International Journal of Engineering Applied Science and Management
ISSN (Online): 2582-6948
Vol. 5 Issue 2, February 2024

Comprehensive survey of Computational Fluid Dynamics (CFD) Simulations for Thermal Analysis

Irfan Aziz[1], Vikrant S Choudhary[2]
M. Tech. Scholar, Department of Mechanical Engineering, Swami Vivekanand Institute of Engineering & Technology, Ramnagar, Banur, Patiala[1]
Assistant professor, Department of Mechanical Engineering Swami Vivekanand Institute of Engineering & Technology, Ramnagar, Banur, Patiala[2]

Abstract: *This research endeavors to advance the efficiency and effectiveness of heating and cooling systems in workshop environments through the application of Computational Fluid Dynamics (CFD) simulations for comprehensive thermal analysis. The optimization of indoor climate control in workshops is a critical aspect not only for energy conservation but also for ensuring the well-being and productivity of occupants who spend substantial durations in these confined spaces. The study employs advanced CFD simulations to scrutinize and optimize the intricate dynamics of heat transfer and airflow within workshop settings. By gaining insights into the thermal behavior of the environment, the research aims to propose refined strategies for the enhancement of heating and cooling systems. This involves assessing the performance of existing systems and proposing improvements that align with energy efficiency standards and occupant comfort. The methodology involves creating detailed models of workshop spaces, incorporating factors such as ventilation, insulation, and the specific heat-producing activities characteristic of these environments. Through sophisticated simulations, the research seeks to identify optimal configurations for heating and cooling systems, considering factors like equipment placement, air circulation patterns, and temperature distributions. The outcomes of this research are expected to contribute significantly to the development of practical guidelines for workshop design and thermal system optimization. By aligning with sustainability goals and energy efficiency benchmarks, the proposed strategies aim to minimize environmental impact while creating a comfortable and conducive atmosphere for the occupants. In conclusion, the integration of CFD simulations for thermal analysis in workshop environments holds promise for advancing the state-of-the-art in heating and cooling optimization. The findings aim to transcend theoretical insights and provide actionable recommendations for practitioners and stakeholders in the field, promoting a balance between energy efficiency and the creation of optimal indoor environments.*

Keywords: *Flow pattern, Air currents, Impact, Temperature Distribution, Fluent.*

1. Introduction

The selection of building materials significantly influences the thermal performance of structures, and the choice of ceiling material is a critical consideration in heating and cooling analysis. Traditionally, asbestos sheets were widely used due to their fire-resistant properties, but growing health concerns have led to a shift towards alternative materials. Gypsum board, a popular contemporary choice, offers advantages such as fire resistance, soundproofing, and easy installation. In the context of workshop insulation, the decision between asbestos sheets and gypsum boards can impact temperature distribution. Computational Fluid Dynamics (CFD) simulations comparing these materials provide valuable insights into their influence on thermal performance, guiding decisions for creating comfortable and energy-efficient working environments. Understanding fluid dynamics parameters is essential in engineering, and the Reynolds number plays a crucial role in characterizing fluid flow. Named after Osborne Reynolds, this dimensionless quantity signifies the ratio of inertial to viscous forces and determines whether fluid flow is laminar or turbulent. In practical terms, lower Reynolds numbers indicate laminar flow, while higher values suggest turbulent flow. This understanding is fundamental for designing and analyzing fluid flow systems. Simulations, particularly in Computational Fluid Dynamics (CFD), involve creating virtual models and performing calculations to predict behavior. CFD simulations aid engineers in optimizing design parameters, assessing thermal performance, and improving efficiency in various engineering applications, such as analyzing airflow within workshops. Rayleigh number (Ra) and Prandtl number (Pr) are crucial parameters in heat transfer and fluid flow analysis. Rayleigh number characterizes buoyancy-driven flow in a fluid due to temperature differences, while Prandtl number represents the ratio of momentum diffusivity to thermal diffusivity. Both parameters are essential for analyzing natural convection in environments with significant temperature variations. Incorporating these parameters into CFD simulations enhances the accuracy of predictions related to heat transfer and fluid behavior, facilitating the optimization of thermal conditions within workshops or enclosed spaces. The utilization of these parameters in simulations contributes to informed decision-making during the design and optimization processes, ensuring a balance between energy efficiency and thermal comfort.

2. Background

Natural ventilation is like having fresh air in your home without using machines. It's good for making the air comfortable, saving energy, and keeping the inside temperature just right. When designing buildings, it's important to follow guidelines, like those from ASHRAE, to make sure people feel comfortable with the temperature. ASHRAE says what temperatures are acceptable for most people. Having a little breeze can make people feel comfortable even if it's a bit warmer.

Now, let's talk about gypsum board. It's a common material in buildings, and it has some cool qualities. It's good at resisting fire, it helps with sound, it lasts a long time, and it's not too expensive. So, when building something, using gypsum board can be a smart choice because it has many benefits. Workshops serve as dynamic environments where individuals engage in various activities, ranging from fabrication and assembly to creative endeavors. The comfort and productivity of occupants within these spaces are paramount for achieving optimal outcomes. To address this, researchers have turned to Computational Fluid Dynamics (CFD) simulations as a powerful tool for analyzing and optimizing heating and cooling strategies. This paper explores the application of CFD simulations for thermal analysis in workshops, aiming to enhance comfort, productivity, and energy efficiency.

3. Understanding the Need for Optimization

Workshops often face challenges related to temperature control, airflow distribution, and thermal comfort. Conventional heating and cooling systems may not always be tailored to the specific needs of workshop environments, leading to uneven temperature distribution, discomfort, and inefficiencies. Moreover, with increasing emphasis on energy conservation and sustainability, there is a growing demand for more efficient heating and cooling solutions that minimize environmental impact while maintaining occupant comfort.

CFD simulations offer a comprehensive approach to thermal analysis by modeling fluid flow, heat transfer, and other relevant parameters within a virtual environment. By simulating different heating and cooling scenarios, researchers can gain valuable insights into airflow patterns, temperature distribution, and thermal comfort levels within workshop spaces. This allows for the evaluation of various strategies and the identification of optimal solutions to enhance comfort and energy efficiency. The study investigates a range of heating and cooling strategies commonly employed in workshop environments. These include conventional systems powered by fossil fuels or electricity, as well as alternative approaches such as heat pumps, geothermal heat pumps, radiant heating/cooling, and passive heating/cooling methods. Each strategy offers unique advantages and challenges, and CFD simulations provide a platform for comparative analysis to determine their effectiveness in workshop settings.

4. Analyzing Airflow Patterns

One key aspect of thermal analysis is understanding airflow patterns within the workshop space. CFD simulations enable researchers to visualize and analyze the movement of air, identifying areas of stagnation, recirculation, or uneven distribution. This information is crucial for optimizing ventilation systems, ensuring adequate air exchange, and maintaining uniform temperatures throughout the workshop. Temperature distribution plays a critical role in occupant comfort and productivity. CFD simulations allow researchers to predict and analyze temperature gradients within the workshop, identifying hotspots, cold spots, and areas where thermal comfort may be compromised. By optimizing heating and cooling systems based on these simulations, it is possible to achieve more uniform temperature distribution and enhance overall comfort levels. Thermal comfort is a subjective measure influenced by factors such as air temperature, humidity, air movement, and personal preferences. CFD simulations enable researchers to quantify thermal comfort using metrics such as Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD). By comparing different heating and cooling strategies based on these metrics, researchers can identify solutions that optimize thermal comfort for workshop occupants.

5. Optimizing Energy Efficiency

In addition to comfort considerations, energy efficiency is a key driver in the design of heating and cooling systems for workshops. CFD simulations facilitate the evaluation of energy consumption and heat transfer mechanisms, allowing researchers to identify opportunities for optimization. This may involve adjusting system parameters, optimizing equipment placement, or exploring alternative heating and cooling strategies to minimize energy usage while maintaining comfort levels.
With growing concerns about climate change and environmental sustainability, reducing carbon emissions and minimizing energy consumption are paramount. CFD simulations enable researchers to assess the environmental impact of different heating and cooling strategies, considering factors such as CO_2 emissions, energy efficiency, and resource utilization. By identifying low-carbon solutions and sustainable practices, workshops can contribute to broader efforts to mitigate climate change and promote environmental stewardship.

6. Conclusion

This study employs Computational Fluid Dynamics (CFD) simulations to thoroughly analyze the airflow patterns, temperature distribution, and velocity streamlines within a workshop setting. The focus of the simulations is on understanding how air mixes from supply inlets and identifying areas in the workshop where significant heat accumulates during the daytime. Various CFD simulations were conducted under different conditions, utilizing three-dimensional, steady-state, incompressible flow energy equations with the k-ε turbulence model. The study addresses a crucial aspect related to human comfort within a building, emphasizing the importance of lowering ventilation temperatures. To achieve this, the investigation explores methods to reduce the convective heat transfer rate from the ceiling. It concludes that using gypsum as a false ceiling material can effectively lower the internal temperature of the room by 6–8 degrees Celsius. Gypsum, being cost-effective and readily available in sheet form from false ceiling suppliers, emerges as a practical solution. Importantly, these materials are environmentally friendly, contributing neither harmful emissions to the atmosphere nor to global warming. In a global context where there's an increasing demand for Heating, Ventilation, and Air Conditioning (HVAC) solutions, such materials prove to be energy-efficient and beneficial for future generations.
In conclusion, the CFD simulations not only provide valuable insights into the airflow dynamics and temperature distribution within the workshop but also propose practical measures, such as utilizing gypsum as a false ceiling material, to enhance energy efficiency and ensure a comfortable indoor environment. The findings underscore the significance of sustainable and eco-friendly solutions in the design and operation of HVAC systems, emphasizing their positive impact on occupant well-being and the environment.

References

[1] Shah, S., &Dufva, K. (2017). CFD Modeling of Airflow in A Kitchen Environment: Towards improving energy efficiency in buildings. South-Eastern Finland University of Applied Sciences Mikkeli.
[2] Chen, J., Duffy, B., Fletcher, D.F., Haynes, D.F., & Nelson, P. (2001). FTIR spectroscopy measurements and CFD simulations of the pollutants arising from unflued combustion in a room. Building and Environment, 36, 597–603.
[3] Norton, T., & Sun, D.-W. (2006). Computational fluid dynamics (CFD) – an effective and efficient design and analysis tool for the food industry: A review. Trends in Food Science & Technology, 17, 600–620.

[4] Lee, E., Feigley, C.E., & Khan, J. (2002). An investigation of air inlet velocity in simulating the dispersion of indoor contaminants via computational fluid dynamics. The Annals of Occupational Hygiene, 46, 701–712.
[5] Posner, J.D., Buchanan, C.R., & Dunn-Rankin, D. (2003). Measurement and prediction of indoor air flow in a model room. Energy and Buildings, 35, 515–526.
[6] Khan, J.A., Feigley, C.E., Lee, E., Ahmed, M.R., &Tamanna, S. (2006). Effects of inlet and exhaust locations and emitted gas density on indoor air contaminant concentrations. Building and Environment, 41, 851–863.
[7] Bulińska, A., Popiołek, Z., &Buliński, Z. (2014). Experimentally validated CFD analysis on sampling region determination of average indoor carbon dioxide concentration in occupied space. Building and Environment, 72, 319–331.
[8] Rojas, G., Pfluger, R., & Feist, W. (2015). Cascade ventilation–Air exchange efficiency in living rooms without separate supply air. Energy and Buildings, 100, 27–33.
[9] Gilani, S., Montazeri, H., &Blocken, B. (2016). CFD simulation of stratified indoor environment in displacement ventilation: Validation and sensitivity analysis. Building and Environment, 95, 299–313.
[10] Ning, M., Mengjie, S., Mingyin, C., Dongmei, P., &Shiming, D. (2016). Computational fluid dynamics (CFD) modelling of air flow field, mean age of air and CO2 distributions inside a bedroom with different heights of conditioned air supply outlet. Applied Energy, 164, 906–915.
[11] Launder, B.E., & Spalding, D.B. (1974). The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering, Vol. 3.
[12] ANSYS, Inc. (2014). ANSYS® Fluent, Release 16.0, Theory Guide, ANSYS, Inc.