Identifying the Practical Application of Computational Fluid Dynamics Techniques

[1]Nilesh Parmar, [2]Dr. Annapurna Ramakrishna Shinde
[1]Student, [2]Professor
Department of Mathematics
Dr. A.P.J. Abdul Kalam University, Indore, India

Article Info
Page Number: 12872-12876
Publication Issue:
Vol. 71 No. 4 (2022)

Article History
Article Received: *25 January 2022*
Revised: *30 February 2022*
Accepted: *15 March 2022*

Abstract
For issues involving one or more of the following phenomena fluid flow, heat transfer, mass transfer, and chemical reaction computational fluid dynamics (CFD) is a well-established instrument of choice. This article provides a concise summary of the many ways fluid mechanics may be put to use in everyday life, including in the fields of aviation, engineering, shipbuilding, and more. It lays out the path for the future of computational fluid mechanics research and identifies the problems and gaps in the current state of the field. It is argued that CFD offers many prospects for the study of industrial activity, and that these prospects will grow in importance and breadth as computer technology and software continue to improve.

Keywords: Heat Transfer, Fluid Dynamics, Finite, Differential Equations, Numerical Solutions

---

Introduction

Computational fluid dynamics (CFD) is a relatively new subfield of fluid mechanics that use computational techniques and numerical algorithms to solve and analyse fluid flow issues. There are three primary approaches to research and experimentation in the subject of fluid mechanics. Theoretical or analytical fluid mechanics is the first discipline. Writing down equations, fiddling with them, and solving them by hand are all part of the theoretical fluid mechanics process. The theory of fluid mechanics includes the study of equations such as the Navier-Stokes, which control the motion of incompressible fluids. Second, experimental fluid mechanics is a field that employs many engineers and physicists. Flow and the influence of different disturbances, forms, and stimuli on flow are studied via real physical experiments in experimental fluid mechanics. Pool-generated waves, airflow research in real wind tunnels, waterflow in actual pipes, etc. Finally, the field of computational fluid dynamics (CFD) is attracting an increasing number of professionals from a wide variety of academic disciplines. Waves on water, an aircraft in a wind tunnel, and water moving through pipes may all be simulated in CFD, but now it's all done digitally. as opposed to real, tangible 3D things. The physical principles that regulate the motion of the fluid's molecules are represented by equations that are coded into a computer model. To simulate the visual experience of doing physical experiments, the outcomes of the flow are written to files that may be viewed as images or animations.

CFD simulations and the mathematical models, which are encoded in the computer programme, are validated by comparison to the precise answers in circumstances where an analytical, or theoretical, solution exists. Validation refers to this kind of comparison testing. The state of CFD is not yet advanced enough to employ solutions to problems without verifying them with known, analytical, or accurate solutions. However, validation is not the same as verification; rather, it is a test to ensure that the implemented, coded model faithfully reflects the conceptual, mathematical description and the solution that was meant to be modelled.

However, there are many cases when no analytical answer exists. The employment of computational methods is commonplace in such circumstances. When there isn't a clear answer, CFD may be utilised to get close. When time, money, or other factors make a computer solution preferable, CFD is often used. Ease of use may be attributed to factors like travel duration or threat level.

I. Methods Used In Numerical Solution Ofcomputational Fluid Dynamics

There have been many different numerical solutions developed during the last several decades. The key distinction is the use of discrete and algebraic equations in the regional discrete method to problem solving. CFD numerical solution makes use of the finite element technique, the finite volume approach, and the finite difference method. The finite volume approach is the most used CFD solver method.

Finite Difference Method

The first technique used for computer numerical simulation was the finite difference method (FDM). In place of a continuous solution region, this technique employs a separation grid with a discrete number of nodes. In order to build the algebraic equations with unknown values on the grid nodes, the finite difference approach is utilized to discretize the derivative of the control equation using the difference of the function value on the grid node. This approach is a direct numerical solution to the differential issue that yields an algebraic problem.

Finite volume method

The Finite Volume Method (FVM), also known as the Control Volume Method, is a popular technique for spatial discretization. An integral can be calculated in the domain it governs, and the underlying conservation equation can be solved directly; its origins may be traced back to these two factors. These control differential equations all have a similar form, thus they may be solved in the same way.

Finite element method

The finite element method (FEM) is often used to solids when doing structural analysis, but it may be utilized for fluids as well. The weighted margin approach and the variation principle provide its theoretical foundation. Essentially, the key to finding a solution is to break up the area to be calculated into discrete, non-overlapping pieces. To interpolate the function, choose a few convenient nodes in each instance. Each variable's node value or derivative is combined with the chosen interpolation function to form a linear expression (shape function) that is then used to rewrite the differential equation. The last step is to use a discretization technique to solve the differential equation by using either the variation principle or the weighted margin approach. Weight functions and interpolation functions are constructed with various finite element techniques.

II. Application Of Computational Fluid Dynamics

Aerospace was the first industry to use Computational Fluid Dynamics (CFD). As it has progressed and matured, it has found widespread use in fields as diverse as shipbuilding, chemistry, industrial design, etc. A positive outcome was also seen in the field and based on data from the real scenario.

Here is how computational fluid dynamics (CFD) is used in the car industry. Using CFD technology to examine the car flow field is not only cheap and quick, but also provides a deeper and more thorough knowledge of fluid motion than tests. We are able to comprehend not just the global and regional outcomes of motion, but also the The methodical procedure is becoming more well-known. The majority of computational fluid dynamics (CFD) applications in the automotive industry deal with the modelling of the vehicle's surrounding flow environment.

The wind resistance coefficient and other design characteristics of interest may be determined by computing the flow field around a vehicle's exterior, which yields information on the pressure field, velocity field, aerodynamic force, and aerodynamic moment acting on the vehicle's surface. There is a deeper comprehension of the flow mechanism and the field dispersion of the flow. The results of the comparison with the experiment show that adjusting the geometric characteristics of the automobile body will result in an optimised design. The exterior flow field diagram of a particular automobile body is shown in Figure 1.

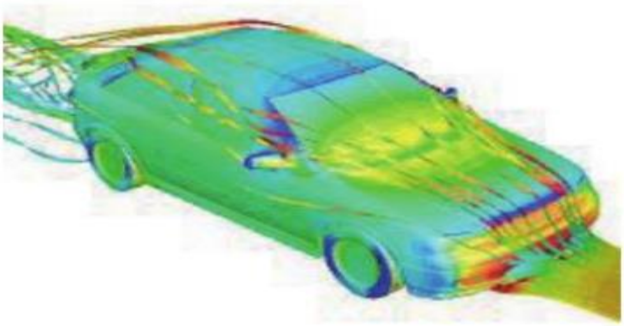

Figure 1: Flow field on the surface of the automobile

.

The design of suitable components or installation dimensions may further minimise wind resistance, and this can be done by both the computation of the exterior flow field of the complete vehicle and the calculation and analysis of the local flow field, which includes things like rearview mirrors, flow deflectors, and wheels. Foreign countries have only conducted preliminary qualitative analysis, but this is undeniably a significant area of research, for the driving conditions of multiple vehicles (three types of meeting, overtaking, and platooning), because its aerodynamic characteristics are a transient process. It's significance for research on managing stability in this condition cannot be overstated. Overtaking is seen in Figure 2's static pressure distribution diagram.

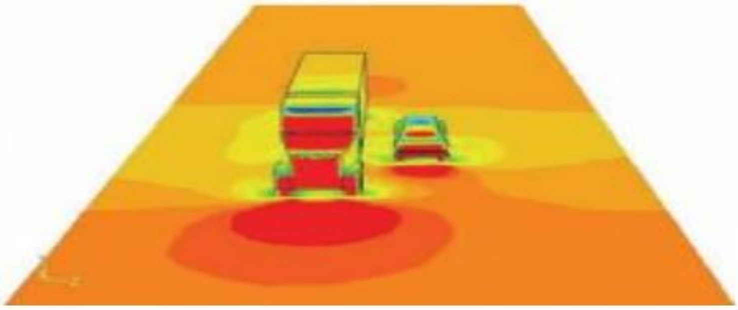

Figure 2: Representation of Static pressure of two overtaking vehicles

The car's interior flow field may also be analyzed using CFD. The heat transfer model may be used to simply determine the temperature field and the velocity field within the climate-controlled vehicle. It is possible to determine the impact of the car's air-conditioning refrigeration system by analyzing the distribution features of the temperature field and the velocity field within the vehicle. The trick is finding methods to lower the air conditioner's energy use and the car's gas usage. The temperature distribution of a Dongfeng Motor Co., Ltd. front seat is shown in Figure 3 with the radiant heat transmission of the geometric surface and the convective heat transfer of the surrounding area taken into account. The buttocks and back become excessively hot because of the friction created between the human body and the seat, and the air flow is disrupted. It is possible to optimize the location of the car's air conditioner, air intake, and air conditioner via an examination of the flow of air throughout the vehicle, which may also help passengers find the most comfortable seating arrangement.

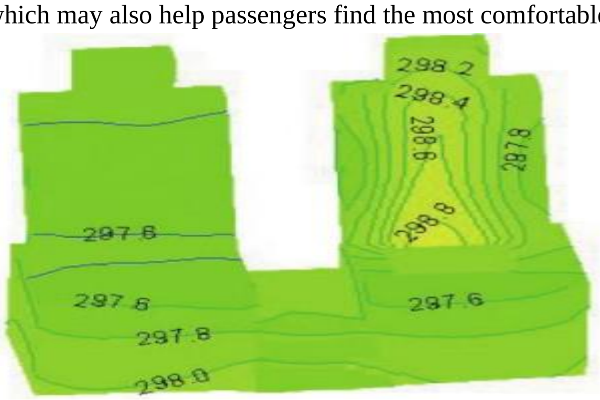

Figure 3:Field distribution of Surface temperature in front seats

CFD is also commonly utilized in the design of hydraulic brakes, hydraulic torque converter blades, cooling fan blades, and oil pumps. The streamline distribution close to the blades in the torque converter's turbine runner is seen in Figure 4. Figure 4 shows a turbine blade (in black) and a vortex (shown by the streamline above the right blade). The creation of secondary flow, outflow, and vortex may be precisely reproduced using three-dimensional simulation computation of the internal flow field of the hydraulic torque converter, and the calculation results can be validated through tests. The theory of three-dimensional flow fields allows for the development of a technique for analyzing designs. Improve the hydraulic torque converter's inefficient transmission, unreliable design, and lengthy development time.

Figure 4: Streamline distribution near turbine blades

In practical applications, such as calculating rotor speeds and flow fields, Computational Fluid Dynamics (CFD) has shown to be an effective tool. Computational fluid dynamics demonstrates the findings. It is of great use in resolving real-world engineering issues. The following is the detailed procedure followed in the computational stream of physical science:

- Construct a model, and then represent it mathematically in light of the relevant domain expertise.
- To analyze different issues and their solutions, use the appropriate tools.

In order to solve real-world engineering problems, computational fluid dynamics (CFD) is used. The following are some of the ways in which it benefits the user:

Before anything further, a more in-depth investigation of fluid flow, the real transfer condition in matter and energy, may be performed. The second advantage of using CFD to analyse an issue is that it allows for the modification of Test parameters, conditions, and other items involved in the test process that would otherwise be difficult to get using a more conventional research methodology. Thirdly, it drastically reduces the amount of time needed for planning and study. The method may be improved using simulation data (point #4) and it can be used in high-risk, high-temperature conditions (point #5).

The field of computer hydrodynamics engineering has seen widespread adoption and steady advancement in the direction of software in recent years. Professional software may be found in a wide variety of business applications. Using programs like CFX and FLUEN, as well as others, we are able to efficiently solve a wide range of challenging and complicated engineering technology challenges, simplifying their solution in the process. It should be emphasized, however, that certain disciplines have not yet reached a mature stage for using computational fluid software. Therefore, in order to further boost its application impact, it is required to deepen its research and improve the elements of function improvement, calculation accuracy, and simplicity of operation.

Combining the computational fluid dynamics simulation tool with today's state-of-the-art computers has led to a novel use of CFD in the real world. The time and money spent on development are cut down significantly. To take the actual application of CFD to the next level and speed up its development, the fundamental objective of CFD researchers in the design and research process is to rapidly and appropriately create efficient and accurate viscous flow calculation techniques.

III. Advantages And Disadvantages Of Cfd

Following are the benefits of the CFD:

- Insight—CFD analysis allows us to practically peek into a design to examine how it functions, which is very useful for devices or system designs that are difficult to analyse or test via experimentation. The insights gained through CFD are invaluable. CFD allows us to see numerous phenomena that would otherwise be hidden from view.

- Foresight—CFD software allows us to speculate on what could occur in a particular scenario. We can quickly foresee the design's performance and test several versions till we find the best one.
- Effectiveness - the insight we get allows us to make more strategic designs that provide better outcomes. By shortening the time it takes to get from concept to prototype, CFD is a useful tool.
  CFD has a number of benefits, but it also has a few of drawbacks.
- Physical models of real-world processes provide the basis for CFD solutions.
- Numerical mistakes are inevitable when using a computer to solve an equation.
- Imprecision in the numerical models that causes truncation mistakes.
- The correctness of the CFD solution is highly dependent on the starting or boundary conditions supplied to the numerical model; • Round-off errors as a result of the computer's limited word size.

IV. Conclusion

Computational fluid dynamics (CFD) is a subfield of fluid mechanics that finds use in a variety of contexts. Significant progress has been made in the use of CFD for drying, sterilization, mixing, and refrigeration in recent years. Because CFD relies on a large number of approximations and assumptions, the simulation results should be checked in the real world. General use of CFD in industrial processing will continue in the 21st century despite the fact that there are still certain barriers to its general implementation, such as the impossibility to accurately simulate huge 3-D issues on a reasonably priced computer, especially in large-scale sophisticated facilities.

References: -

[1] Permanasari, A. A., Rusli, M. H., Puspitasari, P., et al. Computational fluid dynamics heat transfer analysis of double pipe heat exchanger using nanofluid MnFe2O4 with ethylene glycol/water. IOP Conference Series: Materials Science and Engineering, 2021, 1034(1): 012065 (9pp).

[2] Shiralashetti, Siddu&Deshi, A.. (2017). Numerical solution of differential equations arising in fluid dynamics using Legendre wavelet collocation method. International Journal of Computational Materials Science and Engineering. 06. 1750014. 10.1142/S2047684117500142.

[3] Gedik E. Experimental and numerical investigation on laminar pipe flow of magneto-rheological fluids under applied external magnetic field. Journal of Applied Fluid Mechanics. 2017; 10(3).

[4] Sochi T. Fluid flow at branching junctions. International Journal of Fluid Mechanics Research. 2015; 42(1).

[5] Ahmadloo E, Sobhanifar N, Hosseini FS. Computational Fluid Dynamics Study on Water Flow in a Hollow Helical Pipe. Open Journal of Fluid Dynamics. 2014; 4(2):133.

[6] Li X, Wang S. Flow field and pressure loss analysis of junction and its structure optimization of aircraft hydraulic pipe system. Chinese Journal of Aeronautics. 2013; 26(4):1080-92.

[7] Hirani AA, Kiran CU. CFD simulation and analysis of fluid flow parameters within a Y-Shaped Branched Pipe. IOSR Journal of Mechanical and Civil Engineering, 10 (1). 2013:31-4.

[8] Motlagh YG, Ahn HT, Hughes TJ, Calo VM. Simulation of laminar and turbulent concentric pipe flows with the isogeometricvariational multiscale method. Computers & Fluids. 2013; 71:146-55.

[9] Heil, M., Hazel, A. L. Fluid-Structure Interaction in Internal Physiological Flows. Annual Review of Fluid Mechanics, 2011, 43(1): 141-162.