Volume 12, No.11, November 2025
Journal of Global Research in Mathematical Archives
ISSN 2320 – 5822
UGC Approved Journal
RESEARCH PAPER
Available online at http://www.jgrma.info

A REVIEW OF COMPUTATIONAL FLUID DYNAMICS (CFD) APPROACHES FOR THERMAL ANALYSIS AND OPTIMIZATION

Dr. Abid Hussain[1]
[1] Professor, School of Computer Application & Technology & Dean, Research, Career Point University, Kota
abid.hussain@cpur.edu.in

Abstract: Numerous scientific and technological applications, including differential equations, combustion, aircraft, autos, refrigeration, propulsion, heat exchangers, & nuclear engineering, depend heavily on the coupling of heat transfer with fluid flow. Computational fluid dynamics (CFD) has been effectively applied over the years to address a variety of conjugate heat transfer and fluid flow issues using computers. The article provides an in-depth discussion of thermal and cooling methods based on advanced Computational Fluid Dynamics (CFD) technologies in engineering. The analysis of key CFD techniques—i.e., conjugate heat transfer, radiation-convection modeling, and multiphase phase-change processes—aims to achieve precise simulation of coupled thermal phenomena. Various methods, including Design of Experiments (DOE), Response Surface Methodology (RSM), domain decomposition, and AI-enabled approaches, are applied to CFD to enhance computational efficiency. Practical applications in biomedical, mechanical, and missile engineering demonstrate the role of CFD in predicting performance, reducing costs, and improving designs. A thorough literature review also helps identify the current trends and challenges in CFD-based thermal analysis. The study presents an intelligent and efficient integrated framework for the CFD-driven thermal system design.

Keywords: Computational Fluid Dynamics (CFD), Thermal Simulation, Multiphase Heat Transfer, Conjugate Heat Transfer, Radiation-Convection Modeling.

1 INTRODUCTION

Heat transfer is largely dependent on thermal operations, which significantly influence product quality and safety. In the SDG era, energy efficient heat exchanging equipment is essential. A significant area of study on a seldom discussed restriction of fluid flows is convective heat transfer in circular pipes. Convective heat transfer is the principal method of heat transmission, when flow takes place in pipes, such as heat exchangers. It might be enhanced by altering the working fluids' thermophysical characteristics and/or changing the flow's geometry or boundary conditions [1]. Introducing nanoparticles into the base fluid is one way to do this. There is a growing need to produce fluids with improved heat transfer performance, as conventional heat transfer fluids exhibit poor heat transfer performance.

A double-pipe heat exchanger represents one of the many options available for heat exchangers. The system allows heat to be transferred between two fluids by means of conduction across convection inside the fluid and the double pipe wall dividing the fluids[2].Computational fluid dynamics (CFD) is now essential to many fundamental research projects and business issues. Because of advances in computer technology & new numerical techniques, CFD is becoming increasingly important. The fast growth of urbanization and industrialization has led to a sharp rise in the use of fossil fuels. Encouraging energy conservation as well as consumption reduction can help satisfy market demands, safeguard the environment, ease resource restrictions, and advance industrial upgrading [3]. To improve energy utilization, a heat exchanger is an essential tool for transferring heat between solid materials and fluids, or between multiple fluids. It is widely used in many different industries, including the petrochemical, aerospace, and electric generation sectors [4], whereby heat transfer coefficients & total thermal performance may be greatly enhanced by optimizing heat exchanger construction and operation conditions.

Energy processes inside the hydrogen economic chain have been shown to be reliably predicted using AI-based machine learning. In order to tackle the challenges of fluid motion modeling, artificial intelligence (AI)[5] has become a revolutionary technology. The majority of social, scientific, technical, and business domains have been interested in machine learning (ML) in recent years. The development of ML has been fueled by developments in AI and associated techniques, strengthened by "big data" and the availability of affordable computing architectures. ML was first developed at the intersection of computer science and statistical [6]. Construction engineering, aerospace, healthcare, materials research, education, financial modeling, and marketing are among the industries that use machine learning.

1.1 Structure of the paper

The paper is structured in the following way: Section 2 deals with CHT, radiation-convection modeling, multiphase heat transfer and significant CFD techniques, Section 3 compares the optimization techniques, such as DOE, RSM, domain decomposition, and AI, to enhance the efficiency of a simulation, Section 4 describes the CFD in the biomedical, mechanical, and missile engineering, Section 5 examine CFD in thermal analysis and the final section 6 is the conclusion with future directions.

2 COMPUTATIONAL FLUID DYNAMICS STRATEGIES FOR THERMAL SIMULATION

In many cases, flow-related problems are tackled through a few scientific methods, which are theoretically based (analytical) fluid dynamics, experimentally based fluid dynamics, and the use of computational fluid dynamics (CFD) for numerical solutions.

2.1 Conjugate Heat Transfer Modeling

Conjugate heat transfer models are an ancient concept dating back over 50 years. To get analytical or numerical estimates of the heat transfer coefficient that are near to values discovered empirically, one must focus on conjugating the state of the boundary at the fluid–solid interface using coupled analysis of heat transfer[7]. Conjugate Heat Transfer (CHT) evaluation is the term used to describe such a linked field investigation [8]. It should be noted, however, that coupled-field evaluation is not limited to CHT studies that combine fluid and solid domains. Figure 1 shows an example of the CHT:

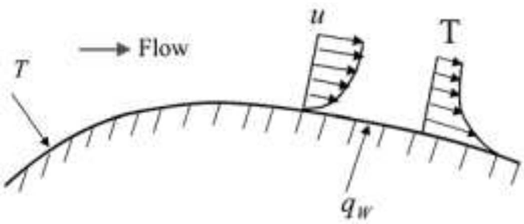

Figure 1: Heat transfer that occurs as a combination of methods includes firstly convection which takes place from a solid surface to a fluid moving outside and secondly the process of internal heat conduction.

2.1.1 Conjugate Heat Transfer Investigations in Cooled Turbine Blades using the Preconditioned Density Based-Methods

A preconditioned density technique was utilized to examine a cooled turbine blade's heat transfer characteristic. In the fluid domain, the Favre-averaged compressible Navier-Stokes equations in polar coordinates are considered. The Weiss & Smith preconditioning matrix-based approach has been used to address several numerical problems in low-Mach-number flows. The fluid flow solver has been linked with a Fourier heat conduction solver to undertake conjugate heat transfer investigations [9]. The construction of a conduction solver has taken into consideration the Fourier heat conduction equation in integral form[10]. Convective fluxes have been calculated using central differencing in conjunction with an artificial dissipation matrix. On the other hand, gradients calculated at the cell centroids were employed in the derivation of diffusive fluxes by central differencing.

2.1.2 Time-Accurate CFD Conjugate Evaluation of Transient Heat Transfer Coefficient Measurements in a Channel Equipped with Pin Fins

The main aim of this research was to determine the exact answer by examining the errors in transient heat transfer coefficient measurements arising from zero-dimensional or one-dimensional heat transport assumptions. When using the 0-D or 1-D exact solutions, the transient techniques require that the HTC and the freestream or core temperature characterizing the convection environment are constants, in addition to assuming that the conduction into the solid is 0-D or 1-D. A precise substitute to improve forecast accuracy has been suggested: computational fluid dynamics-based CHT research.

2.1.3 Simulation of Conjugate Heat Transfer for a Thermocouple Sensor in a Low-Temperature Nitrogen Gas Environment:

The CHT has been used to study how a thermocouple responds to an abrupt change in the surrounding temperature. The computational study has used the OPENFOAM computational framework. Incompressible transient equations of laminar flow in the fluid domain have been resolved using the PISO (Pressure Implicit with Splitting of Operators) method. The fluid domain has been thought of as a nitrogen gas field, whereas the solid region has been modelled as a thermocouple section. To ensure continuity of temperature and heat flow, the interface boundary between the fluid and solid domains has been established.

2.2 Radiation and Natural Convection Model

The radiation and the natural convection model explain the transfer of heat in which both thermal radiation and natural convection take place. Within the procedure, surfaces release and capture radiation based on temperature and radiance, whereas buoyancy-driven air movement conveys heat by convection. The technique frequently requires to resolve the connected energy and momentum equations to consider both causes, and it is usually along with the boundary conditions for temperature as well as those for surface characteristics. It is routinely used in the fields of enclosures, electronic cooling, and building design to analyze temperature distribution and optimize thermal performance.

2.2.1 Thermal Radiation Modeling

The CFD simulation tool, containment FOAM, serves to simulate the current tests. It is based on analyses of mixing and transport processes, as well as the pressurization of large dry PWR containments. Open FOAM, an open-source CFD toolset, serves as its foundation. In containment FOAM, several more containment-specific numerical models, as well as libraries, are built [11]. Models of buoyancy turbulence, multi-species transport, condensation, thermal radiation, and aerosol transport, among others, are included in the list of most frequent applications. It is possible that the thermal radiation heat transmission even, which is not the main heat transfer mechanism, cause a significant change of the density field and, thus, the temperature distribution. Because radiation heat transfer can affect buoyancy-driven flows and mixing processes, it must be accounted for in containment flows.

2.2.2 Natural Convection Heat Modeling:

Its providing dependable and energy-efficient passive cooling systems, Natural Convection Heat Transfer plays a vital role in resolving these issues [12]. Because of their practical significance, horizontal and sloped closed rectangular enclosures are extensively used as structures for natural convection research. Internal items are frequently housed within these enclosures in a variety of heating circumstances, which have a substantial impact on heat transfer and flow behaviour. Internal items increase complexity by altering the thermal and flow fields, thereby affecting the system's overall effectiveness.

2.3 Multiphase and Phase-Change Heat Transfer

The movement of thermal energy throughout a process involving a temperature differential that later results in temperature changes and redistribution is referred to as heat transfer. Heat transfer and multiphase flow are essential in both established and developing fields of engineering study. Novel technologies and creative applications for multiphase flow and heat transfer are being developed and researched in an unceasing stream and exhibit significant promise due to the rapid growth of diverse multidisciplinary disciplines and technologies. Phase shift thermal energy storage technique offers a lot of potential to improve the stability of erratic renewable energy sources & increase the cost-effectiveness of energy use systems [13]. Phase-shift thermal energy storage offers significant potential to improve the stability of erratic renewable energy sources & increase the cost-effectiveness of energy use networks.

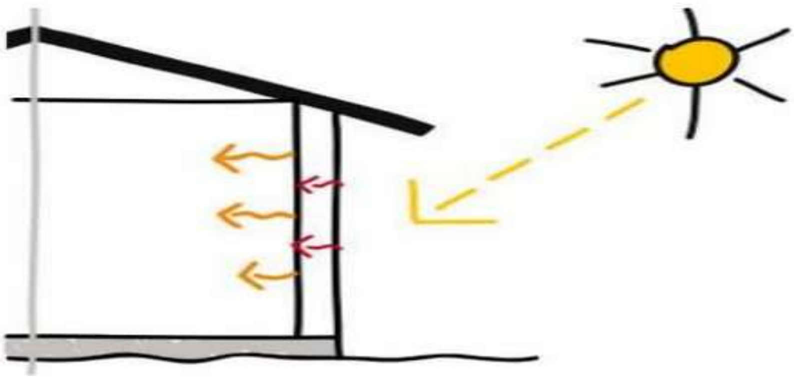

Figure 2: Heat Transfer Process

Figure 2 describes a heat transfer process through a wall room. It simply depicts a house with sunlight striking the exterior wall. The arrows show the heat from the is being reflected into the wall. This heat reflection clearly demonstrates the effect of solar radiation onto to the wall surface.

2.3.1 Multiphase Heat Transfer Methods:

The technique of raising the rate of heat transfer—or, more accurately, enhancing performance—is known as heat transfer enhancement [14]. Techniques for increasing heat transfer are the subject of numerous experimental studies.

- **Nucleate Boiling:** The movement of thermal energy throughout a process involving a temperature differential that later results in temperature changes, including redistribution, is referred to as heat transfer. It is simple to enumerate a number of nucleate boiling research procedures, some of which are unrelated.
- **Film Boiling:** Film boiling causes the generation of a vapour column over the part surface. This isolating blanket significantly reduces the heat flux from the component's surface at this phase, which is primarily due to radiation. The vapour cloud begins to break down in certain portions of the component when the surface and core cool sufficiently that the heat flow is insufficient to sustain film boiling. On the same portion, film boiling can take anything from few seconds to tens of seconds.
- **Evaporation:** The process of vaporizing huge amounts of volatile liquid to produce a concentrated product is called evaporation. Evaporators are devices that facilitate evaporation. To produce latent heat of vaporization, heat is supplied to

the evaporator [15], which then transfers it to the evaporating liquid. Typically, steam is utilized as a heat source. Evaporation refers to a very peculiar and unique surface phenomenon, in which mass is carried up from the surface. Therefore, there is no boiling.

2.3.2 Phase Change Heat Transfer

In a variety of technological applications, including electronic cooling, chemical reactions, space temperature management, microfluidic preparation, and biomedical engineering, etc., vapor-liquid phase shift has long been of significant interest.

- **Condensation:** High-heat-flux electronic machinery can be cooled by a vapour chamber. A vapour chamber experimental setup was developed in this work to visually examine the heat transfer performance of a proposed vapour chamber with a side window during condensation and evaporation. Additionally, the observed temperatures of the fluid close to the evaporator surface as well as the vapour near the condenser surface were used to determine the condensation and evaporation heat transfer coefficients. The creation of a high-performance vapour chamber requires an understanding of boiling, evaporation, & condensation processes, as well as the behavior of vapor-liquid two-phase flow.
- **Melting:** In particular, melting heat is crucial in changing the fluid's temperature delivery. Transportation of colloidal nanoparticles within fluids is a key scientific issue today and has plenty of scope in the field of nano dynamics. These particles' arbitrary measurement is explained by Brownian motion, and their thermally induced behavior is reflected in thermophoresis [16]. The flow of all properties can be significantly affected by the interaction among these events. Activation energy is one of the most important factors to consider in many processes, such as chemical reactions. It is an essential part of chemical processing as well as associated industries since it reflects the energy threshold which has to be exceeded for interactions within the fluid.
- **Solidification:** Since the composition of the solid result typically changes from that of the original liquid when a liquid comprising two or more components solidifies, fluid mechanics may serve an essential part in the phase transitions that precede solidification. For instance, in the semi-conductor business, liquids having similar amounts of gallium and germanium may partly solidify to make virtually pure germanium, whereas salty water in the polar oceans solidifies to form nearly pure ice.

3 OPTIMIZATION STRATEGIES INTEGRATED WITH CFD

An optimization procedure based solely on semi-analytical frameworks is frequently used, and the evaluation is restricted to single design factors [17], or CFD optimization is typically not feasible if computer resources are scarce. An optimization strategy that initially investigates possible optimal setups can reduce the computational cost of CFD simulations.

3.1 Design of Experiments (DOE) and Resource Surface Methodology (RSM)

The components may be analyzed and optimized using CFD, greatly reducing the cost of experimental testing. DOE and CFD techniques were used to improve the bow form of a tanker hull. This resulted in a hull capable of reducing the extra friction in waves. Combining CFD and RSM analysis [18] can shorten computation times and be an effective optimization technique.

3.1.1 Implementation of Design of Experiments with CFD (DOE-CFD)

The computational fluid dynamics (CFD) approach may be used to simulate the hydrodynamic response of classifiers to changes in operating parameters, as hydraulic classifiers operate according to the governing principles of fluid flow within a predetermined shape. Additionally, CFD employs a highly persuasive, non-intrusive virtual modelling approach with strong visualization capabilities, so the information gathered shows which modifications meet the design requirements. In addition to the benefits of the CFD approach, engineers can only model the process response using a one-factor-at-a-time approach while maintaining constant levels for other factors; as a result, interaction effects among dependent variables are neglected [19]. Applying a methodical strategy to research to ensure one may concurrently evaluate all elements is an improved method which has lately caught engineers' attention. This method, known as design of experiments (DOE), uses statistical analysis to provide valuable insights into how operational parameters interact and how the system functions as a whole.

3.1.2 Implementation of the Resource Surface Methodology with CFD

The response surface methodology (RSM) is the most often used technique to assess the most suitable circumstances of numerous factors in research assessing the influence of numerous variables on one or more response values [20]. RSM is a special experimental design that may be used to reduce resource waste, increase operational process utilization, and develop protocols for innovative products. The ability to examine several parameters with a limited number of samples is RSM's unique characteristic in experimental design. Once data collection is complete, regression analysis is used to determine the mathematical relationship between the system response and its influencing factors.

3.2 Domain Decomposition Strategies

The multi-block structured mesh's domain decomposition is a complex process[21]because physical block calculations and block-to-processor ratios vary widely.

- **Circular Decomposition of Regions:** The initial process of distributing computing power to different areas involves ordering the physical blocks from the one with the least computational demand to the one with the highest computational demand and noting their positions in the array, termed Sort. Following the procedure, the spatio-temporal distribution happens considering the specific workings of individual regions. Since the total computation may not be evenly distributed among the processors, an ideal number of CPUs is first determined; if this improves load balance, the distribution is adjusted by adding one more CPU. Let Wi be the workload for the ith area, NP be the total number of processors, and ND be the total number of distributed-memory parallel regionsLet Wi represent the workload of the ith region, representing all distributed-memory parallel regions.

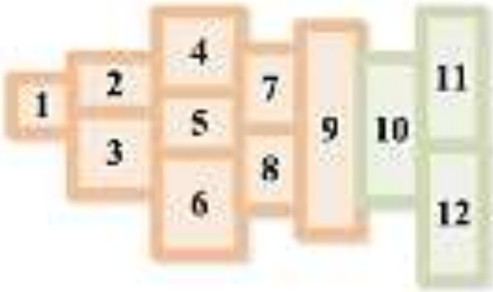

Figure 3: Circular Decomposition of Multi-Regions

Figure 3 clearly depicts the blocks into which the computational domain has been partitioned. The first group (blocks 1–9) is mainly the simulation area and where the majority of the calculations are performed. The blocks (10–12) are considered as a connecting or secondary region. The arrangement of these blocks indicates how the domain is divided for parallel processing, which, in turn, helps maintain good mesh connectivity, evenly distribute the workload, and efficiently assign processors. This configuration results in faster and more robust CFD experiments.

- **Undirected Graph:** Undirected graph division fundamentally differs from general graphs in that the blocks that correspond to the regions indicate the vertices of the undirected graph, which may be divided throughout the modeling process. The linking features among physical blocks are represented by the connecting edge. It should be noted that segmenting undirected graphs is a dynamic procedure in which the number of vertices, their weights, the number of edges, and their weights are always changing.

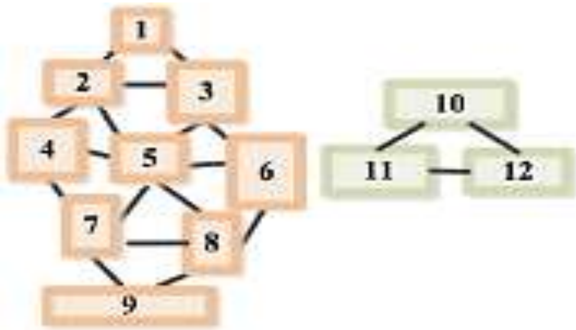

Figure 4: Undirected Graph

Figure 4 uses an undirected graph to show how the computing pieces are connected. There are many interactions in that region because the nine blocks form a dense mesh. Nevertheless, the three blocks from smaller, simpler groups can constitute an interface or additional space. Because of the aforementioned factor, the arrangement shown illustrates block-to-block communication and the need for appropriate domain decomposition to speed up parallel CFD simulations.

3.3 AI-Based Optimization

A revolutionary method in product design engineering is the combination of AI in Multiphysics fluid dynamics modelling [22]. The effectiveness of classic Computational Fluid Dynamics (CFD) methods, which are very powerful, is often limited by difficult processing, costly high simulations, and the impossibility of dealing with Multiphysics coupled phenomena. In this scenario, industries are seeking faster, more effective, and cheaper solutions [23]. By improving prediction accuracy, speeding up simulation processes, and enabling real-time design iteration across a variety of fluid dynamics applications, such as aerodynamics, thermal management, and fluid-structure interaction, AI-based optimization addresses these issues.

3.3.1 Machine Learning Driven Optimization Strategies

It's significantly increasing computational correctness, economy, and physical consistency [24]. The field of fluid simulations has fundamentally transformed through the application of ML techniques in computational fluid dynamics (CFD). There are several ML models suitable for CFD optimization [25]. The choice of model depends on the task at hand, such as predictive modeling, optimization, or simulation acceleration. Figure 5 shows ML domains:

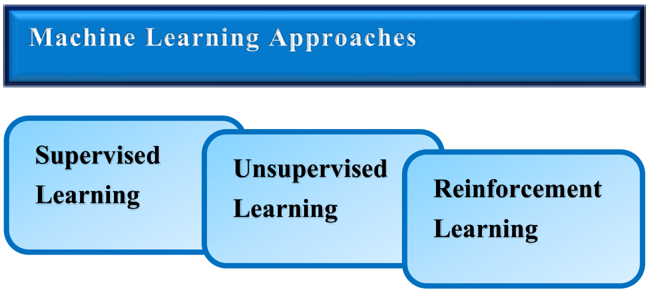

Figure 5: Machine Learning Pipeline

- **Supervised Learning Models (SVM):** These models, like RF or SVM, are used if there are labelled datasets [26]. In various frameworks, supervised learning is often used to reduce the computational domain for simulations, to tune turbulence models, or to estimate boundary conditions.
- **Reinforcement Learning (RL):** When creating control methods for dynamical systems, RL is a very good fit. The set of equations controlling fluid dynamics is a notable example of this a dynamical system. According to recent study findings [27], RL-augmented CFD algorithms can surpass the state of the art in certain areas, such as turbulence modelling.
- **Unsupervised Learning:** The difficulty of this approach is somewhat higher than that of supervised learning. This is because it gives the computer instructions to acquire a talent that do not teach it. Instead of producing categorization, this learning method makes choices that maximize rewards [28]. Unsupervised learning is a technique used by certain self-organizing neural networks to identify latent patterns in unlabeled data.

3.3.2 Data-Driven Surrogate Models

It divides these techniques into three groups according to the kind of discretization:

- **Regular Grids:** This method provided a different way of looking at things for the treatment of non-linear systems in fluid dynamics, especially in the case of dynamic systems. Together, these researches are indicating the growing power of ML in the field of fluid dynamics [29]. They highlight the combination of classical fluid dynamics principles with data-driven technologies as an effective approach, especially in areas such as regular grids and discretization-based methods.
- **Irregular Mesh:** The use of irregular meshes poses a problem for the regular grid-based surrogates, which in turn encourages the use of flexible solutions such as Graph Neural Networks (GNNs) for various structures and sizes. The GNS model and MeshGraphNets. Both approaches use GNNs, but the first depicts physical systems as particles in a graph, using learned message passing for dynamics computation, while the second performs complex events on unstructured meshes, demonstrating the networks' ability to handle irregular topology.
- **Lagrangian Particles:** There are multiple ways to represent fluids, and one of the most widely used among them is the particle-based Lagrangian representation, which is also very popular. However, in complex environments, fluid samples are usually composed of hundreds of thousands of particles, or even more.

4 IMPLICATIONS OF CFD FOR THERMAL SYSTEM ARCHITECTURE

Computational Fluid Dynamics (CFD) sees wide applications in different fields. The following section discusses the various applications:

4.1 Biomedical Engineering

In the biomedical field, CFD is frequently used to address challenging problems. CFD is becoming a vital factor in the development of modern designs and refinements through computer simulations, leading to lower operational costs and higher efficiency. Below is a list of many CFD uses in biomedical engineering:

- **Heart Pumping:** The sophisticated and versatile CFD technology is used in cardiac blood flow models, besides pressure, velocity, and flow pattern analysis throughout the cardiac cycle. The method offers a wide range of uses, among which the

evaluation of pumping efficiency, the detection of disturbances such as turbulence or backflow, and the creation of artificial medical devices, such as valves and heart pumps, are the most important.
- **Blood flows through arteries and veins:** CFD simulates the blood flow through the vessels and thus, it can easily detect shear stress, pressure changes, and flow characteristics. This technology helps recognize the risks of plaque accumulation or aneurysms, study the effects of stenosis, and develop optimal designs for stents or grafts for surgical procedures.
- **Air flow in lungs:** The flow behavior in the respiratory tract can be examined, and the lungs' distribution of the ventilation airflow analyzed by the use of CFD simulations. Inhalation of drugs via inhalers, assessment of ventilator effectiveness, and diagnosis of respiratory disorders such as asthma or COPD are some applications where this technique is highly useful.

4.2 Mechanical Engineering

Computational fluid dynamics (CFD) is used in systems to simultaneously study and optimize heat transfer, aerodynamics, and fluid flow. It achieves this by simulating the complex flow patterns and allowing the designer to see through them. CFD is used in the industrial [30], automotive, and aerospace sectors not only to replace a great number of physical tests with simulations, but also to speed up design, increase reliability, and promote creativity.

- **Heat Exchanger:** The following categories of research in different kinds of CFD has been used to study heat exchangers: thermal analysis, fouling, pressure loss, and fluid flow maldistribution throughout the design and optimization stage.
- **Combustion of IC Engines:** The representation of a spark ignition (SI) engine is created using computational fluid dynamics, and the engine model is then applied to an intermediate internal combustion (IC) engine. The development of a model is a support tool for engineering students in grappling with the interactions among species' movement, flow patterns, temperature distributions, and chemical reactions in SI engines.
- **Turbo Machinery:** The movement and heat exchange in nuclear and thermal power plants, as well as HVAC systems, were considered for casting simulation, and the manufacturing of plastics through injection molding. In the hydrodynamics of torpedoes, ships, submarines, etc.

4.3 Missile Engineering

CFD is one of the most important methods in the missile design process because it enables aerodynamic analysis at supersonic speeds, simulates shock waves, and investigates pressure and temperature distributions. It not only allows drag reduction by changing the missile shape, but also improves its performance by enabling safe assessment and optimization of its thermal loads during re-entry. CFD not only cuts down on the amount of wind tunnel testing required but also makes the whole design process faster, better performance and greater mission reliability.

- The Aerothermal environment and the aerodynamic loads (force & moments) are among the most important factors in missile engineering, and they are already becoming more complicated due to the increasing complexity of modern structures. Aerodynamic data are used to evaluate manoeuvrability, mission performance, and stability and control.
- The exploratory method's test findings are extended and supplemented by the use of CFD.
- Furthermore, CFD's flow visualization may be quite helpful throughout every stage of wind tunnel testing. Surface pressure distributions, wake-line filaments, and other visual aids provide opportunities to "observe" the flow and its interactions with various aircraft components.
- In the wind tunnel testing phase, the CFD bridge provided knowledge for crucial design choices [31]. For instance, the scale-model design greatly benefits from inviscid calculations.

5 LITERATURE REVIEW

This section commits to the literature review, which focuses on the use of Computational Fluid Dynamics (CFD) in Thermal Analysis, and one of its major advantages is that it can accurately represent the interaction between heat and fluid motion. Even the problems concerning turbulence modeling and experimentation have not been completely solved, albeit the efficiency and precision have been improved. For further enhancements in thermal techniques, CFD remains an essential method.

Gadhavi *et al.* (2025) used this simulation to test a number of basic principles of the finite volume discretization technique, including the energy equation model and the K-Epsilon model. The aim of this work is to create safer, more efficient Li-ion batteries. This paper makes the case for the advancement of lithium-ion batteries, which ultimately lead to improved battery pack performance [32].

Kang *et al.* (2025) conducted a thermal study of a BLDC motor with a semi-closed outer rotor designed for drone applications, focusing on the cooling ventilation effects generated by the motor's airflow. The thermal analysis method based on CFD–LPTN offers a useful and verified approach to assessing the thermal performance of air-cooled motors in drones for different operating conditions [33].

Jegede, Fayomi and Azubuike (2024) the enhancement of thermodynamic performance of thermal systems utilizing energy, exergy and advanced simulation methods is the aim of the study. Special attention is given to the growing influence of Computational Fluid

Dynamics (CFD) in maximizing the system's efficiency. Integration of energy and exergy analyses with CFD, as outlined in this thorough review, is suggested as a pathway for the design and operation of thermal systems, thereby supporting sustainability and reducing environmental harm [34].

Goyal *et al.* (2024) focus on three key factors influencing battery cooling: cell placement geometry within the battery pack, the use of heat sinks, and the spacing between the fins around the radiator. The simulations compare the square 16-cell battery pack and the honeycomb 16-cell pack, the effectiveness of heat sink designs with fins versus without fins, and the thermal performance of different fin spacings (5mm, 7.5mm, 10mm, and 12.5mm) in radiators. The findings demonstrate that honeycomb structure results in a lesser number of hotspots but overall cooling is provided by an inline square battery back and heat sinks with fins provide better cooling efficiency [35].

Park *et al.* (2024) studied thermal robustness of an axial-flux permanent magnet synchronous motor (AFPMSM) for underwater propulsion applications. The dual-rotor with a yokeless and segmented armature structure is considered to achieve high power density. A cooling system through water with a channel has been mounted on the windings of the motor which lets the water get into the cooling channels by itself through hydrodynamic pressure created during the propelling. The Computation fluid dynamics (CFD) is employed for the model evaluation [36].

Dehgosha *et al*. (2023) use CFD to conduct a thermal analysis of a 2.3 kW, completely enclosed, rotor-excited axial-flux switching permanent magnet (RE-AFSPM) machine for electric vehicles. Electromagnetic analysis has used the three-dimensional finite element method to calculate losses with precision. The average temperature of the components in steady-state has been calculated for the conditions of normal operation, different speeds, and different values of external convection coefficients [37].

Table 1 highlights recent studies on Computational Fluid Dynamics (CFD) in Thermal Analysis, highlighting the study focus, techniques, significant achievements, advantages as well as applications.

Table 1: Existing Literature of Recent Studies on Computational Fluid Dynamics in Thermal Analysis

Reference	Study Focus	Technology Used	Key Contributions	Improvements Reported	Application Domain
Gadhavi et al. (2025)	Safer & efficient Li-ion batteries	Finite Volume Method, Energy Equation, k-ε Model	Applied CFD principles to enhance battery safety and performance	Improved battery pack efficiency	Lithium-ion battery systems
Kang et al. (2025)	Thermal analysis of drone BLDC motors	CFD–LPTN framework	Validated airflow-induced thermal cooling model	Accurate evaluation of motor cooling under various conditions	Drone propulsion systems
Jegede, Fayomi & Azubuike (2024)	Thermodynamic performance of thermal systems	CFD, Energy & Exergy Analysis	Integration of CFD with energy–exergy methods	Improved system efficiency & sustainability	Thermal/energy systems
Goyal et al. (2024)	Battery cooling effectiveness	CFD, Heat Sinks, Geometry Analysis	Study of cell arrangement & radiator fin spacing	Honeycomb reduces hotspots; finned heat sinks enhance cooling	Battery thermal management
Park et al. (2024)	Thermal stability of AFPMSM for underwater propulsion	CFD, Water-cooling channels	Dual-rotor yokeless structure with natural hydrodynamic cooling	Enhanced power density & thermal stability	Underwater motors
Dehgosha et al. (2023)	Thermal analysis of AFSPM motor for EVs	CFD + 3D FEM	Loss calculation and steady-state temperature evaluation	Accurate thermal prediction & performance optimization	Electric vehicle motors

6 CONCLUSION AND FUTURE WORK

The purpose of CFD simulation is to examine the flow field and heat transfer mechanisms in syngas cooling heat exchangers. The primary approach used in the study is Computational Fluid Dynamics (CFD) for the thermal simulation and analysis of complex fluid-thermal interactions, with high accuracy across the whole system, which can be applied in different engineering domains. It is through the application of main CFD techniques - such as conjugate heat transfer, radiation-convection modeling, and multiphase phase-change mechanisms - that the research emphasizes the coupling of physical phenomena to obtain accurate thermal predictions. The integration of optimization strategies such as DOE, RSM, domain decomposition, and AI-based methods has greatly improved simulation efficiency and, at the same time, enabled CFD-based design exploration. The practical applications in biomedical, mechanical, and missile engineering have provided further evidence of CFD's capability not only to reduce experimental costs but also to improve system performance and accelerate innovation.

Research shows that the future of MPLS VPN will cover the need for cost-effective, robust, and comprehensive services. The development of Multiphysics coupling, PINNs, adaptive meshing, and GPU acceleration will not only improve speed but also accuracy, allowing very efficient CFD applications in aerospace, biomedical, autonomous systems, and next-generation thermal designs.

REFERENCES

[1] W. Kang, Y. Shin, and H. Cho, "Economic Analysis of Flat-Plate and U-Tube Solar Collectors Using an Al2O3 Nanofluid," *Energies*, vol. 10, no. 11, 2017, doi: 10.3390/en10111911.

[2] A. S. Alhulaifi, "Computational Fluid Dynamics Heat Transfer Analysis of Double Pipe Heat Exchanger and Flow Characteristics Using Nanofluid TiO2 with Water," *Designs*, vol. 8, no. 3, 2024, doi: 10.3390/designs8030039.

[3] B. Hong *et al.*, "Evaluation of disaster-bearing capacity for natural gas pipeline under third-party damage based on optimized probabilistic neural network," *J. Clean. Prod.*, vol. 428, p. 139247, 2023, doi: https://doi.org/10.1016/j.jclepro.2023.139247.

[4] Z. Chen *et al.*, "Optimization Design and Performance Study of a Heat Exchanger for an Oil and Gas Recovery System in an Oil Depot," *Energies*, vol. 17, no. 11, 2024, doi: 10.3390/en17112631.

[5] S. Narang, "Zero-Trust Security in Intrusion Detection Networks: An AI-Powered Threat Detection in Cloud Environment," *Int. J. Sci. Res. Manag.*, vol. 4, 2025, doi: 10.38124/ijsrmt.v4i5.542.

[6] N. Malali, "Model Validation and Governance for AI / ML in Actuarial Applications," *TIJER – Int. Res. J.*, vol. 12, no. 4, 2025.

[7] B. John, P. Senthilkumar, and S. Sadasivan, "Applied and Theoretical Aspects of Conjugate Heat Transfer Analysis: A Review," *Arch. Comput. Methods Eng.*, no. March 2020, 2018, doi: 10.1007/s11831-018-9252-9.

[8] R. T. Patel, Rutvik, "Advancements in Data Center Engineering : Optimizing Thermal Management , HVAC Systems , and Structural Reliability," *Int. J. Res. Anal. Rev.*, vol. 8, no. 2, pp. 991–996, 2021.

[9] H. Zhang, Z. Zou, Y. Li, J. Ye, and S. Song, "Conjugate heat transfer investigations of turbine vane based on transition models," *Chinese J. Aeronaut.*, vol. 26, pp. 890–897, 2013, doi: 10.1016/j.cja.2013.04.024.

[10] P. B. Patel, "Thermal Efficiency and Design Considerations in Liquid Cooling Systems," *Int. J. Eng. Sci. Math.*, vol. 10, no. 3, pp. 181–195, 2021.

[11] R. Kapulla, L. Xiongguo, S. Kelm, U. Doll, S. Paranjape, and D. Paladino, "Importance, influence and limits of CFD radiation modeling for containment atmosphere simulations," *Nucl. Eng. Des.*, vol. 411, p. 112408, 2023, doi: https://doi.org/10.1016/j.nucengdes.2023.112408.

[12] A. Jobby, M. Khatamifar, and W. Lin, "A Comprehensive Review on the Natural Convection Heat Transfer in Horizontal and Inclined Closed Rectangular Enclosures with Internal Objects at Various Heating Conditions," *Energies*, vol. 18, no. 4, 2025, doi: 10.3390/en18040950.

[13] X. Zhai *et al.*, "Phase change thermal energy storage: Materials and heat transfer enhancement methods," *J. Energy Storage*, vol. 123, p. 116778, 2025, doi: https://doi.org/10.1016/j.est.2025.116778.

[14] C. S. Meena, A. Kumar, S. Roy, A. Cannavale, and A. Ghosh, "Review on Boiling Heat Transfer Enhancement Techniques," *Energies*, vol. 15, no. 15, 2022, doi: 10.3390/en15155759.

[15] A. Prasad and G. R. Prasobh, "A Review on Evaporation," vol. 7, no. 2, pp. 273–285, 2022, doi: 10.35629/7781-0702273285.

[16] K. Sudarmozhi, D. Iranian, and N. Alessa, "Investigation of melting heat effect on fluid flow with brownian motion/thermophoresis effects in the occurrence of energy on a stretching sheet," *Alexandria Eng. J.*, vol. 94, pp. 366–376, 2024, doi: https://doi.org/10.1016/j.aej.2024.03.065.

[17] M. Della Monica and M. Bernardini, "Strategies for CFD Investigation and Optimization of the External Aerodynamics of a Sounding Rocket," *Aerotec. Missili Spaz.*, 2025, doi: 10.1007/s42496-025-00299-9.

[18] J. Taghinezhad, S. Abdoli, V. Silva, S. Sheidaei, R. Alimardani, and E. Mahmoodi, "Computational fluid dynamic and response surface methodology coupling: A new method for optimization of the duct to be used in ducted wind turbines.," *Heliyon*, vol. 9, no. 6, p. e17057, Jun. 2023, doi: 10.1016/j.heliyon.2023.e17057.

[19] H. Khoshdast, V. Shojaei, and H. Khoshdast, "Combined application of computational fluid dynamics (CFD) and design of experiments (DOE) to hydrodynamic simulation of a coal classifier," vol. 1, pp. 9–22, 2017, doi: 10.22059/ijmge.2016.218483.594634.

[20] H.-Y. Chen and C. Chen, "A Study of the Response Surface Methodology Model with Regression Analysis in Three Fields of Engineering," *Appl. Syst. Innov.*, vol. 8, no. 4, 2025, doi: 10.3390/asi8040099.

[21] X. Lv, J. Xu, and Z. Liu, "Optimization strategies for multi-block structured CFD simulation based on Sunway TaihuLight," no. February 2023, pp. 1–18, 2025, doi: 10.1002/eng2.12661.

[22] V. Shah, "Traffic Intelligence In Iot And Cloud Networks: Tools For Monitoring, Security, And Optimization," *Int. J. Recent Technol. Sci. Manag.*, vol. 9, no. 5, pp. 138–147, 2024.

[23] D. Patel, "AI-Enhanced Natural Language Processing for Improving Web Page Classification Accuracy," vol. 4, no. 1, pp. 133–140, 2024, doi: 10.56472/25832646/JETA-V4I1P119.

[24] A. M. Aly, "Machine Learning Reshaping Computational Fluid Dynamics: A Paradigm Shift in Accuracy and Speed," *Fluids*, vol. 10, no. 10, 2025, doi: 10.3390/fluids10100275.

[25] N. Prajapati, "The Role of Machine Learning in Big Data Analytics: Tools, Techniques, and Applications," *ESP J. Eng. Technol. Adv.*, vol. 5, no. 2, pp. 16–22, 2025, doi: 10.56472/25832646/JETA-V5I2P103.
[26] S. Hazra and I. A. Chowdhury, "Advancing Computational Fluid Dynamics through Machine Learning : A Review of Data-Driven Innovations and Applications," no. October, 2024, doi: 10.46610/JFMMD.2024.v06i02.005.
[27] M. Kurz and O. Philipp, "Deep Reinforcement Learning for Computational Fluid Dynamics on HPC Systems," 2022.
[28] I. C. Udousoro, "Machine Learning: A Review," *Semicond. Sci. Inf. Devices*, vol. 2, no. 2, pp. 5–14, Nov. 2020, doi: 10.30564/ssid.v2i2.1931.
[29] H. Wang *et al.*, "Recent Advances on Machine Learning for Computational Fluid Dynamics : A Survey," vol. 14, no. 8, pp. 1–22, 2023.
[30] V. Thakran, "Impact of Advanced Materials in Enhancing the Mechanical Properties of Piping Systems for Stress Analysis," *Int. J. Recent Technol. Sci. Manag.*, vol. 7, no. 4, pp. 66–74, 2022.
[31] R. K. Raman, Y. Dewang, and J. Raghuwanshi, "A review on applications of computational fluid dynamics," no. July, 2018.
[32] P. Gadhavi, P. Shah, R. Trivedi, and P. Parikh, "Analysis of Li Ion Battery using Computational Fluid Dynamics," in *2025 International Conference on Advances in Modern Age Technologies for Health and Engineering Science (AMATHE)*, 2025, pp. 1–5. doi: 10.1109/AMATHE65477.2025.11081206.
[33] J. Kang *et al.*, "Thermal Analysis of Outer Rotor BLDC Motor for Drone Considering Airflow Ventilation Cooling," *IEEE Access*, vol. 13, pp. 128518–128531, 2025, doi: 10.1109/ACCESS.2025.3586240.
[34] E. O. Jegede, O. S. Fayomi, and U. G. Azubuike, "Thermodynamic Performance Enhancement in Thermal Systems: A Review of Energy, Exergy, and Computational Fluid Dynamics Analyses," in *2024 IEEE 5th International Conference on Electro-Computing Technologies for Humanity (NIGERCON)*, 2024, pp. 1–5. doi: 10.1109/NIGERCON62786.2024.10926957.
[35] D. Goyal, R. Kaur, A. Sood, I. Sain, S. Bhandari, and A. Kumar, "CFD and Thermal Analysis of Electric Vehicle Battery Pack," in *2024 IEEE 5th India Council International Subsections Conference (INDISCON)*, 2024, pp. 1–6. doi: 10.1109/INDISCON62179.2024.10744203.
[36] J.-H. Park, J.-G. Lee, Y.-H. Jeong, K. Kim, I. Park, and D.-K. Hong, "CFD Thermal Analysis of Axial-Flux Permanent Magnet Synchronous Motor with Water Cooling Channels for Underwater Propulsion Applications," in *2024 27th International Conference on Electrical Machines and Systems (ICEMS)*, 2024, pp. 2233–2236. doi: 10.23919/ICEMS60997.2024.10921300.
[37] P. Dehgosha, A. Zarghani, H. Torkaman, and A. Ghaheri, "Three-Dimensional Thermal Analysis of a Rotor-Excited Axial Flux Switching Permanent Magnet Machine by Computational Fluid Dynamics Method," in *2023 3rd International Conference on Electrical Machines and Drives (ICEMD)*, 2023, pp. 1–6. doi: 10.1109/ICEMD60816.2023.10429151.