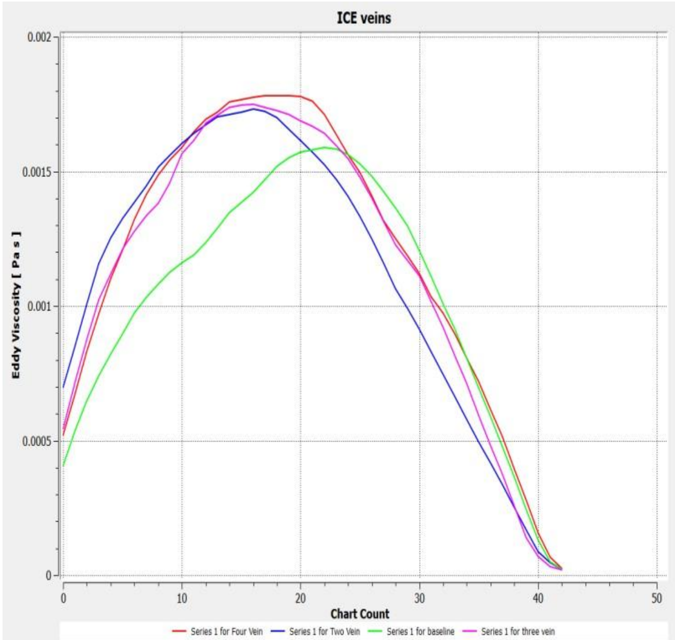
After selecting graphical result, we select the variable as Eddy viscosity. and after loading we will get a result as shown in the figure.

From the graph it is clear that eddy viscosity is maximum for

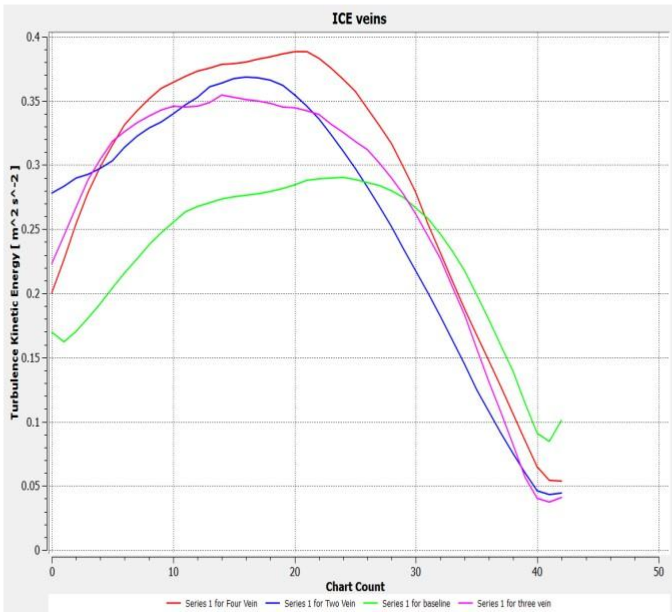
2vane at the beginning compared to baseline, 3vane, 4vane. so we get more turbulent characteristics.

Turbulent kinetic energy

After selecting graphical result, we select the variable as Turbulant kinetic energy. and after loading we will get a result as shown in the figure.



Graph result of eddy viscosity

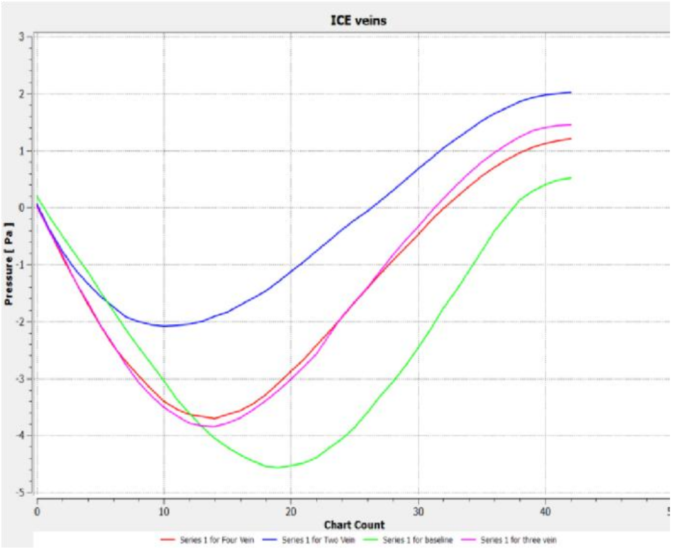


Graph result of turbulent kinetic energy

From the graph it is clear that turbulent kinetic energy is maximum for 2vane at the beginning compared to baseline,3vane,4vane.so we get more turbulent characteristics.

Pressure

After selecting graphical result, we select the variable as Pressure, and after loading we will get a result as shown in the figure.



Graph result of pressure

From the graph it is clear that pressure is maximum for 2vane compared to baseline,3vane,4vane.

Conclusion

So we can conclude that a 2vane can enhance the performance of a GDI engine compared to three vane and 4 vane.A 3vane is assymetric so vortex get interact each other ,which wont happen in case of 2 vane and 4 vane since they are symmetric. So there will be an increase in the torque of the vehicle. We will get better air fuel mixing due to the turbulent flow air to the cylinder, due to which the amount of fuel which remains unburnt will be very less and the pollution is reduced.

Reference

Velte CM, Hansen MOL, Okulov VL (2009) Helical structure of longitudinal vortices embedded in turbulent wall-bounded flow. J Fluid Mech 619:167–177

Paul B, Ganesan V (2010) Flow field development in a direct injection diesel engine with different manifolds. Int J Eng Sci Technol 2(1):80–91 Martins J, Teixeira S, Coene S (2009) Design of an inlet track of a small IC Engine for swirl enhancement, In: Proceedings of the 20th international congress of mechanical engineering, Gramado, 15–20 Nov 2009 Murali Krishna B, Mallikarjuna JM (2009) Tumble flow analysis in an unfired engine using particle image velocimetry, In: Proceedings of world academy of science, engineering and technology, vol 30

Lee J-W, Kang K-Y, Choi S-H, Jeon C-H, Chang Y-J (2000) Flow characteristics and influence of swirl flow interactions on spray for direct injection diesel engine, In: Seoul 2000 FISITA world automotive congress, Seoul, 12–15 June 2000

Prasad BVVSU, Sharma CS, Anand TNC, Ravikrishna RV (2011) High swirl-inducing piston bowls in small diesel engines for emission reduction, Appl Energy 88:2355–2367

Versteeg HK, Malalasekhara W (1995) An introduction to computational fluid dynamics: the finite methods. Longman Group Ltd, London

Auriemma M., Caputo G., Corcione F.E., Valentino G. and Riganti G. 2003.