CFD modeling with comparison between swirl pipe and GVSTD

Yasar Khan  
Assistant Professor, Department Of Mechanical Engineering,  
Dev Bhoomi Institute Of Technology, Dehradun, Uttarakhand, India

Tirath Singh  
Assistant Professor, Department Of Mechanical Engineering,  
Dev Bhoomi Institute Of Technology, Dehradun, Uttarakhand, India

Vijay Singh Chauhan  
Assistant Professor, Department Of Mechanical Engineering,  
Dev Bhoomi Institute Of Technology, Dehradun, Uttarakhand, India

Faraz Ahmad  
Assistant Professor, Department Of Mechanical Engineering  
Dev Bhoomi Institute Of Technology, Dehradun, Uttarakhand, India

KEYWORDS: guide vanes, swirl pipe, swirl ratio, compression ignition(CI) engine, CFD, turbulent kinetic energy(TKE), ANSYS Fluent.

ABSTRACT
In the fluid flow one of the aspect that the engineers and scientists tries to keep low is the turbulence, because turbulence is the randomness and it makes the flow uncontrolled by creating randomness. Due to turbulence there is loss of pressure head. Despite of many drawbacks turbulence finds its place in some applications. This paper includes the CFD modeling in ANSYS FLUENT 16.0 of swirl pipe and guide vane swirl and tumble device (GVSTD) to create turbulence for some applications, the designs were made in CATIA V5R18. The main emphasis in this paper is kept on the uniformity of air-fuel mixture in CI engine. To achieve maximum output and least emissions there is the need to minimize the ignition delay and there should be complete combustion. For the complete combustion one should generate the better air flow inside the cylinder. The two different geometries were made to generate better air flow. The factors that decide that the air flow is good are tangential velocity, turbulence kinetic energy and swirl ratio. The objective is to generate swirling motion in the air in CI engine. Also it finds the application in the alternate fuel engines due to the low volatility and less tendency of vaporization of alternate fuels.

INTRODUCTION
The very important thing that plays an important role in our lives is the energy. But without conversion of energy its use is very limited therefore to make the use of energy more versatile, there is need to work in the area of energy conversion. After research of many decades we are successful to some extent in achieving our goal. But now there is a need to achieve more efficiency in energy conversion.

One of the important mechanical devices that convert energy is an IC engine. For the very first time the term IC engine came into existence was in 1680, when an internal combustion engine was developed and uses gunpowder as the fuel by the Dutch physicist, Christian Huygens, later four-stroke engine in 1876 was patented by Nikolaus August Otto, known as “Otto cycle”. Another important invention of that time is of Rudolph Diesel in 1892 who designed more heavier and powerful engines than gasoline engines.

In general a CI engine is better than SI engines in terms of performance and efficiency but creates much pollution and noise. CI engines have higher compression ratio that means they have better thermal efficiency, power output of CI engines is also better than SI engines. But a CI engine emits more pollution than SI engines this one factor limits its use in...
light vehicles. Many attempts were made and still going on to improve CI engines. Besides this the ignition delay is the major problem in CI engine.

Ignition delay leads to the charge accumulation in the combustion zone and leading to abnormal combustion producing extreme pressure and vibrations. Also the ignition delay leads to incomplete combustion hence increasing emissions. These problems could be rectified through some factors such as on increasing compression ratio of air due to increased density of the compressed air the temperature of air increases and the auto-ignition of fuel begins. This results in good inter-mixing and closer contact between air and fuel molecules and hence reduces the time of reaction, a good combustion at appropriate crank angle always incorporates the good conversion of thermal energy.

One more aspect is important in this type of engine i.e. good air flow system. CI engine incomplete intermixing of air and fuel molecules results in incomplete combustion or uncontrolled combustion in both the cases the results are worst. In incomplete combustion there is loss of fuel and power both and uncontrolled combustion results in rough working of engine as due to non-uniform mixture the amount of fuel accumulation in certain position in cylinder will be more and combustion of more amount of fuel will results in abrupt change in the power.

To meet these parameters some work had been carried initially, Ricardo et.al [1] is the first to study the significance of turbulence on combustion and heat transfer in the IC engine. He particularly worked on the effect of increasing flame speed on knock. Bharadwaj et.al [2] studied swirl and tumble flow motion using CFD and they observed that swirl flow requires energy to generate the vortex during the suction stroke and the energy required by swirl flow comes from the kinetic energy of the air entering the cylinder through inlet valve.

Kamal et.al [3] carried out CFD analysis in order to design and test a different geometry changes in inlet valve. They provided a curved blades on the neck of poppet valve and preferred them over the conventional shrouded poppet valve. They observed that masking of inlet valve improves swirl rate and improves performance and reduce exhaust emissions. Paul and Ganesan [4] studied the flow in a diesel engine with different manifold i.e. helical, spiral and helical-spiral combination manifold under transient condition. They observed that helical-spiral offers less volumetric efficiency but gives the maximum swirl ratio. Mohiuddin [5] investigated the swirl effect due to designed swirl adapter and compared results with normal engine. He observed that effect of swirl comparing to normal turbulence shows an increase in the power as well as torque. Shrirao and Sambhe [6] analyzed the effect of swirl induction by internally threaded manifolds i.e. by acme, buttress and knuckle threads with constant pitch in diesel engine. In this experimental work they concluded that inlet manifold with buttress threads gives better performance as it had achieved higher swirl ratio than other configurations. Saad et.al [7] used a guided vane swirl and tumble device in inlet manifold and studied the in-cylinder air flow characteristics generated in order to improve air-fuel mixing in diesel engine using biodiesel. They found that using this device the in-cylinder parameters i.e. in-cylinder pressure, TKE, and velocity were improved. Meena and Priscilla [8] done a comparison study by using CFD modeling the effect of conventional and helical inlet port on the swirl flow. They found that pressure and temperature distribution for helical port is higher than conventional port, helical port creates higher swirl motion and turbulent kinetic energy and helical port provides higher volumetric efficiency. Kumar et.al [9] investigated the in-cylinder air flow after optimizing the number of guide vanes to improve performance and emissions of diesel engine. They found that the five vanes is the optimized number of vanes since it decreased 21.2% of brake specific fuel consumption and 30.7% of NOx respectively and brake thermal efficiency is increased 23.04% as compared with normal engine at full load. Chinche et.al [10] has also compared the results of different geometrical changes in inlet manifold in diesel engine with the help of experiment. They concluded that inclined intake manifold, guide vanes and the internal threading in the intake manifold effects the air-fuel mixing and hence thermal efficiency increases whereas brake specific fuel consumption and exhaust emissions are reduced.

In this paper we have also tried to simulate the two different geometries to get the swirl flow with the intention for further use it in the inlet manifold of the CI engine.

**Design of GVSTD and swirl pipe**

For the design of GVSTD and swirl pipe the past literature demonstrates that for GVSTD larger the surface area of the vanes, more will be the turbulence created. In the case of swirl pipe more the lobes gets defined more will be the swirl motion but it also decreases the volumetric efficiency due to the obstacle in the path of air. Compromising these factors, the GVSTD designed in this project is with four vanes arranged at 90° to each other with a 35° twist angle, and the swirl pipe is designed by joining thirteen different cross-sections with the help of multi-section solid option in CATIA V5R18 gradually converting the circular geometry into the four-lobed section.
Simulation setting

The simulation of both the geometries were done in ANSYS 16.0 the grid independence test is done on one of the geometry and it was found that on refining the mesh there is considerable change in the flow properties upto the element size of 0.8mm with elements 8,18,213 further decreasing the element size shows no significant change in the tangential velocity but only increases the simulation time.

The governing equations used to simulate the model for fluid flow were the continuity equation and momentum equation. The corresponding differential equation for the continuity equation can be written as:

$$\nabla \cdot \mathbf{U} = 0$$

Where \( \mathbf{U} \) is three dimensional flow velocity in the \( x, y \) and \( z \) directions. The momentum equation is based on the Newton’s second law where the forces considered are the body and the surface forces on the control volume and can be written as:

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

This is known as the Navier-Stoke or diffusion equation. In this equation \( p, g_x \) and \( \mu \) are the fluid pressure, body force and dynamic viscosity respectively.

The turbulence model Shear stress transport(SST) is used in this simulation. The SST is an eddy-viscosity model where the \( k-\epsilon \) model in the outer region of an outside of the boundary layer and the \( k-\omega \) model in the inner boundary layer are combined. The flow is consider to be the fully developed turbulent flow the model includes a 5% initial turbulence
intensity. The initial velocity at the inlet is set as 20 m/s and the pressure at the outlet is atmospheric.

**Results and discussion**

The factors that we have considered to judge the better geometry that is producing swirling motion are the velocity, pressure drop and the turbulence kinetic energy (TKE). Since the swirling motion depends on the kinetic energy i.e. velocity of the air, the turbulence kinetic energy will show the turbulence that will be created and the pressure graph will show the pressure drop that is an important aspect during fluid flow.

The fig. 3 shows the pressure drop across the different geometries and the comparison shows that the pressure drop across swirl pipe is more than the GVSTD. This is the case because the pressure in swirl pipe decreases and converts into velocity of air.

The comparison in the velocities of air is shown in fig. 4 shows that the velocity generation in the swirl pipe is more than the GVSTD, it is quite evitable because the decrease in the pressure increases the velocity of the air, therefore the more velocity will create more kinetic energy and the kinetic energy is needed for the swirl flow. The variation is might be due to the obstruction that is created by the guide vanes in GVSTD whereas in the swirl pipe the air enters into gradually converted lobed section and glides smoothly along the different lobes.

The fig. 5 defines the turbulence kinetic energy, it is the mean kinetic energy of the per unit mass associated with the eddies in the turbulent flow. Therefore more the TKE more will be the turbulent eddies formation and more the flow will be turbulent. The TKE comparison between the geometries shows that the loss in TKE is more for swirl pipe than for the GVSTD.

![Fig. 3 Variation of pressure of GVSTD and Swirl pipe along the length of the pipe](image1)

![Fig. 4 Variation of velocity of GVSTD and Swirl pipe along the length of the pipe](image2)

![Fig. 5 Variation of TKE of GVSTD and Swirl pipe along the length of the pipe](image3)
The fig. 6 shows the velocity vectors of GVSTD and swirl pipe at different angles the length of GVSTD and swirl pipe are different therefore three planes were taken at different positions to judge the velocity vectors i.e. plane 1 is at 20% of length, plane 2 is at 50% of length, plane 3 is at 90% of the length. The velocity vectors shows that the vectors in swirl pipe are more swirling in nature than the vectors of GVSTD.

<table>
<thead>
<tr>
<th>Plane 1</th>
<th>GVSTD</th>
<th>Swirl pipe</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td></td>
</tr>
<tr>
<td>Plane 2</td>
<td><img src="image3.png" alt="Image" /></td>
<td><img src="image4.png" alt="Image" /></td>
</tr>
<tr>
<td>Plane 3</td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
</tbody>
</table>

**Conclusion**

These geometries due to their high capabilities of producing turbulence have extensive use in diesel engines. As the diesel fuel has heavier molecules which make it difficult to atomize. Improper atomization leads to improper combustion and produces less power and more emissions also these geometries could be used in industries having transportation problem of coarse particle carrying fluid and also the alternate fuels engine such as biodiesel. GVSTD
and swirl pipe could be used in the intake manifold of CI engine to create more turbulence that could help in the atomization of diesel fuel to improve the combustion and reduce emissions and carbon deposits, and in between the pipe in which fluid having coarse particle is flowing to create swirl and tumble to restrict the blockage and minimize the power input. In this research, ANSYS-Fluent was used to run the simulation and it is found that where swirl pipe increases the velocity the GVSTD offers less pressure loss and turbulent kinetic energy loss. Therefore still there is the need of changing the profiles of vanes in the case of GVSTD and of lobes in the case of swirl pipe. Therefore, the outcomes of this research is that inserting GVSTD and swirl pipe will increase the turbulence but depends on the area of application in the case where the pressure drop is not an issue we can use the swirl pipe to generate turbulence but when pressure drop is an important aspect GVSTD could be used.

References