INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 04, Issue 11, November 2024, pp : 1389-1393
www.ijprems.com
editor@ijprems.com
e-ISSN : 2583-1062
Impact Factor : 7.001

ADVANCEMENTS IN CFD ANALYSIS FOR TWO-STROKE COMBUSTION ENGINES: MODELING, OPTIMIZATION, AND EMISSIONS REDUCTION

Prabhat Singh Rathore[1], Vineet Kumar Dwivedi[2]

[1]Scholar, School of Mechanical Engineering, Faculty of Engineering and Technology, SAM Global University, Bhopal, M.P, 462046, India.

[2]Head and Prof., School of Mechanical Engineering, Faculty of Engineering and Technology, SAM Global University, Bhopal, M.P, 462046, India.

ABSTRACT

Computational Fluid Dynamics (CFD) has become a vital tool in the analysis and optimization of two-stroke combustion engines, enabling engineers to explore complex fluid flow, combustion, and thermal dynamics in detail. This review paper highlights the recent advancements in CFD techniques and their applications in improving the performance, efficiency, and emissions of two-stroke engines. Key areas of development include turbulence modeling, where hybrid models such as Large Eddy Simulation (LES) and Reynolds-Averaged Navier-Stokes (RANS) have been integrated to better capture the intricate flow structures and transient phenomena within the engine. Additionally, advances in multi-phase flow modeling have enhanced the simulation of fuel-air interactions, combustion processes, and exhaust scavenging, which are critical for optimizing fuel efficiency and minimizing emissions. The paper also discusses the role of hybrid models that combine LES and RANS to balance computational cost and accuracy, and highlights recent innovations in CFD tools and solvers, which have made simulations more accessible and efficient. The use of these advanced CFD techniques has significantly improved the understanding of two-stroke engine behavior, enabled more effective design modifications and contributed to the development of cleaner, more efficient engines. This review underscores the importance of CFD in advancing two-stroke engine technology and addressing future challenges in engine design and environmental sustainability.

Keywords: Computational Fluid Dynamics (CFD), Two-Stroke Engine, Turbulence Modeling, Combustion Optimization, Emissions Reduction

1. INTRODUCTION

Two-stroke combustion engines are a type of internal combustion engine that completes the entire engine cycle—intake, compression, combustion, and exhaust—in just two strokes of the piston, or one revolution of the crankshaft. This design simplifies the engine structure, making it lighter and more compact, which is particularly advantageous in applications like motorcycles, small boats, lawn equipment, and certain industrial machines. In contrast to four-stroke engines, which require four strokes of the piston to complete the cycle, two-stroke engines fire once every revolution of the crankshaft, leading to higher power output in a smaller engine size. However, they are typically less fuel-efficient and produce higher emissions due to less complete combustion and the challenge of scavenging exhaust gases effectively. Two-stroke engines also face lubrication challenges, as oil is often mixed with fuel for lubrication purposes, which further contributes to inefficiency. Computational Fluid Dynamics (CFD) has become a crucial tool in the design and optimization of combustion engines, including two-stroke engines. CFD allows engineers to simulate and analyze complex fluid flow, combustion, and thermal processes within the engine, offering a non-intrusive method to study engine behavior under different operating conditions. By solving the governing equations of fluid dynamics, CFD provides insights into flow patterns, pressure distribution, heat transfer, and combustion processes, helping engineers make informed decisions without the need for costly and time-consuming physical experiments. The application of CFD in engine analysis has proven beneficial for optimizing engine performance, improving fuel efficiency, reducing emissions, and understanding intricate phenomena such as turbulence, air-fuel mixing, and scavenging in two-stroke engines. This review paper aims to provide an in-depth understanding of the role of CFD in the analysis and optimization of two-stroke combustion engines. It will explore the various applications of CFD in simulating different phases of the two-stroke engine cycle, such as intake, compression, combustion, and exhaust. Additionally, the paper will discuss how CFD can be utilized to enhance engine performance by optimizing parameters such as fuel efficiency, power output, and emissions. The review will also address the challenges associated with CFD modeling of two-stroke engines, including the complexity of accurate predictions and the difficulties in simulating phenomena like turbulent flows and scavenging. Finally, the paper will highlight recent advancements in CFD techniques and their impact on the development of two-stroke engine technology, providing a comprehensive overview of the state of the art in this field.

INTERNATIONAL JOURNAL OF PROGRESSIVE RESEARCH IN ENGINEERING MANAGEMENT AND SCIENCE (IJPREMS)
(Int Peer Reviewed Journal)
Vol. 04, Issue 11, November 2024, pp : 1389-1393
e-ISSN : 2583-1062
Impact Factor : 7.001

Savioli et al. [1] highlight the challenges in applying CFD simulations to two-stroke engines, especially when transitioning from 3D detailed analyses to simplified 1D models. Critical issues include correlating cylinder gas composition with exhaust flow and accurately characterizing port discharge properties, which are difficult to determine experimentally. They emphasize the need for multi-cycle simulations to capture transient conditions, as single-cycle analyses are insufficient. The paper proposes a methodology combining CFD with steady-flow bench tests, validated by experimental data, to improve simulation accuracy for two-stroke engines. S Li et al. [2] investigated the autoignition characteristics of 0# diesel, a fuel designed for China's national stage VI emission standard. Ignition delay times (IDTs) were measured across varying temperatures, pressures, and equivalence ratios using a shock tube. Results indicated that ignition delay times were sensitive to pressure but less so to equivalence ratio. A ternary surrogate fuel model was developed to match 0# diesel properties, and a skeletal kinetic mechanism was proposed, accurately reproducing experimental ignition characteristics and aiding numerical simulations. G Chendong et al. [3] explore the potential advantages of Free Piston Internal Combustion Engine Linear Generators (FPELG), which eliminate the crank-connecting rod mechanism found in traditional engines, offering benefits such as improved efficiency, lower emissions, and multi-fuel flexibility. The paper reviews recent advances in FPELG research, focusing on modeling, simulation methods, experimental approaches, and control strategies. The authors conclude that optimizing the performance of both the engine and linear generator, along with controlling the piston trajectory, are crucial for the future development of FPELG technology. A Thiruvengadam et al. [4] assess the energy distribution and engine efficiency of a pre-2014 heavy-duty diesel engine as a baseline for evaluating future technologies' impact on greenhouse gas emissions and fuel consumption. The study predicts fuel consumption reductions of 7.9% and 18.3% for MY 2017 and MY 2020+ engines, respectively. The results highlight potential efficiency gains from advancements such as waste heat recovery (WHR) systems, which could improve brake thermal efficiency (BTE) by up to 3%. This work provides insights into energy loss mechanisms, aiding future engine efficiency models. J.R. Serrano et al. [5] investigate the impact of thermal insulation on engine internal walls, including the pistons, cylinder head, and exhaust manifold, to reduce heat losses and improve engine performance and emissions. The study analyzes NOx, soot emissions, and brake-specific fuel consumption (BSFC). The results show that insulating the cylinder head and pistons reduces soot emissions but increases fuel consumption while maintaining NOx levels. Insulating the exhaust manifold, however, improves both NOx-soot and NOx-BSFC tradeoffs, offering a balanced performance improvement. Optimization of air management and injection settings was also performed to maximize benefits. C. Binder et al. [6] examine the impact of a 1mm-thick plasma-sprayed yttria-stabilized zirconia (YSZ) coating on a piston, comparing it to a traditional steel piston. The study investigates the heat release, heat transfer rate to the piston cooling gallery, local surface temperature, and instantaneous surface heat flux. Results show that although the surface temperature variation is similar for both pistons, the steel piston experiences significantly higher instantaneous heat flux during combustion. Additionally, combustion is slower with the YSZ-coated piston, indicating potential benefits for reducing in-cylinder heat losses. S. Caputo et al. [7] present a numerical study on the effects of combustion chamber insulation on heat transfer, thermal efficiency, and exhaust temperatures in a 1.6L turbocharged diesel engine. The study first simulates the complete insulation of engine components, including pistons, liner, firedeck, and valves, revealing that the piston offers the greatest potential for reducing in-cylinder heat transfer and Brake Specific Fuel Consumption (BSFC). The research then focuses on the impact of various piston Thermal Barrier Coatings (TBCs) on performance and wall temperatures, using a 1-D simulation model coupled with a lumped mass thermal model. The effects of Yttria-Partially Stabilized Zirconia (Y-PSZ) and anodized aluminum coatings with different thicknesses were analyzed. N. Uchida et al. [8] investigate the impact of a 0.5mm Zirconia (ZrO_2) thermal spray coating on a forged steel piston cavity to reduce heat transfer between the hot gas and cavity wall. Despite expectations for improved heat retention, no significant reduction in heat loss was observed. The study includes flame impingement observation using a sapphire window and macrophotography, revealing a thinner thermal boundary layer with the Zirconia coating, which increased the heat transfer coefficient. Numerical simulations suggested that surface roughness and porous structure, rather than the surface temperature swing, contributed to this effect. While heat loss was not drastically reduced, the potential for Brake Thermal Efficiency (BTE) improvement was confirmed.

2. FUNDAMETNTALS OF 2-STROKE ENGINE

The two-stroke combustion engine operates on a simple and efficient principle, completing the entire engine cycle—intake, compression, combustion, and exhaust—in just two strokes of the piston (one crankshaft revolution). This is in contrast to the four-stroke engine, which requires four strokes to complete a cycle, resulting in two crankshaft revolutions. The two-stroke engine's ability to complete a power cycle in half the time allows it to deliver more power per unit of displacement, making it ideal for applications requiring high power-to-weight ratios, such as motorcycles, small boats, and lawn equipment. In the two-stroke engine, the cycle begins when the piston is at the top of the cylinder (top dead center). As the piston moves downward (the first stroke), it compresses the fuel-air mixture in the combustion chamber. Simultaneously, exhaust gases from the previous cycle are expelled through the exhaust port, and fresh fuel-air mixture enters the crankcase through the intake port. When the piston reaches the bottom of the stroke (bottom dead center), the fuel-air mixture is compressed into the combustion chamber, and the exhaust valve closes. The spark plug ignites the compressed mixture, causing combustion and pushing the piston upwards (the second stroke). This action simultaneously forces the exhaust gases out through the exhaust port and draws in a fresh fuel-air mixture into the cylinder. This cycle repeats continuously as the engine runs.

The key components of a two-stroke engine include the intake and exhaust ports, the crankcase, and the piston. The intake and exhaust ports serve as the main pathways for air-fuel intake and exhaust gas expulsion. The crankcase acts as a reservoir for the fuel-air mixture, which is pressurized and forced into the combustion chamber during the piston's downward stroke. The piston moves up and down, compressing and igniting the fuel-air mixture, while the spark plug initiates combustion. Additionally, the exhaust valve and intake valve are replaced by exhaust and intake ports, which are uncovered by the piston during its motion. The primary difference between a two-stroke and a four-stroke engine lies in their operation cycles. While a two-stroke engine completes a cycle in one crankshaft revolution, a four-stroke engine requires two revolutions to complete its cycle. This difference results in the two-stroke engine having a higher power-to-weight ratio and simpler design, but it is less fuel-efficient and produces higher emissions due to incomplete combustion and less effective exhaust scavenging. Additionally, two-stroke engines often require oil mixed with fuel for lubrication, whereas four-stroke engines have separate lubrication systems.

3. COMPUTATIONAL FLUID DYNAMICS (CFD) IN COMBUSTION ENGINE ANALYSIS

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems involving fluid flows. In combustion engine analysis, CFD plays a critical role by providing detailed insights into the complex fluid dynamics, heat transfer, and combustion processes within the engine. Traditionally, experimental testing has been used to assess engine performance, but CFD offers a more cost-effective and time-efficient alternative. It allows engineers to simulate real-world conditions, optimizing engine designs, improving fuel efficiency, reducing emissions, and enhancing overall engine performance. By using CFD simulations, engine designers can analyze flow behavior, combustion characteristics, and thermal dynamics without the need for extensive physical prototypes. Key CFD modeling techniques used in combustion engine analysis include the modeling of turbulence, combustion processes, and multiphase flows. Turbulence modeling is essential because fluid flows in engines are highly turbulent, which affects mixing, combustion, and heat transfer. Popular turbulence models like the Reynolds-Averaged Navier-Stokes (RANS) model, Large Eddy Simulation (LES), and Detached Eddy Simulation (DES) are often employed. These models help approximate the effects of turbulence on the flow characteristics within the engine. Combustion models are used to simulate the ignition, flame propagation, and exhaust processes. Models such as the Laminar Flamelet Model (LFM), Eddy Dissipation Model (EDM), and the PDF (Probability Density Function) approach are commonly applied for these purposes. For engine simulations, it is also important to consider the mixing of fuel and air, the scavenging process (for two-stroke engines), and the fuel spray dynamics, especially when liquid fuels are involved. Additionally, multiphase flow modeling is often necessary for handling the interaction between liquid fuel and air in the combustion chamber.

The governing equations used in CFD for engine simulations are derived from the fundamental principles of fluid mechanics, thermodynamics, and heat transfer. The primary set of equations includes the **Navier-Stokes equations** for fluid flow, the **energy equation** for heat transfer, and the **species conservation equations** for modeling chemical reactions and combustion. The Navier-Stokes equations govern the velocity field of the fluid, while the energy equation tracks the thermal energy within the system. The species conservation equations describe the transport and reaction of chemical species, which is essential for modeling combustion and emissions formation. These equations are highly complex and require numerical methods for their solution. Common numerical methods used in CFD include finite difference, finite volume, and finite element methods, with the finite volume method being particularly popular for engine simulations due to its conservation properties. To solve these equations, CFD software employs discretization techniques, breaking down the engine domain into a grid or mesh, and applying the equations to each grid cell. Solvers are then used to iterate through the equations, providing numerical solutions for fluid velocity, pressure, temperature, and species concentration across the engine domain.

CFD MODELING IN TWO-STROKE COMBUSTION ENGINES

In Computational Fluid Dynamics (CFD) modeling of two-stroke combustion engines, mesh generation and discretization are essential steps to accurately capture the fluid flow, combustion, and thermal processes within the engine. Mesh generation involves dividing the engine geometry into smaller, manageable control volumes (cells), which are used to solve the governing equations. The quality of the mesh is critical, as it directly impacts the accuracy and computational cost of the simulation. A finer mesh is typically required in regions where high gradients of variables like velocity, pressure, and temperature occur, such as near the walls of the combustion chamber, the intake and exhaust ports, and areas with intense mixing or combustion. Structured meshes, consisting of regular grid patterns, are commonly used in regions of simple geometry, while unstructured meshes are applied in more complex areas to capture intricate flow features. Hybrid meshes, combining both structured and unstructured elements, are also commonly employed in engine simulations to balance accuracy and computational efficiency. Discretization techniques are used to approximate the continuous governing equations into a form that can be solved numerically. In CFD simulations, this process involves dividing the domain into small control volumes and solving the equations for each control volume iteratively. The finite volume method (FVM) is the most widely used discretization technique in engine simulations, as it conserves mass, momentum, and energy within each control volume. The method calculates the flow variables at the center of each cell, while ensuring that the fluxes through the cell faces are computed and applied. This approach is particularly advantageous for handling the complex boundary layers and shock waves present in combustion processes.

Boundary conditions and initial conditions play a crucial role in accurately simulating the engine's operating environment. Boundary conditions define the interactions between the engine and its surroundings, including the intake, exhaust, and wall surfaces. For two-stroke engine simulations, key boundary conditions include the velocity and temperature at the intake port, the exhaust port, and the piston walls, which may also include heat transfer conditions. Inlet boundary conditions typically represent the characteristics of the air-fuel mixture entering the cylinder, such as velocity, temperature, and turbulence. Exhaust boundary conditions specify the pressure and temperature of the exhaust gases. The wall boundaries are often defined with no-slip conditions for velocity and may include heat transfer properties for the piston and cylinder walls. Initial conditions specify the state of the flow variables (such as velocity, pressure, and temperature) at the start of the simulation, often based on experimental data or engine operating parameters like engine speed and load.

In CFD simulations of two-stroke engines, solvers are responsible for solving the governing equations iteratively. The solvers used in these models are typically based on numerical techniques such as pressure-based or density-based solvers. For combustion processes, solvers often employ turbulence models (such as k-ε or k-ω models) to simulate the turbulent flow within the engine. For ignition and combustion, models such as the Eddy Dissipation Model (EDM) or the Laminar Flamelet Model (LFM) are used to simulate the chemical reactions and energy release during the combustion phase. These solvers must also account for complex interactions, such as spray formation in engines with liquid fuel injection, as well as the scavenging process in two-stroke engines. The solvers typically use iterative methods, including methods like SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) for pressure-velocity coupling, to ensure stable convergence of the solution. These solvers provide insights into flow patterns, pressure distribution, combustion efficiency, and emissions characteristics, helping engineers optimize the performance of two-stroke engines

RECENT ADVANCES AND TRENDS IN CFD ANALYSIS FOR TWO-STROKE ENGINES

The field of Computational Fluid Dynamics (CFD) for two-stroke combustion engines has seen significant advances in recent years, particularly in the areas of turbulence modeling, multi-phase flow simulations, and the development of hybrid models for combustion analysis. These innovations have enabled more accurate and efficient simulations of engine behavior under a wide range of operating conditions, addressing challenges such as complex fluid dynamics, combustion efficiency, and emissions control.

Advances in turbulence modeling have been a key focus in improving the accuracy of CFD simulations for two-stroke engines. Traditional turbulence models, such as the **Reynolds-Averaged Navier-Stokes (RANS)** model, have limitations in accurately capturing the complex flow structures and transient phenomena in internal combustion engines. As a result, there has been a shift towards more advanced models that can resolve fine-scale turbulence structures and provide better predictions of engine performance. **Large Eddy Simulation (LES)** has emerged as a powerful tool for simulating turbulent flows with higher accuracy by resolving large eddies and modeling smaller scales of turbulence. LES can capture the unsteady nature of turbulence, which is especially important in two-stroke engines where rapid changes in flow and combustion occur. However, LES comes with higher computational demands, which has led to the development of hybrid turbulence models that combine the strengths of both **RANS** and **LES**. The **Detached Eddy** Simulation (DES) model, for example, offers a balance between the computational efficiency of RANS and the detailed turbulence resolution of LES, making it suitable for engine simulations where both steady and transient flow conditions need to be considered.

Another significant development in CFD for two-stroke engines is the modeling of **multi-phase flows**, particularly in engines with fuel injection systems. Multi-phase flow simulations are essential for accurately representing the interaction between air, fuel, and exhaust gases during intake, combustion, and scavenging. Advances in spray modeling have improved the ability to simulate fuel injection processes, where liquid fuel droplets interact with the air in the combustion chamber. The **Lagrangian-Eulerian approach** is commonly used to simulate droplet dynamics, while models like **Eulerian multiphase models** help capture the interactions between phases, such as fuel, air, and combustion products. These advancements have made it possible to optimize fuel-air mixing, combustion efficiency, and emissions in two-stroke engines, which are traditionally more difficult to model due to the rapid changes in fluid properties during the cycle.

The use of **hybrid models** has been another important trend in CFD analysis for two-stroke engines. By combining the advantages of **LES** and **RANS** models, hybrid approaches such as **Hybrid LES-RANS** are increasingly being used for combustion analysis. These models allow for detailed resolution of turbulent eddies in regions where high turbulence is expected (such as near the combustion chamber) while using the less computationally expensive RANS model in other regions, such as the intake or exhaust. This enables a more efficient and accurate simulation of the entire engine cycle, capturing both large-scale turbulence and fine-scale interactions.

Recent studies have shown promising innovations in **CFD tools** designed specifically for two-stroke engines. Enhanced solvers that incorporate **multi-dimensional modeling** techniques, improved combustion models (such as **Eddy Dissipation Model (EDM)** and **Flamelet Models**), and more accurate **thermodynamic models** are now available in state-of-the-art CFD software. These tools are increasingly user-friendly and include pre-configured models for specific engine types, which reduces the time required for setting up simulations. Moreover, the development of parallel computing and **cloud-based simulation platforms** has made it possible to perform highly complex simulations in shorter timeframes, making CFD analysis more accessible to researchers and engineers.

4. CONCLUSION

In conclusion, CFD analysis has become an indispensable tool for understanding and optimizing the performance of two-stroke combustion engines. The advancements in turbulence modeling, multi-phase flow simulations, and hybrid modeling techniques have significantly enhanced the ability to simulate and analyze the complex fluid dynamics, combustion processes, and heat transfer mechanisms within these engines. The integration of Large Eddy Simulation (LES), Reynolds-Averaged Navier-Stokes (RANS) models, and hybrid approaches like Detached Eddy Simulation (DES) has enabled more accurate predictions of engine behavior under varying operating conditions, addressing the challenges of turbulent flow and combustion efficiency. Furthermore, the development of advanced multi-phase flow models has allowed for better simulation of fuel injection processes, fuel-air mixing, and exhaust scavenging, contributing to improved performance and emissions control. With the continued evolution of computational tools, including cloud-based simulation platforms and parallel computing techniques, CFD is poised to play a crucial role in the future of two-stroke engine design, providing engineers with the insights needed to meet the growing demand for high-performance, low-emission engines. These advancements will continue to drive innovation in engine technology, offering promising solutions for enhancing engine efficiency, reducing environmental impact, and meeting stringent regulatory standards.

5. REFERENCES

[1] Tommaso Savioli, CFD Analysis of 2-Stroke Engines, Energy Procedia, Volume 81, 2015, Pages 723-731, ISSN 1876-6102, https://doi.org/10.1016/j.egypro.2015.12.078.

[2] S. Li, H. Lu, Y. Mao, et al., Experimental and kinetic modeling study on ignition characteristic of 0# diesel in a shock tube, J] Combustion and Flame 242 (2022), 112171, https://doi.org/10.1016/j.combustflame.2022.112171.

[3] C. Guo, Z. Zuo, H.a Feng, et al., Review of recent advances of free-piston internal combustion engine linear generator, Appl. Energy 269 (2020), 115084, https:// doi.org/10.1016/j.apenergy.2020.115084.

[4] A. Thiruvengadam, S. Pradhan, P. Thiruvengadam, et al., Characterization of energy distribution and efficiency in a modern heavy-duty diesel engine [J], SAE International Journal of Engines 13 (4) (2020) 583–599.

[5] J.R. Serrano, F.J. Arnau, J. Martin, et al., Analysis of engine walls thermal insulation: performance and emissions [Z], SAE International (2015), https://doi.org/ 10.4271/2015-01-1660.

[6] C. Binder, F. Abou Nada, M. Richter, et al., Heat loss analysis of a steel piston and a YSZ coated piston in a heavy-duty diesel engine using phosphor thermometry measurements, SAE International Journal of Engines 10 (4) (2017) 1954–1968.

[7] S. Caputo, F. Millo, G. Cifali, et al., Numerical investigation on the effects of different thermal insulation strategies for a passenger car diesel engine [J], SAE Int J Engines 10 (4) (2017) 2154–2165

[8] N. Uchida, H. Osada, A new piston insulation concept for heavy-duty diesel engines to reduce heat loss from the wall [J], SAE Int J Engines 10 (5) (2017) 2565–2574.