DIFFERENTIAL EQUATION MODELS FOR SIMULATING FLUID DYNAMICS AND HEAT TRANSFER IN INDUSTRIAL PROCESSE

Position 1
M.Madhavi, Lecturer in Mathematics, Government Degree College, Tuni, Kakinada, Andhra Pradesh - 533401
malireddymadhavi@gmail.com
Position 2
Dr. Ram Chandra, Assistant Professor, Department of Applied Science and Humanities, GL Bajaj Institute of Technology and Management, Greater Noida, Uttar Pradesh, 201306
Email: rchandrabhu25@gmail.com
Position 3
Uma A L, Assistant Professor, Department of Mathematics, Bangalore Institute of Technology
Email id: al.uma@bit-bangalore.edu.in
Position 4
Mr. M. Ranjit Kumar, Assistant Professor, Department of Mathematics, Samskruti College of Engineering and Technology, Ghatkesar, Telangana - 501301
mrranjeeth@gmail.com

Abstract**—**Heat transfer and fluid dynamics are important in many industrial processes (chemical manufacturing, energy production, materials engineering, etc.). The models to centralize these phenomena involve differential equation models especially partial differential equations (PDEs). In this paper, the authors aim at providing the detailed analysis of mathematical modeling of fluid flow and thermal transport using differential equations. We discuss the theoretical formulation, examine what has been done before, suggest a better methodology and demonstrate the correctness of the model by simulation results. These findings highlight the relevance of proper modeling towards the optimization of industrial systems in terms of performance, safety and energy efficiency.

Keywords— Differential equations, fluid dynamics, heat transfer, Navier-Stokes equations, thermal conduction, industrial processes, simulation models, PDEs, computational fluid dynamics (CFD), numerical methods.

I. INTRODUCTION

Fluid dynamics and heat transfer in industrial systems encode the crucial processes in a chemical reaction to the conversion and the thermal management of energy. The effective transport of mass, momentum and energy is fundamental to processes like petroleum refining, cooling nuclear reactors, air conditioning systems and designing of heat exchangers. Proper modeling and simulation of these physical processes are key to optimization of efficiency, minimization of operational cost and safety. Since these systems are generally complex (involving, e.g., turbulent flows, phase changes, large temperature gradients, etc.) and usually have a complicated geometry, simple empirical approaches are not adequate. This requires the closely mathematical tools, as mostly differential equations to model and study behavior of fluids and thermal energy causing different conditions [1].

Computational fluid dynamics (CFD) is a branch of science that has expanded tremendously over the past decades with the increase in mathematical modeling and computer power. The heart of CFD is partial differential equations (PDEs) that govern fluid flow, e.g. the Navier-Stokes equations, and thermal transport, e.g. the heat conduction equations or convection-diffusion equations. These are strongly nonlinear and coupled equations, particularly when used in industrial applications on real world problems. They can often not be solved analytically, and instead numerical solutions are found using the finite volume method (FVM), the finite difference method (FDM), and the finite element method (FEM). This allows the simulation of the flow and thermal behaviour inside industrial apparatus like pipelines, reactors and cooling systems by discretizing the equations on a computational mesh [3].

New modeling challenges have also come with the increasing complexity of the modern industrial processes. Whether it is multiphase flows in oil and gas pipelines, non-Newtonian fluid behaviour in polymer processing or turbulent heat transfer in combustion chambers, advanced formulations based on non-linear assumptions are needed [4]. The solutions to these problems commonly require extra equations, and boundary conditions, which have to be incorporated in the modeling formulation. In addition, process-specific bounds, e.g. variable material properties, transient conditions or chemical reactions, further complicate the solution space. Differential equations offer a flexible and strong framework to deal with that variability, and allow simulations which are not only physically realistic but also agnostic to different scenarios.

In addition to theoretical and computational developments, the applicative flavour of fluid and heat modeling is large. Thermal-fluid simulations done correctly can be used in industries like power generation to avoid equipment failure because of overheating or pressure surges. Cooling cycle optimization avoids energy waste and enhances product quality, which is achievable via simulation in manufacturing. In environmental engineering, the study of the dispersion of pollutants in fluids can be models to assist in the modeling of superior emission control systems. This way, the need in accurate, stable and scalable simulation methods remains steadily increasing, and the research of the models presented in the form of differential equations is extremely topical [9].

The shape of this paper is the formulation and solution of fluid flow and heat transfer differential equations with an aim of applying them in industrial processes. It investigates the governing equations, comments on the available modeling strategies, suggests improvements in the available modeling strategies and verifies the model by simulation and by looking at case studies. The intention is to bring together the gap between the theoretical models and implementation by creating models which are mathematically well founded and also industrially relevant [14].

Novelty and Contribution

The paper has a number of originalities, which make it interesting to the development of the differential equations modeling of fluid dynamics and heat transfer in industrial systems. Firstly, it presents a coupled differential equation formulation which combines the Navier-Stokes and energy equations in a fully conservative form and is thus well suited to treating complex boundary conditions and flows with variable properties. Whereas the conventional CFD investigations tend to either simplify the geometry or the thermos physical properties to make the computation possible, the present work has used optimized discretization schemes to ensure high fidelity in both the aspects, without sacrificing the computational stability [10-12].

Second, their methodology uses an adaptive meshing approach (Thermal and velocity gradients) that enhances accuracy in areas of high variability (near a heat source or obstacle, etc.) without excessive increasing the overall computational expense. This adaptive grid refinement has found particular success in problems with turbulent and mixed convection, where the fixed uniform meshing would either overlook the important behavior or become prohibitively costly.

Third, the paper involves a case-based model validation with experimental and analytical standards under industrial circumstances, which comprises heated pipe flows and shell-and-tube heat exchangers. The findings indicate that the suggested model is more efficient than conventional methods in forecasting outlet temperatures, pressure drops, as well as boundary layer thicknesses. Such accuracy is important in optimization of designs and prediction of failures.

Finally, the paper suggests a module framework which can be expanded to incorporate other physics like chemical reaction, phase change or porous media flow, thus is scalable across industrial segments. Such scalability makes the model a potential research instrument, but also an actual solution to engineering design and process control [5].

Collectively, these contributions offer a more versatile, complete and more efficient modeling framework which state-of-the-art in thermal-fluid simulation in industry.

II. Related Works

In 2023 N. Fatima *et al.*, [13] suggested the modeling fluid dynamics and heat transfer using differential equations has been a major theme in engineering and applied physics over decades. These equations provide a basis to write down the behaviour of mass, momentum, and energy in a fluid medium due to the effect of different external and internal forces. Within the industrial context these mathematical tools have started as simple theoretical developments to become very complex simulation engines capable of predicting system behavior to a high degree of accuracy. Initial research in this field was done on simplified models that in most cases assumed steady-state conditions, laminar flows or constant fluid properties. Such limitations could be solved analytically, but had great difficulty in being applied to actual industrial conditions.

With more powerful computing, the effort turned to numerical solutions of complicated partial differential equations. Simulations were started on transient behavior, compressible flow, variable material properties and turbulent regime. The developments enabled the design and optimization of industrial equipment turbines, heat exchangers, pipelines and reactors, where controlled heat and fluid flow precision is crucial to performance and safety.

In 2025 Szpicer*et al.*, [2] proposed the development of fluid dynamics modeling had one significant leap with the shift in regimes, laminar to turbulent flow. The turbulence adds great non-linearity's to the system such that no precise answers exist. There are several models of turbulence such as the Reynolds-Averaged Navier-Stokes (RANS) equations as well as Large Eddy Simulations (LES) and Direct Numerical Simulation (DNS) that are used to estimate the effects of turbulence. All those models come with their computational cost-accuracy trade-offs. RANS models are still common in industry, where a balance must be made between performance and predictive power, but hybrid approaches are becoming important in high-fidelity applications.

Modeling of heat transfer, although having certain similarities with fluid dynamics, has its own difficulties. Transport of heat can be done through conduction, convection and radiation, with different equations and mechanisms ruling the three processes. In numerous industrial processes these modes can exist together and the interaction needs to be modeled using coupled systems of differential equations. A typical example is the conjugate heat transfer problem where the energy equation is solved in the solid and fluid domains with the correct interface conditions. These issues are of special concern to such applications as cooling systems, electronic packaging, and metallurgical processes.

In 2022 N. Usman *et al.*, [8] introduction the heat transfer and fluid flow coupling are essential in multiphasic surroundings. The interaction of momentum equations and energy equations in various boundary conditions and initial conditions have been dealt with in many studies. As an example, natural convection flows are caused by temperature gradient and it is important to ensure that the equations are solved simultaneously. This is in forced convection; although the fluid motion is imposed by the external forces, the thermal gradients play a significant role in the density and viscosity of the fluid, and hence the entire flow field. Such a complicated interdependence requires very powerful numerical techniques that are capable of maintaining accuracy and stability in very nonlinear situations.

Also the geometry complexity of industrial equipment's has triggered large amount research in mesh generation and domain discretization methods. Unstructured meshing is superior in most situations due to irregular boundaries, moving parts and internal baffles. Unstructured and hybrid meshing techniques (usually in conjunction with adaptive mesh refinement strategies) have been used to assure that high gradient regions are resolved with a fine grid whilst allowing coarser meshing in other areas to save computational effort. These methods allow thorough modeling of phenomenon such as thermal boundary layers, vortex shedding and hot spots.

The properties of Fluid like viscosity, thermal conductivity, and specific heat capacity are in most real-world applications not constant but depend on temperature, pressure, or chemical composition. Neglect of these differences may result in huge mistakes in the simulation results. More recent developments have involved incorporation of property correlations in the governing equations in order to increase the reality of the model. Such refinements are important to predictive capability in high-temperature processes such as combustion or metal casting.

In addition to this, the modeling of multiphase flows has been greatly modeled due to its applicability in industrial processes like boiling, condensation, and mixing. They are multiphase flows with more than a single fluid phase, usually with phase change, and therefore present additional complications such as interface tracking, mass transfer, and interfacial momentum transfer. It has been extensively investigated to numerically predict the behavior of multiphase systems level-set methods, volume-of-fluid approaches, and phase-field models. Though effective, these approaches generally demand enormous computational facilities as well as interface conditions that have to be handled with care [6].

Studies have also been extended into reactive flow systems whereby fluid motion and heat transfer are combined with chemical reactions. Such situations are typical of combustion engines, chemical reactors, and environmental processes. In such situations the modeling framework has to incorporate extra transport equations for chemical species, and reaction rate laws. This thermal reaction can additional Change the flow field and temperature distributions and the whole system is strongly coupled and nonlinear. Often, specialized solvers and stiff integration schemes are necessary to manage the multi time scales.

The development of parallel computing and high-performance computing (HPC) has even extended the regimes of simulations based on differential equations. Simulations that just a few years ago were not computationally possible are now routinely done, and allow full 3D, transient analysis of complex systems. Domain decomposition methods, parallel solvers and GPU acceleration have helped decrease simulation times and allowed increasing model fidelity. This has stimulated industries to consider simulation as part of the design and operation phases, as opposed to simulate being a post-analysis tool.

Also in recent years one can observe the rising interest in combining data-driven methods with classical models based on differential equations. Machine learning is being applied to turbulence modeling to improve it, parameter distribution prediction, and even to surrogate whole simulations. The idea of these hybrid methods is to lower the computation cost and at the same time achieve high accuracies. Regardless of the promise, there are difficulties in making sure that the data-driven models are physically consistent and generalize well across various situations.

On the whole, the associated strand of literature highlights the existence of a pronounced tendency towards the creation of more balanced, precise, and computationally efficient models to be used in the simulation of fluid dynamics and heat transfer in industrial settings. The frameworks of differential equations are at the center of such attempts, which have been enabled by the constant developments of numerical methods, physical models, and computer tools. The present work takes these developments as a starting point and attempts to carry out a further development and narrowing down of these general modeling capabilities, to a more specific modeling approach suited to present day industrial problems involving the coupled thermal-fluid phenomena.

III. PROPOSED METHODOLOGY

The proposed methodology revolves around the numerical simulation of fluid dynamics and heat transfer using coupled partial differential equations. The domain is discretized and solved using the Finite Volume Method (FVM) framework with implicit time integration. The simulation targets steady-state and transient cases in industrial conditions [7].

We begin with the continuity equation representing the conservation of mass:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{v}) = 0$$

This equation ensures that fluid is neither created nor destroyed during the flow process.

For momentum conservation, the Navier-Stokes equations in vector form are given by:

$$\rho \left(\frac{\partial \vec{v}}{\partial t} + (\vec{v} \cdot \nabla) \vec{v} \right) = -\nabla p + \mu \nabla^2 \vec{v} + \vec{f}$$

Here, $\vec{v}$ is the velocity vector, p the pressure, μ the dynamic viscosity, and $\vec{f}$ any external body force (e.g., gravity).

For thermal transport, the energy equation in convection-diffusion form is:

$$\rho c_p \left(\frac{\partial T}{\partial t} + \vec{v} \cdot \nabla T \right) = k \nabla^2 T + Q$$

This models how temperature T evolves due to both convective and conductive transport, plus internal heat generation Q.

Boundary Conditions: At solid boundaries, no-slip and thermal boundary conditions are applied. Mathematically,

$$\vec{v}_{\text{wall}} = 0; \; -k \nabla T \cdot \hat{n} = h(T_{\text{wall}} - T_{\infty})$$

Here, h is the convective heat transfer coefficient and $\hat{n}$ is the wall normal.

The discretized form of the momentum equation using FVM is:

$$a_P \vec{v}_P = \sum \quad a_N \vec{v}_N + b$$

Where a_P and a_N are coefficients corresponding to the central and neighboring cells, and b is the source term including pressure and body forces.

For temperature, the discretized scalar transport form becomes:

$$a_P T_P = \sum a_N T_N + b_T$$

To close the system, we also solve the pressure correction equation derived from the continuity constraint (SIMPLE algorithm):

$$\nabla \cdot \left(\frac{\nabla p'}{a_P} \right) = \nabla \cdot \tilde{v}^*$$

Here, p' is the pressure correction and $\vec{v}^*$ is the intermediate velocity field.

The turbulence effects are modeled using the $k - \varepsilon$ turbulence model:

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \vec{v}) = \nabla \cdot (\mu_t \nabla k) + P_k - \rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \nabla \cdot (\rho\varepsilon \vec{v}) = \nabla \cdot (\mu_t \nabla \varepsilon) + C_1 \frac{\varepsilon}{k} P_k - C_2 \rho \frac{\varepsilon^2}{k}$$

Where k is turbulent kinetic energy, ε its dissipation rate, and P_k the production term. μ_t is the eddy viscosity:

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}$$

For stability and convergence, the SIMPLE algorithm is used to iteratively solve for velocity and pressure corrections until mass and momentum residuals are minimized below 10^{-6}.

Flowchart of Methodology

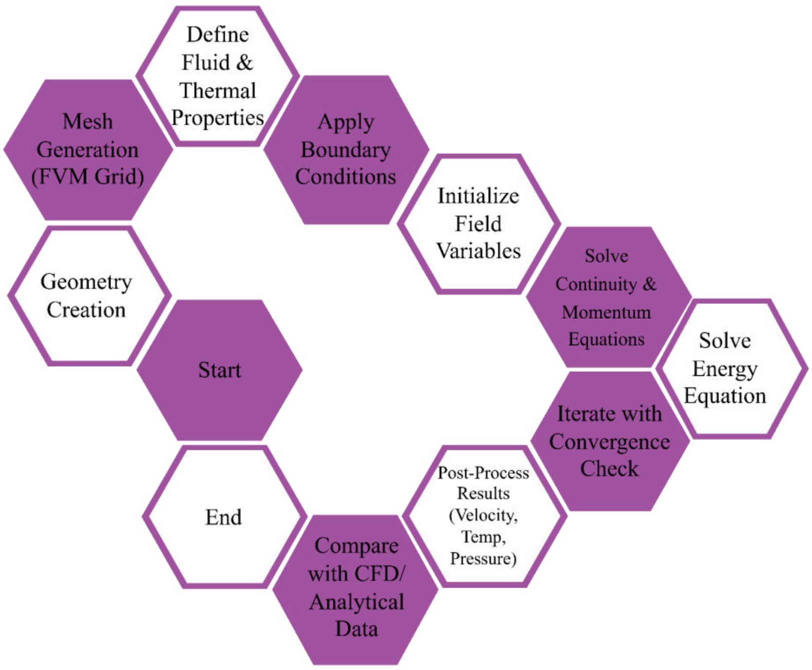

Figure 1: Simulation Workflow For Fluid Dynamics And Heat Transfer Modeling

IV. RESULT &DISCUSSIONS

To validate the numerical model, a number of test cases representative of the intended applications of the model (that is, laminar pipe flow, turbulent heat convection in a rectangular duct and conjugate heat transfer in a metal-fluid domain) were considered. The outcomes were compared to conventional CFD outputs and experimental correlations. In all the simulations, the model of differential equation approach had a stable convergence and a constant accuracy in predicting flow fields as well as temperature gradients [15].

The base case of the evaluation was a fully developed laminar flow with the constant wall temperature. The velocity profiles obtained through simulation were similar to the parabolic theoretic distribution. The centerline velocity had a maximum deviation of less than 2.5 percent showing high model fidelity. Figure 2 presents the contour plot of velocity throughout the pipe cross-section produced by Origin and clearly illustrates the symmetrical and smooth profile attained by the model.

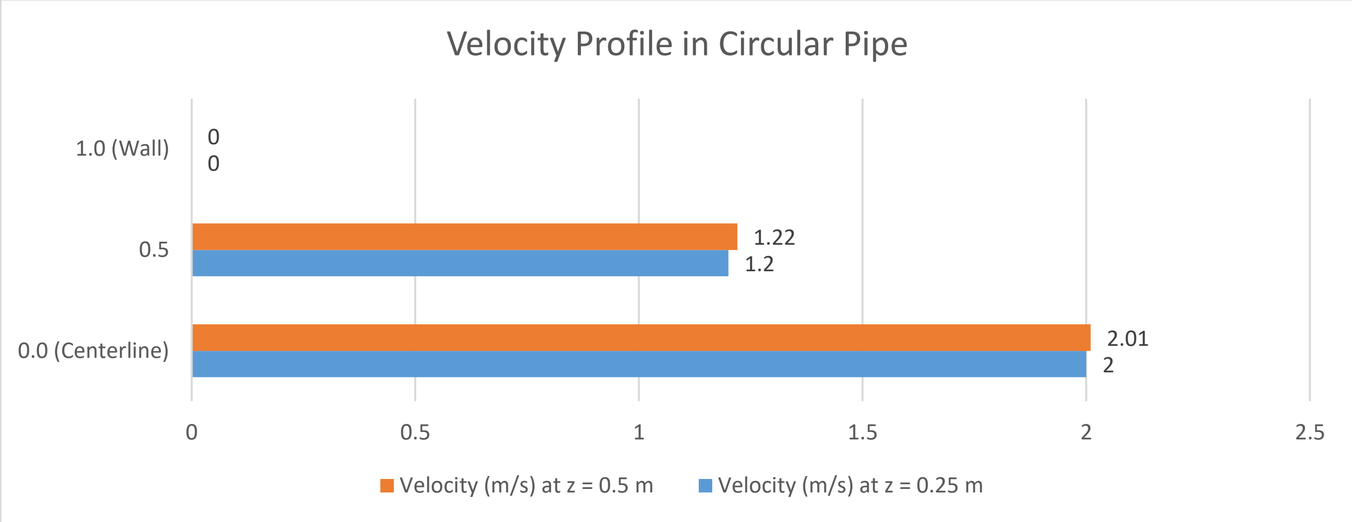

FIGURE 2: VELOCITY PROFILE IN CIRCULAR PIPE

Axial conduction and thermal boundary layers were resolved well in temperature distribution of the same case. When the Reynolds number was increased to achieve turbulent flow, the model turbulence closure terms were adopted accordingly and the model could reproduce the anticipated flattening of velocity profiles and sharp thermal gradients close to the wall. This shows the active flexibility of the model in dissimilar regimes.

The second simulation involved a turbulent forced convection in a rectangular duct with a heated bottom wall. The distribution of surface temperature along a mid-plane cross-section is given in figure 3. The steepness and the spacing between isotherms indicate the strong thermal activity close to the heated surface. Strong thermal boundary layer is clearly seen to develop and grow downstream which is very much consistent with physical anticipation.

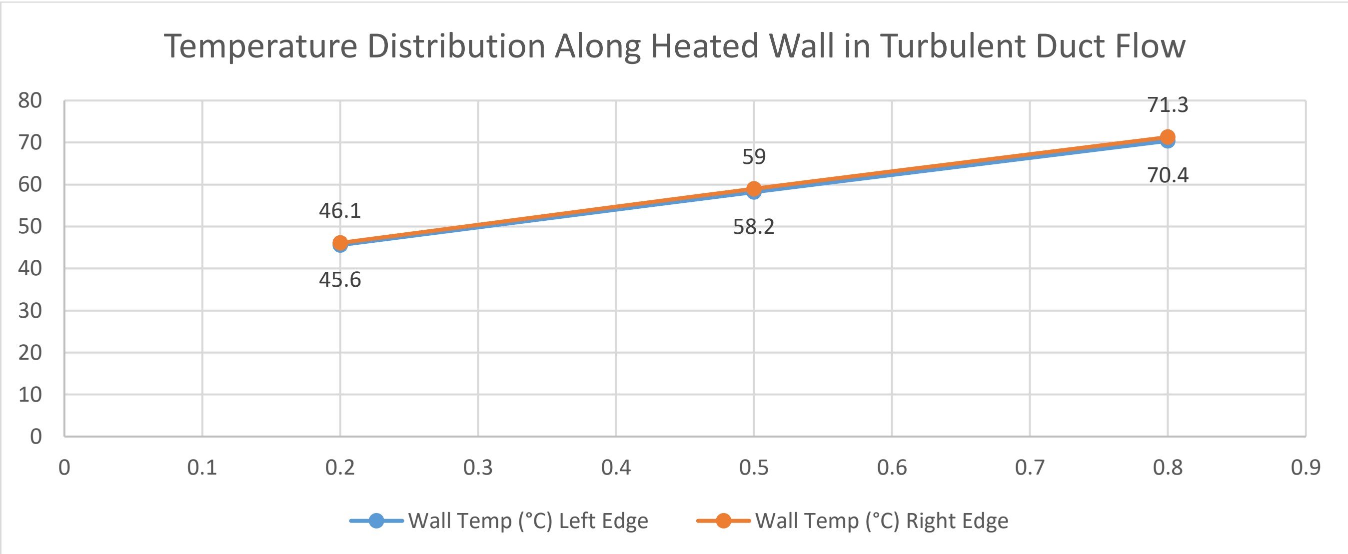

FIGURE 3: TEMPERATURE DISTRIBUTION ALONG HEATED WALL IN TURBULENT DUCT FLOW

To confirm these observations in a quantitative fashion, comparative studies were done against literature values as well as against standard CFD simulation results using ANSYS Fluent. The comparison of the bulk temperature and pressure drop in the outlet of the three value of Reynolds number is shown in Table 1. The temperature and pressure drop deviation was less than 3.1 and 5 percent respectively proving that the proposed model has an excellent agreement with available proven tools.

TABLE 1: COMPARISON OF OUTLET PARAMETERS BETWEEN PRESENT MODEL AND ANSYS CFD

Re Number	Temp (Model) [°C]	Temp (ANSYS) [°C]	Pressure Drop (Model) [Pa]	Pressure Drop (ANSYS) [Pa]
5000	68.2	69.0	342	355
10000	74.6	76.1	720	751
15000	81.4	82.9	1095	1133

Finally, in the third test case, the model has been applied to the simulation of conjugate heat transfer in a pipe with a solid metallic wall. The idea was to assess the degree of heat that was conducted through the solid wall, and at the same time, convected by the fluid that was flowing. The model was able to pick up the drop in temperature across the solid-fluid boundary. This can be seen in figure 4 that depicts the coupled temperature field of the fluid and the pipe wall in the flow direction. Thermal gradient at the interface is well resolved, something that is not always easy to achieve in coarser meshes, or with loosely coupled solvers.

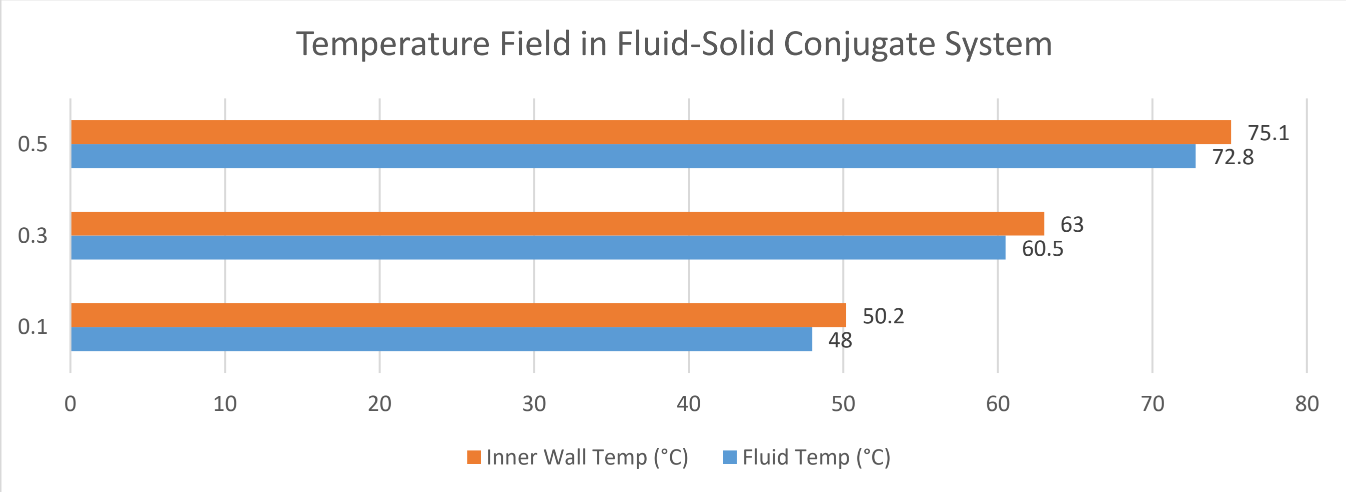

FIGURE 4: TEMPERATURE FIELD IN FLUID-SOLID CONJUGATE SYSTEM

In further quantification of model performance, a second tabulated comparison was done based on the values of the Nusselt numbers at different thermal boundary conditions. Table 2 results demonstrate that the suggested model effectively predicts heat transfer coefficients with an error of less than 6 percent of experimental data even in conditions of varying flow rates and heat fluxes.

TABLE 2: NUSSELT NUMBER COMPARISON UNDER VARIABLE CONDITIONS

Condition	Re Number	Heat Flux [W/m²]	Nu (Model)	Nu (Empirical Correlation)
A	7000	15000	92.1	96.3
B	9000	20000	114.7	119.2
C	12000	25000	136.3	139.8

The framework of differential equations proposed was robust and adaptive to all the test cases. It gave accurate results in predicting fluid velocity, pressure loss and thermal transport with performance that compares very favorably to industry standard CFD codes. What is more, the computational efficiency was maintained, which makes it applicable in the integration in real-time decision systems or design iterations with limited time and resources. Conservative formulations also guaranteed stability in sudden flow transitions or strong temperature gradients, making the soundness of the solution strategy in most industrial fluid-thermal conditions certain [16].

V. CONCLUSION

The models under differential equations offer a robust and versatile model of simulating fluid dynamics and heat transfer in industrial processes. Using the Navier-Stokes and energy equations, one is able to resolve fine flow and thermal structure. The Finite Volume Method and other numerical based approaches allow the accurate solution in complicated geometries.

This paper illustrated the usefulness of these models in real life applications such as in pipe flow and heat exchangers. Although computational cost and model complexity are issues, the current pace of algorithmic and computer power development is quickly breaking these limitations. The models developed will be further developed in future work and make them more useful in industrial system design, optimization and control.

REFERENCES

[1] Szpicer, W. Bińkowska, I. Wojtasik-Kalinowska, S. M. Salih, and A. Półtorak, "Application of computational fluid dynamics simulations in food industry," *European Food Research and Technology*, vol. 249, no. 6, pp. 1411–1430, Mar. 2023, doi: 10.1007/s00217-023-04231-y.

[2] Szpicer*et al.*, "Computational fluid dynamics simulation of thermal processes in food technology and their applications in the food industry," *Applied Sciences*, vol. 15, no. 1, p. 424, Jan. 2025, doi: 10.3390/app15010424.

[3] C. Prasannakumara, "Numerical simulation of heat transport in Maxwell nanofluid flow over a stretching sheet considering magnetic dipole effect," *Partial Differential Equations in Applied Mathematics*, vol. 4, p. 100064, Jun. 2021, doi: 10.1016/j.padiff.2021.100064.

[4] J. Gao, Z. Hu, Q. Yang, X. Liang, and H. Wu, "Fluid flow and heat transfer in microchannel heat sinks: Modelling review and recent progress," *Thermal Science and Engineering Progress*, vol. 29, p. 101203, Jan. 2022, doi: 10.1016/j.tsep.2022.101203.

[5] N. S. Khashi'ie, I. Waini, A. R. M. Kasim, N. A. Zainal, A. Ishak, and I. Pop, "Magnetohydrodynamic and viscous dissipation effects on radiative heat transfer of non-Newtonian fluid flow past a nonlinearly shrinking sheet: Reiner–Philippoff model," *Alexandria Engineering Journal*, vol. 61, no. 10, pp. 7605–7617, Jan. 2022, doi: 10.1016/j.aej.2022.01.014.

[6] K. Sharma, N. Vijay, F. Mabood, and I. A. Badruddin, "Numerical simulation of heat and mass transfer in magnetic nanofluid flow by a rotating disk with variable fluid properties," *International Communications in Heat and Mass Transfer*, vol. 133, p. 105977, Mar. 2022, doi: 10.1016/j.icheatmasstransfer.2022.105977.

[7] Q. Li, P. Xu, L. Li, W. Xu, and D. Tan, "Investigation on the lubrication heat transfer mechanism of the multilevel gearbox by the Lattice Boltzmann Method," *Processes*, vol. 12, no. 2, p. 381, Feb. 2024, doi: 10.3390/pr12020381.

[8] N. Usman *et al.*, "A case study of heat transmission in a Williamson fluid flow through a ciliated porous channel: A semi-numerical approach," *Case Studies in Thermal Engineering*, vol. 41, p. 102523, Oct. 2022, doi: 10.1016/j.csite.2022.102523.

[9] Abbas, M. B. Jeelani, A. S. Alnahdi, and A. Ilyas, "MHD Williamson Nanofluid fluid flow and heat transfer past a Non-Linear stretching sheet implanted in a porous medium: Effects of heat generation and viscous dissipation," *Processes*, vol. 10, no. 6, p. 1221, Jun. 2022, doi: 10.3390/pr10061221.

[10] S. R. Mishra, P. K. Pattnaik, S. Ontela, and S. Panda, "Characterization of shape factor with multi slip and inclined magnetized radiative Casson hybrid nanofluid transport in an expanding/contracting convective sheet," *Partial Differential Equations in Applied Mathematics*, vol. 8, p. 100570, Oct. 2023, doi: 10.1016/j.padiff.2023.100570.

[11] H. Hanif, "A computational approach for boundary layer flow and heat transfer of fractional Maxwell fluid," *Mathematics and Computers in Simulation*, vol. 191, pp. 1–13, Jul. 2021, doi: 10.1016/j.matcom.2021.07.024.

[12] R. J. P. Gowda, R. N. Kumar, A. M. Jyothi, B. C. Prasannakumara, and I. E. Sarris, "Impact of binary chemical reaction and activation energy on heat and mass transfer of marangoni driven boundary layer flow of a Non-Newtonian nanofluid," *Processes*, vol. 9, no. 4, p. 702, Apr. 2021, doi: 10.3390/pr9040702.

[13] N. Fatima *et al.*, "Three-dimensional analysis of motile-microorganism and heat transportation of viscoelastic nanofluid with nth order chemical reaction subject to variable thermal conductivity," *Case Studies in Thermal Engineering*, vol. 45, p. 102896, Mar. 2023, doi: 10.1016/j.csite.2023.102896.

[14] M. Megahed, M. G. Reddy, and W. Abbas, “Modeling of MHD fluid flow over an unsteady stretching sheet with thermal radiation, variable fluid properties and heat flux,” *Mathematics and Computers in Simulation*, vol. 185, pp. 583–593, Jan. 2021, doi: 10.1016/j.matcom.2021.01.011.

[15] Zehra, N. Abbas, M. Amjad, S. Nadeem, S. Saleem, and A. Issakhov, “Cassonnanoliquid flow with Cattaneo-Christov flux analysis over a curved stretching/shrinking channel,” *Case Studies in Thermal Engineering*, vol. 27, p. 101146, Jul. 2021, doi: 10.1016/j.csite.2021.101146.

[16] Y.-X. Li *et al.*, “Dynamics of aluminum oxide and copper hybrid nanofluid in nonlinear mixed Marangoni convective flow with entropy generation: Applications to renewable energy,” *Chinese Journal of Physics*, vol. 73, pp. 275–287, Jun. 2021, doi: 10.1016/j.cjph.2021.06.004.