STM JOURNALS

International Journal of
Electrical Machine Analysis and Design

Volume 2, Issue 2, 2024
July—December
DOI (Journal): 10.37591/IJEMAD

https://journals.stmjournals.com/ijemad

Review IJEMAD

CFD Analysis: Double Pipe Heat Exchanger, Latent Heat Storage, and BIM-CFD Integration

Vaishnavi Kaje[1], Satyam Shah[1,*], Kulkarni Shirish Prabhakarrao[2]

Abstract

This is a review work covering four research papers. Heat exchangers transport thermal energy across fluids; research in SolidWorks 2014 compared flow simulation to the effectiveness-NTU approach. Slow PCM charging/discharging is a global concern in latent heat storage; research on the influence of flow rate and temperature on melting and solidification durations discovered efficiency in a triple tube heat exchanger design. Advances in computational fluid dynamics for forecasting separated flows in aircraft design are also presented. Designing and analysing heat exchangers, storage systems, and building infrastructure has been completely transformed by the incorporation of Computational Fluid Dynamics (CFD) into thermal systems. The optimisation of double-pipe heat exchangers, the use of latent heat storage systems, and the combination of CFD and Building Information Modelling (BIM) for sophisticated simulation in architectural projects are the three related applications examined in this study. Improving simulation accuracy, energy efficiency, and thermal performance are the main goals. Heat exchanger and thermal energy storage system design and optimisation have been completely transformed by the incorporation of Computational Fluid Dynamics (CFD) into thermal systems analysis. This article examines the use of CFD in the analysis of latent heat storage and double pipe heat exchangers, as well as the new trend of integrating CFD with Building Information Modelling (BIM). The study highlights how important these technologies are for improving thermal performance, increasing energy efficiency, and bridging the gap between design and real-world application.

Keywords: BIM-CFD Integration, thermal performance integration, flow simulation, infrastructure design optimization

INTRODUCTION

Computational fluid dynamics (CFD) allows pilots to manipulate aerodynamic characteristics like lift, incidence angle, speed, and geometry to achieve specific flow conditions. This study explores the evolution of the CFD methodology to study separation problems in military aircraft, highlighting their importance for accurate flight performance prediction.

Advancements in AEC have improved industrial facility design and control, utilizing Building Information Modelling (BIM) for efficient planning, design, and procurement, reducing project time and costs. BIM-CFD integration improves construction design processes by promoting stakeholder involvement and efficient communication. It enhances energy performance simulation and green building design assessment in the built environment. This study reviews BIM and CFD applications in the

***Author for Correspondence**
Satyam shah
E-mail: vaishnavikaje271@gmail.com

[1]Student, Department of Space Engineering, Ajeenkya DY Patil University, Pune, Maharashtra, India
[2]Assistant Professor, Department of Space Engineering, Ajeenkya DY Patil University, Pune, Maharashtra, India

Received Date: October 30, 2024
Accepted Date: November 16, 2024
Published Date: November 25, 2024

Citation: Vaishnavi Kaje, Satyam Shah, Kulkarni Shirish Prabhakarrao. CFD Analysis: Double Pipe Heat Exchanger, Latent Heat Storage, and BIM-CFD Integration. International Journal of Electrical Machine Analysis and Design. 2024; 2(2): 19–24p.

AEC industry, covering architectural design, building management, energy assessment, HVAC system design, and facility safety management, and proposes future research directions.

Renewable energy sources like solar require efficient storage systems for high demand, with PCMs being more efficient than traditional heat storage systems, considering charging and discharging times. In this study, simulation studies with the conditions prevailing to the energy storage required for space heating are presented. The effects of HTF inlet temperature and flow rate on the charging and discharging times were analysed for both configurations hence a comparison is done for the same amount of PCM. Also, the melting and solidification characteristics of PCM are examined with variable HTF flow rate and inlet HTF temperature. The study aims to analyse the thermal performance and effectiveness of a parallel flow double pipe heat exchanger using computational fluid dynamics simulations.

From industrial heat transfer devices to sustainable building concepts, CFD analysis has revolutionised the understanding and use of thermal systems. Latent heat storage systems and double-pipe heat exchangers provide examples of how CFD can improve energy efficiency and performance. The future of comprehensive and data-driven architectural design lies in the combination of CFD and BIM. For more intelligent and flexible thermal management solutions, future research should concentrate on sophisticated multi-physics simulations and real-time CFD analysis integration with IoT-enabled devices.

Because of their simple design and efficient way of controlling the thermal exchange between two fluids, double-pipe heat exchangers are frequently utilised in industrial settings. The optimisation of parameters like flow rate, temperature gradient, and material selection is the main goal of the CFD analysis of these systems.

LITERATURE REVIEW

With a particular emphasis on NATO aircraft, Rizzi and Luckring examined and investigated the use of CFD simulations for forecasting separated flows and the corresponding aerodynamic performance throughout the flying envelope. The review comprises six case studies to evaluate the current capabilities of CFD, a historical sketch of seven decades of CFD developments which have addressed many of the challenges of separated flow predictions, a summary of the physics of flow separation that is particularly challenging to model numerically, and possible futures for further improvements in the simulations using CFD [1].

The representative combined BIM and CFD application cases in content analysis by Kang *et al*. are divided into three different regions. The functionality, interoperability, and sustainability of this kind of integration in architectural, engineering, and construction (AEC) projects are explained in depth. Future studies that integrate CFD and BIM applications are also investigated. Specifically, the more advanced hazard analysis that is proposed demonstrates the power of such integration [2].

Consideration is given to Shukla and Kishan's shell and tube heat exchanger, which has a heat transfer fluid on the tube side and a phase-changing substance on the shell side. Heat transfer fluid (HTF) single and double pass (counter and parallel) configurations are used to examine the impact of flow rate and inlet temperature on melting and solidification times. The fundamental challenge in melting the PCM is that the liquid portion stays at the top during the discharging process while the solid (un-melted) portion accumulates at the bottom during the charging process. This significantly reduces the system's efficiency. Two shell and tube heat exchanger configurations, double and triple tube, were examined in this study in an effort to increase system efficiency, and it was discovered that the latter instance performed better [3].

An apparatus that transfers thermal energy (enthalpy) between two or more fluids, between a fluid and a solid surface, or between solid particles and a fluid, at specific temperatures and in thermal contact, is

called a heat exchanger. In many process industries, heat exchangers are crucial engineering components because their performance has a significant impact on the process's economy and efficiency.

In Solid Works 2014, Mangi *et al*. focused on modelling a shell and tube parallel flow heat exchanger and configuring a flow simulation by adding boundary conditions, performing computations, adding surface parameters, and visualising the resultant flow field with cut plots and flow trajectories [4].

Gorgulu discussed a CFD study of parallel flow twin pipe heat exchanger, emphasising the efficiency and thermal performance of these systems. The study uses comprehensive CFD simulations to assess the heat exchanger's efficiency while taking operating conditions into account. The CFD model incorporates precise geometrical representations and real-world fluid parameters to simulate the parallel flow inside a double pipe heat exchanger [5].

In order to investigate the heat transfer phenomena in phase change materials (PCMs), Abduljalil *et al*. reported earlier research on the numerical modelling of PCMs using self-developed programming and commercial computational fluid dynamic (CFD) software. The implementation of PCMs in various engineering applications, including electronic cooling technologies, building thermal storage, heating, ventilation, and air conditioning (HVAC), is progressively simulated using the CFD (fluent) program [6].

Heterogeneous modelling with realistic aggregate input and distributive analysis of a particular constituent of interest can be carried out using the approach used by Wang *et al*. Evidence of digital damage in the non-destructive measurement is used to demonstrate the need for customised computer vision in the use of concrete composite [7].

He *et al*. suggested a BIM technique to assist with the digital production and detailed geometric design of modular homes. Through talks on its ability to generate the geometry features of 3D-printed modules, a program interfaced within BIM is developed and presented [8].

The ability of the CFD modelling technique to precisely depict the procedures within the thermal storage system with the PCM was confirmed by Maghalseh *et al*. CFD modelling with FLUENT ANSYS was done for validation, and the expected outcomes were contrasted with empirical and mathematical information from the literature [9].

Khairulmaini *et al*. examined the heat transfer performance of STHE using Computational Fluid Dynamics (CFD) simulation combined with Computer Aided Design (CAD). A 10-in PVC shell, a 12-mm copper tube, and the STHE measurements were taken from an earlier study. At flow rates, cold and hot water at 20 and 70°C ran in parallel [10].

COMPUTATIONAL FLUID DYNAMICS IN AEROSPACE

Aircraft performance is influenced by aerodynamic characteristics such as lift-curve slope, maximum lift, angle of attack, and stall. As the angle of attack increases, buffet occurs, leading to chaotic separated flow over the aircraft. This flow can be classified into three sectors: cruise, manoeuvre, and post-stall manoeuvre. Computational fluid dynamics (CFD) has been developed to simulate flight conditions of military aircraft in these regions. Challenges include classical boundary layer separation, shock-induced separation, and hybrid wings with shock/boundary-layer interaction. This study explores the evolution of computational fluid dynamics (CFD) methodology for accurate aircraft performance prediction. It discusses the challenges faced by missiles and launch vehicles, such as high-speed separation, multibody booster separation, and complex vortex-shock interactions. The study also presents examples of separated flow simulations that are still problematical today, and discusses the outlook for further developments to advance CFD to treat even these cases. The authors also highlight the benefits of a teamed and sustained collaborative campaign for understanding complex aerodynamic phenomena.

CFD requires simulation of relevant flow physics with sufficient fidelity for predictive capability. Collaborative assessments are spawning new wind-tunnel tests to better understand focus topics and provide validation-class data for code assessment and improvement. This approach, called "Unit Problems," is fruitful for future advancements in computational aerodynamics. Separation, a hard problem, can be studied in CFD, allowing interpretations coupled to the physics involved. A working definition of separation could be that vorticity from the boundary layer is convect away from the body surface. CFD visualization aids in identifying separation regions, while Poincare's critical-point theory aids in understanding complex flows. Military aircraft operations face separated flow challenges.

BIM Techniques

BIM techniques support design in AEC projects, improving standardization since ISO 19650. However, complexity and data exchange issues may hinder user requirements. This study explores the integration of Building Information Modelling (BIM) and computational fluid dynamics (CFD) technology in the design perspective. BIM is semantically based and object-oriented, while CFD is procedural-oriented. Both models offer precise 3D modelling, allowing engineers to build and modify practical designs quickly. The integration can improve applications in architectural design, energy assessment, HVAC system design, and facility safety management.

Renewable energy sources like solar require efficient storage systems to provide energy during surplus supply and high demand. Solar thermal to hot water systems, are low-efficient and limited in utility. PCMs, which can be organic, inorganic, or liquid metals, are more efficient than sensible heat storage systems. Design considerations include charging and discharging times. This study presents simulation studies on energy storage for space heating, analysing the effects of HTF inlet temperature and flow rate on charging and discharging times. The study examines the melting and solidification characteristics of the PCM with variable HTF flow rate and inlet HTF temperature. The governing equations include conservation equations for mass, momentum, and energy. The enthalpy-porosity model is adopted for modelling PCM melting and solidification processes. The simulation model uses the Energy, Turbulence, and Melting and Solidification module of ANSYS fluent 18.1, considering density variation and specific heat and thermal conductivity.

A heat exchanger transfers thermal energy between two fluids at different temperatures and in thermal contact. Common applications include heating or cooling fluid streams and dissipating them. Shell and tube heat exchangers have a series of tubes, with one set containing the fluid and the other running over the tubes. These heat exchangers are typically used for high-pressure applications due to their robust shape. Flow simulations show temperature distributions along the shell and tube, and the effectiveness of NTU method is used for calculation. The shell and tube heat exchanger's effectiveness was calculated using solid works software and flow simulation. The NTU method yielded an effectiveness of 0.133, a 7.3% difference from the flow simulation method.

Double pipe heat exchangers are essential in thermal engineering for efficient energy transfer across various industrial processes. The parallel flow configuration in these heat exchangers presents unique thermal dynamics that warrant detailed investigation. Recent studies have explored various approaches to enhance the thermal efficiency of double pipe heat exchangers, including the use of nanofluids and structural enhancements. For instance, a study using Titanium Dioxide (TiO_2) nanofluid and ethylene glycol/water mixture analysed heat transfer characteristics using computational fluid dynamics (CFD) simulations. Another study used twisted tape with triangular cut to improve performance. Another study examined the impact of the inner pipe's embedded fin shape on heat transfer efficiency. Another study assessed the effects of helical wire inserts on heat transfer and pressure drop in a double pipe heat exchanger. Another study used Al_2O_3/water nanofluid in a mini double-pipe heat exchanger under turbulent flow conditions, resulting in a 12% increase in heat transfer. This study aims to comprehensively analyse the thermal performance and effectiveness of a parallel flow double pipe heat exchanger using CFD.

CONCLUSION

This study provides a summary of computational fluid dynamics (CFD) capabilities for predicting separated flows relevant to military aircraft operations. Separated flows can affect vehicle performance characteristics and performance enhancements. The study presents a seven-decade summary of CFD methods' evolution, focusing on separated flow simulation capability. Six case studies discuss flow complexity and the prospects for further advances. The study suggests hybrid RANS/LES approaches for capturing unsteady flow content, but acknowledges the need for faster algorithms and collaborative campaigns.

This study reviews the literature on BIM-CFD integration in architectural design, building management, energy, HVAC system design, and facility safety management. It highlights its potential in simulating complex buildings and providing information for decision-making. The study proposes a future study to expand the application of integrated BIM and CFD, focusing on quantitative risk assessment for a more sustainable and comfortable built environment.

The study conducted numerical simulations on the charging and discharging time of a PCM cylinder, revealing that HTF inlet temperature significantly impacts charging and discharging, with partial charging being more effective. The shell and tube heat exchanger design and CFD analysis were compared with the effectiveness of NTU method, revealing significant deviations from actual values.

The study analyses a parallel flow DPHE, revealing effective heat transfer of approximately 77%. The design, characterized by laminar flow conditions, was aided by the use of aluminium due to its excellent thermal conductivity. The findings highlight the importance of considering both complex and simple design approaches in developing efficient thermal systems.

Computational Fluid Dynamics in Military Aircraft Operations:

- Discusses the evolution of computational fluid dynamics (CFD) methods for predicting separated flows in military aircraft operations.
- Highlights the impact of separated flows on vehicle performance characteristics and enhancements.
- Presents six case studies on flow complexity and future advancements.
- Suggests hybrid RANS/LES approaches for capturing unsteady flow content, but acknowledges the need for faster algorithms and collaborative campaigns.
- Reviews BIM-CFD integration in architectural design, building management, energy, HVAC system design, and facility safety management.
- Conducts numerical simulations on the charging and discharging time of a PCM cylinder, revealing HTF inlet temperature significantly impacts charging and discharging.
- Analyses a parallel flow DPHE, revealing effective heat transfer of approximately 77%.
- Highlights the importance of considering both complex and simple design approaches in developing efficient thermal systems.

REFERENCES

1. Arthur Rizzi, Luckring James M. Historical development and use of CFD for separated flow simulations relevant to military aircraft. Aerosp Sci Technol. 2021 Oct; 117: 106940.
2. Kang Ki-Yeob, Xiangyu Wang, Jun Wang, Shuyuan Xu, Wenchi Shou, Yanhui Sun. Utility of BIM-CFD Integration in the Design and Performance Analysis for Buildings and Infrastructures of Architecture, Engineering and Construction Industry. Buildings. 2022; 12(5): 651. https://doi.org/10.3390/buildings12050651
3. Pushpendra Kumar Shukla, Anil Kishan P. CFD Analysis of Latent Heat Energy Storage System with Different Geometric Configurations and Flow Conditions. Proceedings of the 25th National and 3rd International ISHMT-ASTFE Heat and Mass Transfer Conference (IHMTC-2019), IIT Roorkee, Roorkee, India. 2019 Dec 28–31.

4. Mangi Naveen Kumar, Shaik Azger Pasha, Prashanthi K, Yogesh Avula. CFD Analysis of Double Pipe Parallel Flow Heat Exchanger. Int J Sci Prog Res. 2016; 21(01): 51–53.
5. Yasin Furkan Gorgulu. CFD Analysis of Parallel Flow in Double Pipe Heat Exchangers: Evaluating Thermal Performance and Effectiveness. 15th International Congress Scientific Research, China to Adriatic. Baku, Azerbaijan. 2024 Jan 7–9.
6. Al-abidi Abduljalil A, Sohif Bin Mat, Sopian K, Sulaiman MY, Abdulrahman Th Mohammed. CFD applications for latent heat thermal energy storage: a review. Renew Sustain Energy Rev. 2013 Apr; 20: 353–363.
7. Wang M, Yang X, Wang W. Establishing a 3D aggregates database from X-ray CT scans of bulk concrete. Constr Build Mater. 2022 Jan 10; 315: 125740.
8. He R, Li M, Gan VJ, Ma J. BIM-enabled computerized design and digital fabrication of industrialized buildings: A case study. J Clean Prod. 2021 Jan 1; 278: 123505.
9. Maher Mohammad Al-Maghalseh. CFD Model of Shell-and-Tube Latent Heat Thermal Storage Unit Using Paraffin as a PCM. In: Paraffin - Thermal Energy Storage Applications. IntechOpen; 2022 Apr 20. 10.5772/intechopen.95847
10. Khairulmaini M, et al. Analyzing the influence of diameter and winding on heat transfer efficiency in spiral tube heat exchangers: A CAD-Integrated CFD Study Using Solidworks Flow Simulation. J Phys: Conf Ser. 2024; 2688: 012002.

