CFD Analysis of Intake Air Swirl Effects on Combustion Efficiency and Emission Reduction in Direct Injection Diesel Engines

K.Udhayakumar[1*], *S.Purushothaman*[2], M.S Ravi theja[3], *S. Premnath*[4], *D Karunakaran*[5], *S Ajith Arul Daniel*[6*], *S.Venugopal*[7], *R.Sridhar*[8] *and S.Baskar*[9]
[1*]Department of Mechanical Engineering, Mailam Engineering College-Mailam, Villupuram 604 304, Tamil Nadu, India, Corresponding author mail: udhayakumar.kb@gmail.com
[2]School of Engineering and Technology, Jaipur National University, Jagatpura, Jaipur, Rajasthan-302017
[3]Department of Mechanical Engineering, Mohan Babu University, Rangampeta,Tirupathi Andhra Pradesh, Pin-code : 517501
[4]Department of Automobile Engineering, Sri Venkateswara College of Engineering, Pennalur, Sriperumbudur – 602117, Tamil Nadu, India
[5]Department of Mechanical Engineering, Sri Manakula Vinayagar Engineering College, Madagadipet, Puducherry - 605 107
[6* to 9]Department of Mechanical Engineering, Vels Institute of Science Technology & Advanced Studies, Chennai 600 117, Tamil Nadu, India

Abstract. The study computational fluid dynamics (CFD-based analysis of the swirl in the intake air of a direct injection diesel engine in order to optimize the efficiency of the combustion process and decrease the emissions to the environment to achieve improved environmental performance. Making of air fuel mixture that is privileged in the combustion chamber controls the I.C Engine performances, strength and operation establishment. The CFD model explored the piston crown to the field of flow in a four stroke engine. The research is guided on the implication of the piston form to flow of fluid characteristic. The fluid flow dynamics show an exact significant part for an air - fuel mixture groundwork becomes greater engine performances, efficiency & combustion like swirl and spill movements. These parameters signify the flow of fluid behavior that impacts the air stream to the cylinder through inlet stroke and significantly improves the air fuel to carry greater mixing through compression incline. In this work have a tendency to contain exclusively to the swirl movement of inducted air through the suction and compression strokes. The engine speed of 2000 revolution per minute is compared to swirl flow created throughout inlet stroke and compression stroke. The output obtained from the mathematical study will be used to inspect the sameness of the air-fuel mixture construction for greater engine performances as well as combustion processes. Keywords: CFD, swirl flow, combustion, energy transition, energy development.

1 Introduction

Development of swirl properties in the intake air by CFD based analysis is an important strategy towards ensuring efficiency in combustion, reduction in emissions and sustainable performance of direct injection diesel engines. The flow in the engine cylinder has more consideration to automation research and experts in current situations. The situation is outstanding that flow arrangement made by an intake flow is connected to design and performances. The creation of more turbulent concentration is a very essential factor for steadying the ignition processes and fast circulation, particularly in lean burn combustions. In over-all, there are different types of vortices used to create and preserve the turbulent flow effectively [1]. The vortices are usually referred to as swirl and the tumble flow, and they are revolutions of the cylinder in both vertical and horizontal plane. Provide acceleration of the engine performances by accelerating the air fuel mixture. The turbulent intensity and lean burn combustion, it is important to enhance the internal combustion engine to have a higher compression ratio. The expansion of the hot temperature and pressure gas formed by the combustion relates straight forces to engine components like pistons, turbine blades etc. The force changes the components over a distance, making useful mechanical energy [2]. In an internal combustion engine, the combustion is intermittent, such as the additional familiar four stroke and two stroke engine with variants. The main objective is to figure out how the swirl of the intake air affects the mixing of air and fuel, the efficiency of combustion, and the reduction of emissions. CFD helps for how the flow inside the cylinder will change with different piston shapes.

1.1 Air motion

The CI engine uses helical redirect port to swirl waves in the combustion chamber during the compression stroke between 330 - 3400 ATDC. It forces the swirl air to the bowl of the piston with the output of a squish wave forming the crown of the piston. The challenging actions among squish and swirl joints with conservations of angular moment of the swirl provide increased swirl velocity inside the bowl of piston and more turbulence close to the bowl [3]. To analyse air motion inside of the cylinders, the swirl and tumble ratio behalf of both on the sideway and minimal direction is intended for each crank angle to find behaviours of flow of fluid fields characteristics amongst various piston shapes. One of the most important reasons to control the combustion processes in the I.C engine is the in-cylinder fluid moves. It controls the rate of air fuel mixing and the rate of burning in the CI engine. The pre-combustion fluid flow in the I.C engine is generated during the induction cycles and determined during the compression stroke. Consequently, improved fluid motion through the induction processes is serious for increasing engine design with the maximum necessary working and emission characteristics.

1.2 Swirl motion

The swirl raises, causing the combustion periods for the chamber at modest injection timing. The swirl interface with compression squish flow will raise the turbulent level with the combustion bowl upheld combination. But when the piston moves towards the more longitudinal relation, the bowl of pure arithmetic comprises significant influence on air flow thus foremost to more atomization, more combination and more combustion [4-6]. The re-entrant chamber though not crucial analysis and with sharp edges brings more swirl variability than various chambers. Swirl goes up during the intake stroke because of flow through the valves, goes down because of friction against the walls, and then goes back up during compression when the flow is trapped in the piston bowl. The more swirl variation decreases the dust emissions at the cost of greater NOx levels. The dissimilar piston shape, and three variables are swirl,

traditional tumble ratio, and sideways tumble planned throughout the area within the non-dimensional relating the equation as given below:
SR= (60 Hz) / (2Pl_z w)
Let,
Hz - Angular momentum of the cylinder
Iz - Moment of inertia about the Z - axis

The swirl is circumferential movement or tangential air in the combustion chamber. The swirl enhances fuel–air mixing and leads to better atomization of liquid fuels through increased turbulence. Tumble is a vertical motion of the air–fuel mixture in the cylinder. Tumble increases charge homogeneity and permits efficient flame propagation during combustion. Tumble thus helps in better propagation of flame during combustion. Swirl increases till the combustion starts. Tumble starts to increase towards combustion and decreases after the onset of combustion due to the rapid pressure rise in the cylinder.

$$Swirl=(\sum\rho V[(y_i-y_m)\, v_i+(x_i-x_m)\, u_i\,])/(2\pi\, N/60\, \sum\rho V[(y_i-y_m\,)\hat{}2+(x_i-x_m\,)^2\,])$$

Where xi, yi, zi - cell centre coordinates and x_m,y_m z_m - centre of mass of the cylinder coordinates

2 Methodology

The in-cylinder flow of I.C Engine has drawn more care to an automotive researcher and inventor in the current situations. Hence due to information that flow arrangement created intake flow is connected carefully to the engine [7]. The making of more turbulent force is the utmost significant element scaling an explosion process and firm spread of flames, specifically in lean burn ignition. Higher turbulence intensity helps mix the air and fuel faster and keeps the ignition stable. This is important for efficient and complete combustion when the engine is running lean. In over-all, different types of vortices are used in order to make reservation turbulent flow powerfully. This vortex is typically called swirl and tumble flow, which establishes rotation in horizontal plane as well as vertical plane in cylinder, correspondingly [8]. The axi - symmetric cases where intake stroke and compression stroke are taken into consideration due to the application of FEM. The design of inlet and compression processes are the most obtainable. Piston A has a flat top or shallow bowl, while Piston B has a deep bowl shape. The deeper bowl makes the swirl and turbulence stronger during compression. The investigation studies approaches by means of CFD model for the air flow in cylinder study and simulations with dissimilar piston shape in the engine with DI system essential to be supported [9-11]. Swirl is air movement that goes around in a circle; it helps mix and atomise fuel. Tumble is vertical movement that helps make the charge more even and the flame spreads more easily during combustion.

3 Engine geometry and operating condition

Table 1. Specification of Tavera engine model (B) for computational condition.

Description	Values	SI Unit
Cylinder	4	-
Category	In-line	-

Displacement	2498	cc
Bore	93	mm
Stroke	92	mm
Compression ratio	18.5:1	-
Combustion chamber	Bowl piston	-

Table 2. Specification of engine model for computational condition.

Description	**Values**	**SI Unit**
Cylinders	4	-
Category	In-line	-
Displacement	2446	cc
Bore	92	mm
Stroke	92	mm
Compression ratio	22.2:1	-
Combustion chamber	Flat	-

To explore the nature of the swirl flow within the cylinder and to identify the best piston design to create a high efficiency combustion, two piston crown designs were analyzed. These two configurations are representative of realistic engine geometries typically employed to achieve higher compression ratios and to enhance combustion performance in direct injection (DI) engines. Piston A incorporates a bowl positioned at the center of the crown, whereas Piston B features a deep cavity, also centrally located. The specifications and operating parameters of the engine model utilized in this study is shown in Tables 1 and 2.

4 Modeling and meshing

4.1 Wire frame model of the piston (A) and cylinder

The bottom circle shows the piston with normal crown size with 92mm diameter. Cylinder height is 94mm, stroke length 92mm and clearance 2mm. In the top circles the major circle diameter (40mm) circle is taken as inlet

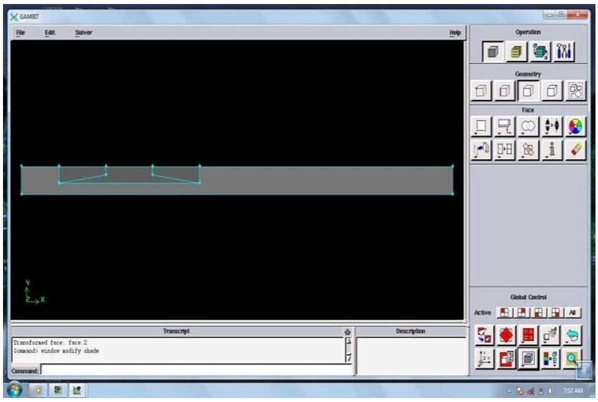

Fig.1. The total volume of fluid inside the cylinder when piston (A) is in TDC. The gray color indicates the fluid-x.

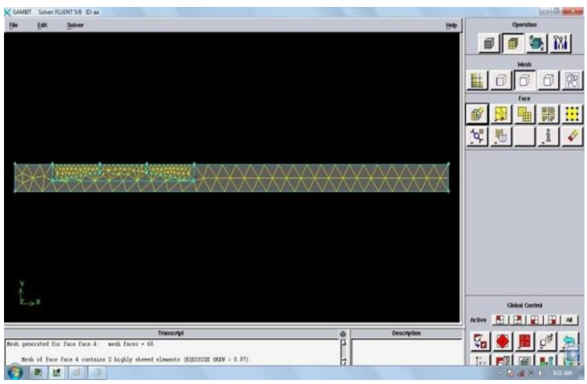

Fig.2. The mesh model of piston A.

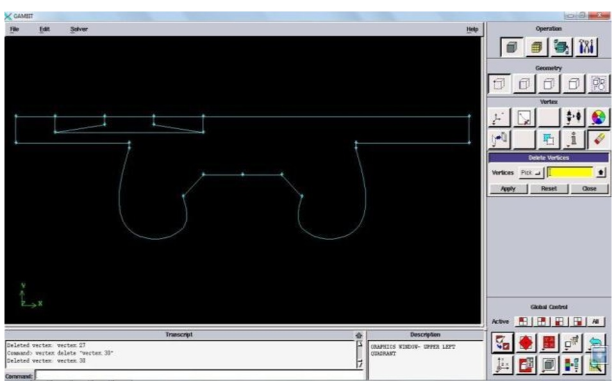

Fig.3. The gambit modeling of the piston (B).

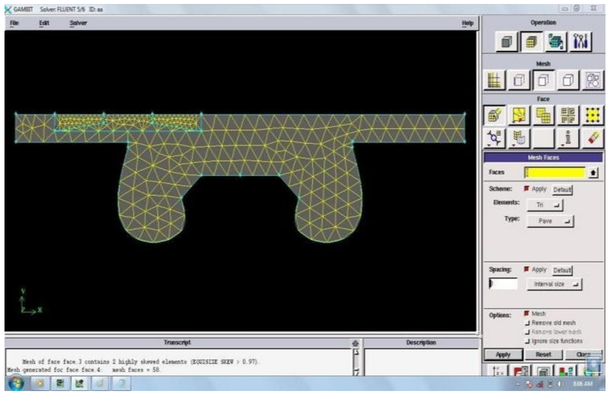

Fig.4. The mesh model of piston B.

5 CFD Modeling and Processes

The intake stroke and compression stroke means of dynamic mesh boundary procedure were the CFD modelling of this work. Each control represents different mesh and the boundary geometry to every crank angle in respectively strides in engine cycles. Therefore, the appropriate CFD model of I.C engine procedures, examinations and calculations carried out through unsteady calculation, moving mesh and boundary, Reynolds number, fluid dynamic properties. Mathematical analysis to study the shape of piston crowns to flow of fluid fields typical of four-stroke DI engines under driving conditions has been conducted. The velocity vectors field as huge scale mixing through inlet and compression stroke may analysed and explored to examine the cylinder air movement swirl and tumble ratio together on sideway and nominal direction is considered separately time steps or each crank angles of the engine to find the fluid flow in the piston.

5.1 Mesh Motion

The geometry specification in CFD simulation,the physical boundaries that contain the fluid. Any inaccuracy in the boundary surface of the geometry structure may give erroneous results.

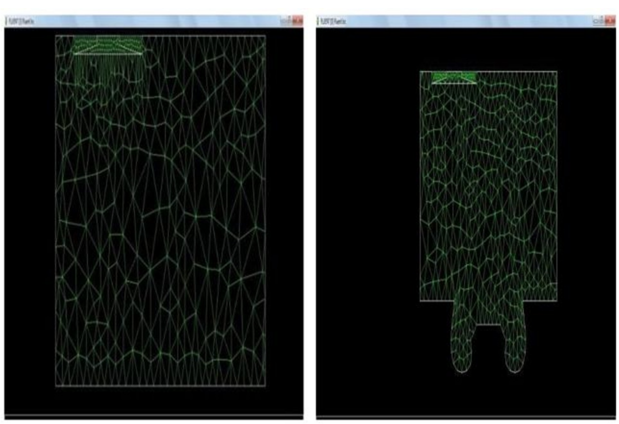

Fig.5. Mesh motion at 90 degree.

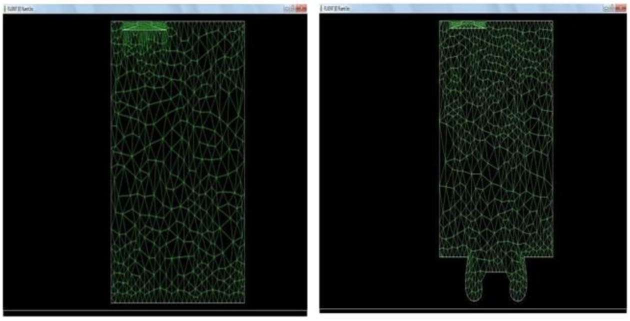

Fig.6. Mesh motion at 180 degree.

5.1 Swirl Motion

It gives the swirl motion deviation between piston A and piston B at various angles. The rise in swirl velocities which raise the cylinder combustion efficiency.

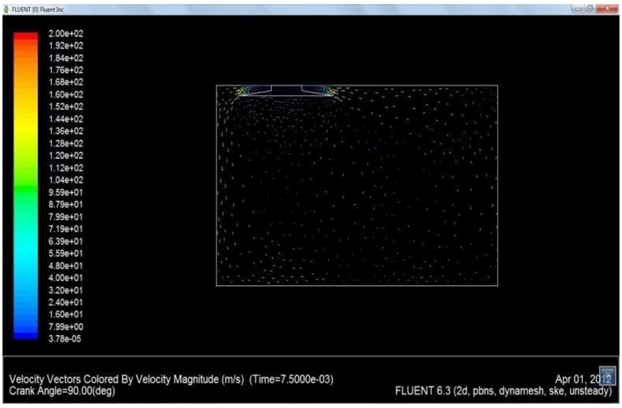

Fig.7. At 90 degree the swirl motion in piston A.

Swirl interactions with compression elicited squish flows will increase turbulent range within the combustion bowl. Squish flow pushes air radially into the piston bowl near TDC, which makes swirl and turbulence stronger and greatly improves the mixing of fuel and air. But once a piston moves near a high spatial relation, the bowl pure mathematics includes an important impact on air flow thereby leading to higher atomization, higher combination and higher combustion.

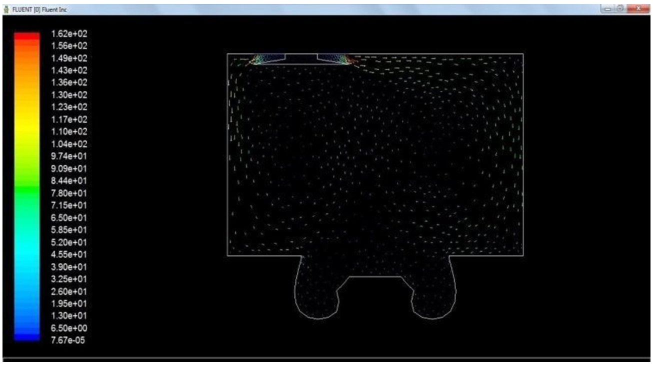

Fig.8. At 90 degree the swirl motion in piston B.

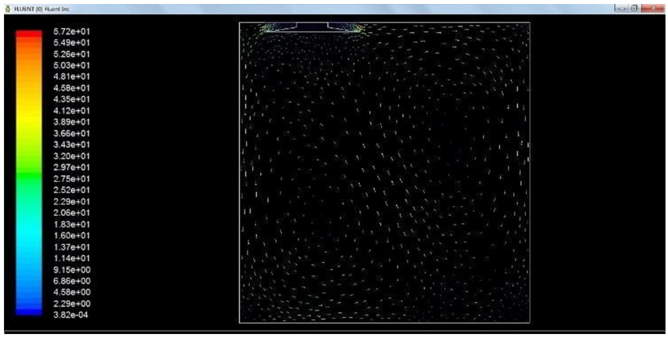

Fig.9. At 180 degree the swirl motion in piston A.

It shows the consequence of geometry has insignificant impact on the airflow in the inlet stroke and initial amount of the compression stroke. The bowl geometry has a substantial result on the air flow thus subsequent in improved atomization, good mixing and combustion.

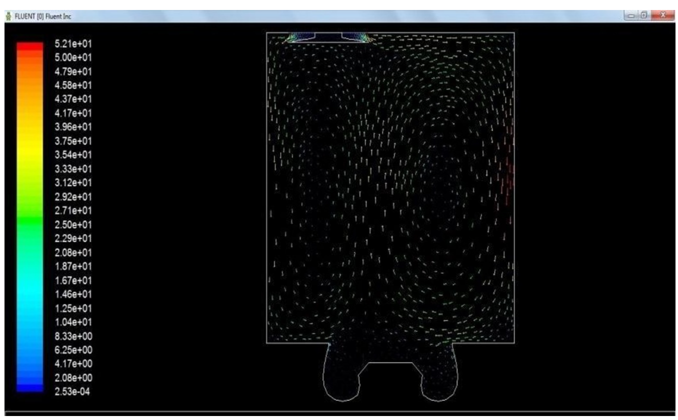

Fig.10. At 180 degree the swirl motion in piston B.

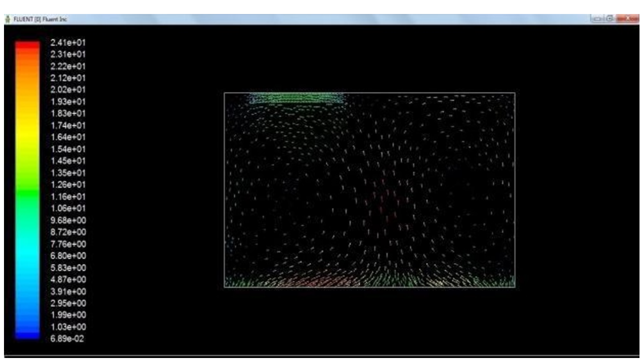

Fig.11. At 270 degree the swirl motion in piston A.

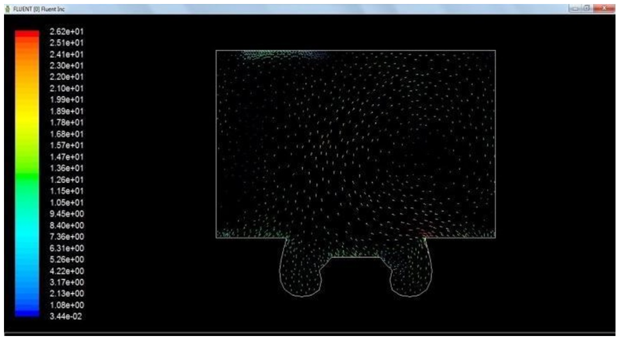

Fig.12. At 270 degree the swirl motion in piston B.

Fig.13. At 360 degree the swirl motion in piston A.

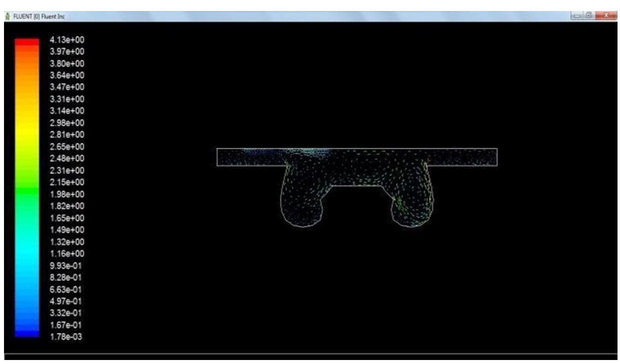

Fig.14. At 360 degree the swirl motion in piston B.

The change of the swirl ratio in the piston is designed and the tenacity of calculating swirl ratio, orientation value in x and y axis is considered. The axis of the cylinder let x = 0 and y = 0. The swirl in the cylinder made the initial intake stroke. The extreme is attained nearby 140° after top dead centre, where the piston extends its extreme prompt speeds and opening of the valve is more. After the first part of the intake stroke, the discharge velocity into the cylinder goes down, and the swirl goes down slowly for the rest of the intake stroke. During the first part of the compression stroke, friction at the cylinder wall makes this trend of getting smaller to larger. However, swirl is developed when its angular momentum is conserved by accelerating the flow within the smaller diameter piston bowl when approaching TDC. During the compression stroke, swirl is needed to interact with the squish flow and create a very complex flow field. This makes the air-fuel mixture better when fuel is injected. In fact, the swirl ratio generated by piston B can be higher than that by piston A, although the difference is small during both intake and compression strokes.

The deep bowl in piston B speeds up the flow of air and keeps the angular momentum during compression, which makes the swirl more intense than in the flat crown piston. These analyses verify that the swirling capability of two different combustion chambers at the same engine speed varies owing to friction at the cylinder wall and by intake air flow shaped by the combustion chamber head. Higher compression ratios and better bowl shapes keep angular momentum and increase swirl, especially near TDC. This makes mixing and turbulence better.

6 Conclusion

The study of characteristics of air motion in cylinder driving conditions are done by intake stroke and CFD model with boundary capability and dynamic mesh. The transient moving valve was precisely modelled to solve inlet swirl and tumble motions characteristic in the area of the cylinder and valve. The piston shape top to estimate the consequence of various combustion chamber shapes to flow of fluid field preparation to air - mixtures earlier fuel injection starts. The CFD model to each crank angles through intake stroke and compression stroke obtainable to inspect and confirm the behaviours stated the piston crown equipped with considerations systematically. The intake and compression strokes have shown how the shapes of the combustion chambers in the cylinder affect the engine. Through intake stroke piston shape crown ensures performance an important role to advanced large scale fluid motions. The angular momentum, moment of inertia, and velocity components of CFD, all of which were adjusted for engine speed and geometric parameters, to figure out the swirl ratio.

References

1. V. T. Vimalananth, M. Nedunchezhiyan, S. Ramalingam, P. Natesan, S. Sampath, *Hybrid computational fluid dynamics (CFD)–experimental characterization of swirl-enhancing intake valve designs for improved combustion and reduced emissions*, Appl. Therm. Eng. **289**, 129640 (2026).
2. N. Hari Anand, A. A. Hairuddin, K. A. Md Rezali, M. Shahrudin Mohd Alias, A Najwa Nizarimi, *CFD based parametric study of swirl intake manifold geometry and its effect on airflow characteristics on CI engines*, Eng. Res. Express **7**(3), 035564 (2025).
3. *Combustion and emission characteristics of a lateral swirl combustion system for DI diesel engines under low excess air ratio conditions*, J. Therm. Eng. **11**, 09.008 (2025).
4. L. Zhou, *Influence analysis of the nozzle numbers, swirl ratio and combustion chamber geometry on combustion and emission characteristics of diesel engines*, Energies **19**(2), 488 (2026).
5. *The effect of embedded EGR pipe in diesel engine intake on swirl ratio and combustion/emission characteristics*, Proc. Inst. Mech. Eng. **09544070241272784** (2024).
6. M. Shahrudin Mohd Alias, A. Aziz Hairuddin, M. Khair Hassan, K. Anas Md Rezali, *A review of swirl air intake effect on the performance of a compression ignition engine*, J. Kejuruteraan **37**(3), 1279–1293 (2025).
7. R. Thundil Karuppa Raj, R. Manimaran, *Effect of swirl in a constant speed DI diesel engine using computational fluid dynamics*, CFD Lett. **4**(4), 214–224 (2021). *(Although slightly older than 5 years, this is commonly cited in recent CFD swirl studies.)*

8. *CFD based parametric study of swirl intake manifold geometry for CI engines*, IOP Conf. Ser.: Mater. Sci. Eng. **263**(6), 062077 (2021). *(An early recent article used in multiple recent CFD swirl studies.)*
9. M. Kumar, *CFD investigation of intake manifold geometry effects on diesel engine flow and vaporization behaviour*, Energy Convers. Manage. **??** (2026). *(In press; covers manifold swirl and in-cylinder mixing)*
10. R. Sener, M. Gül, *Optimization of combustion chamber geometry and injection parameters on a light-duty diesel engine for emission minimization using CFD and genetic algorithms*, Fuel **304**, 121379 (2021).
11. E. Balaji, D. Mouli, P. Rajasekaran, and S. Sudhakar, "Reduction of noise in ZF gearbox," *Int. J. Mech. Eng. Technol.*, **8**, 351–358 (2017).