OPEN ACCESS Freely available online

International Journal of Advancements in Technology

Opinion Article

Study on the Design and Computational Fluid Dynamics of a Diesel Engine

Aamir Farooq*

Department of Computer Science, National Institute of Technology Kurukshetra, Kurukshetra, India

DESCRIPTION

In this study of a 1500 hp 12-cylinder diesel engine, combustion and emissions is analyzed using three-dimensional (3D) Computational Fluid Dynamics (CFD). Maximum in-cylinder pressure, temperature, NOx (nitrogen oxides) can be obtained. The simulation is only performed for compression, combustion, and expansion strokes when the inlet and outlet valves are closed. The compression pressure and temperature values are used as initial conditions for simulation from a one dimensional (1D) AVL program. Since there are no measurements, we will look at the numerical results of various mesh structures and turbulence models. One eighth of a cylinder, a sector of 45 degrees, is used as a computational domain using these conditions. The mesh study was performed using a coarse mesh and a 4x finer mesh. This study found that the difference between the results of the coarse mesh and the fine mesh was acceptable.

Design of Experiments (DoE) based on engine design optimization is widely used in the automotive industry due to its robustness and efficiency. With the advent of more predictive Computational Fluid Dynamics (CFD) models and powerful computational resources, CFD-based engine optimization using DoE proves its effectiveness in the development of advanced engine concepts. In this high fidelity CFD models are used in combination with DoE for engine design optimization, such as building design spaces, formulating and validating reaction surface models, and optimizing and validating model-based designs. To illustrate the process, we show the current application of Gasoline Compression Ignition (GCI) engine combustion optimization, followed by how to effectively implement a CFD guided engine combustion system optimization campaign using DoE. Computational Fluid Dynamics (CFD) includes analysis of fluid flow, heat transfer, and related systems using computer-assisted simulations. With a wide range of industrial and non-industrial applications, it is a very robust tool for product manufacturing. It is widely used in the automotive industry to predict drag and vehicle lift. Computational fluid dynamics requires important knowledge of fluid mechanics, mathematics, and programming. This involves assuming various variables to generate a model that can capture the requirements of a real system.

Industrial utility are keen on computational fluid dynamics to provide unique advantages over experimental-based methods of fluid or flow system design. Allows unlimited detail of the result and helps to complete the fluid system. This significantly reduces system redesign lead times and costs. CFD facilitates the analysis of systems that are difficult to implement in a controlled system. It also has the ability to probe systems that are more destructive than normal power units. Experimental studies are increasingly ignoring experiments in the industry because of the variable costs of staff recruitment and other aspects. Numerical flow simulations, on the other hand, provide large amounts of results at no additional cost and are very cheap to perform.

CONCLUSION

The finite difference method, the finite element method, and the spectral method are three basic numerical solutions, of which the finite difference method is the most commonly used. Numerical algorithms include the integration of basic fluid flow equations over all finite volumes of the region. The resulting integral equation is transformed into a system of algebraic equations. The algebraic equation is then solved by an iterative process. The fundamental difference between the finite volume method and other CFD methods is the integration of the control volume into the finite volume method. The resulting equation has the same properties for any finite size cell. This simple concept makes it easier for engineers to understand fluid flow compared to other methods. The storage of various flow variables, such as enthalpy and velocity within a finite control volume, is expressed to estimate whether it is increasing or decreasing. Computational fluid dynamics codes consist of discretization techniques that are useful for convection, diffusion, and other important transport phenomena.

Correspondence to: Aamir Farooq, Department of Computer Science, National Institute of Technology Kurukshetra, Kurukshetra, India, Tel/Fax: +44 (0)300 019 6175; E-mail: aamirfarooq@ust.edu

Received: 27-Dec-2022, Manuscript No. IJOAT-22-18014; **Editor assigned:** 29-Dec-2022, Pre Qc No. IJOAT-22-18014 (PQ); **Reviewed:** 12-Jan-2023, Qc No. IJOAT-22-18014; **Revised:** 19-Jan-2022, Manuscript No. IJOAT-22-18014 (R); **Published:** 27-Jan-2022, DOI: 10.35248/2329-6631.23.14.226.

Citation: Farooq A (2022) Study on the Design and Computational Fluid Dynamics of a Diesel Engine. Int J Adv Technol. 14:226.