Secure Systems Design and Technology Development. Veliko Tarnovo, Bulgaria
Proceedings of the International Scientific Conference, 2026. Volume II, 107-111

Numerical Modeling of Conjugate Heat Transfer Between a Fluid and a Solid Using ANSYS Fluent

Lyuba Gyurova
dept. Mechatronics
Nikola Vaptsarov Naval Academy
Varna, Bulgaria
l.gyurova@nvna.eu

***Abstract*—The present study develops a numerical model of conjugate heat transfer between a fluid and a solid, which is essential for the optimization of engineering systems such as heat exchangers, electronic cooling systems, and industrial reactors. The simulation is performed using ANSYS Fluent, where heat transfer in both domains and at their shared interface is analysed.**

The modelling procedure includes mesh generation, definition of physical models and material properties, as well as specification of boundary conditions for temperature and fluid velocity. The governing equations of mass, momentum, and energy conservation are solved, with automatic coupling of temperature and heat flux at the fluid–solid interface. The obtained results provide temperature and heat flux distributions, enabling quantitative evaluation of heat transfer efficiency and identification of critical thermal regions.

Keywords—ANSYS Fluent, CFD, conjugate heat transfer, fluid, heat transfer, solid.

I. INTRODUCTION

Conjugate heat transfer (CHT), defined as the simultaneous solution of heat conduction in solids and convective heat transfer in fluids, is a fundamental phenomenon in many engineering applications such as heat exchangers, electronic cooling systems, and thermal devices [1], [2].

Accurate numerical modelling of CHT enables prediction of global thermal performance as well as identification of localized regions with high temperature gradients, which may lead to system inefficiencies or failure [3], [4].

The literature demonstrates extensive application of CHT modelling in different engineering configurations. For example, nanofluid flow simulations in enclosed geometries show strong dependence of heat transfer on geometric parameters and material properties [1]. In thermal management of data centres, CHT models are used to analyse interactions between solid components and cooling fluids, enabling optimization of thermal design [2], [5].

Furthermore, studies on enhanced heat exchangers indicate that modified geometries and induced vertical structures significantly increase heat transfer performance [6], [7]. In modern CFD practice, solid domains are treated as active computational regions rather than simplified thermal boundaries, which leads to more accurate prediction of temperature fields and interface heat flux distribution [3], [4].

The present work develops a conjugate heat transfer model implemented in ANSYS Fluent for a simplified but representative configuration consisting of a solid block subjected to external fluid flow. The objective is to analyse temperature distribution and heat transfer characteristics and to identify thermally critical regions within the system.

II. MATERIALS AND METHODS

Geometry

The investigated configuration consists of a solid rectangular block with dimensions, $L=0.1\ m$, $W=0.05\ m$, $H=0.02\ m$, surrounded by a fluid domain (air) with dimensions $L_f=0.5\ m$, $W_f=0.25\ m$, $H_f=0.1\ m$ (Figure 1).

Online ISSN 3044-7771
https://doi.org/10.68302/std2026.vol2.80

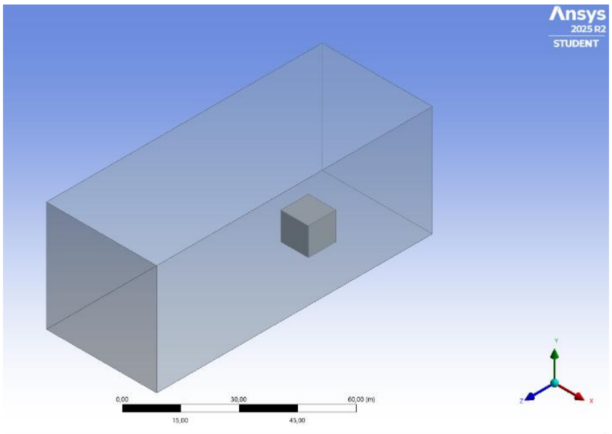

Fig. 1. Computational domain geometry

The air flows around the block from the inlet with velocity $U_{in}=1\ m/s$ and exits through the outlet and the lateral boundaries. The CAD model is used for visualization purposes, while the numerical simulation is performed using the actual physical dimensions of the computational domain.

Boundary Conditions

The following boundary conditions are applied:

- Inlet: uniform velocity $U=1\ m/s$, temperature $T=300\ K$. This condition defines both the incoming momentum and thermal energy of the fluid and governs the development of convective heat transfer along the solid surface;
- Outlet: gauge pressure – $p=0\ Pa$. This condition allows fully developed flow to exit the computational domain without artificial reflections and ensures numerical stability of the solution;
- Walls: All external solid boundaries are treated using a no-slip condition ($u=0$) which represents viscous adherence of the fluid to the wall. The outer surfaces of the computational domain are assumed adiabatic ($q=0$) meaning that no heat exchange occurs between the system and the surrounding environment. Thus, heat transfer is restricted only to interaction between the fluid and the solid body;
- Fluid-Solid Interface (Physical Description): The interaction between the fluid and solid domains is modelled using a conjugate heat transfer approach, ensuring thermal coupling at the interface. The interface treatment in ANSYS Fluent guarantees energy exchange between the two domains without thermal discontinuities [8].

Mathematical Model

The physical basis of conjugate heat transfer includes solving the following governing equations for the fluid and the solid domains:

- Continuity equation (mass conservation) [9], [10].

$$\nabla \cdot u = 0, \tag{1}$$

where:

u *[m/s]* – velocity vector.

- Navier-Stokes equations for incompressible Newtonian fluid (momentum) [9], [10].

$$\rho(u \cdot \nabla)u = -\nabla p + \mu\nabla^2 u, \tag{2}$$

where:

p *[Pa]* – pressure,

μ *[Pa·s]* – dynamic viscosity,

Body forces are neglected (F=0), assuming no buoyancy effects.

- Energy equation (fluid) [11], [12].

$$\rho c_p(u \cdot \nabla T) = \nabla \cdot \left(k_f \nabla T\right), \tag{3}$$

where:

ρ *[kg/m³]* – fluid density;

c_p *[J/(kg·K)]* – specific heat at constant pressure;

T *[K]* – absolute temperature;

k_f *[W/(m·K)]* – thermal conductivity of the fluid.

- Heat conduction (solid) [13].

$$\nabla \cdot (k_s \nabla T) = 0, \tag{4}$$

where:

k_s *[W/(m·K)]* – thermal conductivity of the solid.

Mesh Generation

The computational mesh for the conjugate heat transfer analysis was generated using ANSYS Meshing, employing a hybrid discretisation approach based on the MultiZone method for sweepable body regions, in accordance with the recommendations of the official software documentation [14].

The global element size was set to $2.87\times10^{-3}\ m$, with a maximum allowable size of $3\times10^{-3}\ m$, while the defeature size was defined as $1.435\times10^{-2}\ m$. Curvature-based refinement was applied with a minimum curvature size of 2.87×10^{-2} m and a curvature normal angle of *18°*, ensuring accurate geometric resolution in regions with changing surface curvature.

The computational domain bounding-box diagonal was *114.89 m*, with an average surface area of $1188.2\ m^2$. The minimum edge length was controlled to ensure sufficient geometric fidelity for thermal interface resolution.

Mesh quality was controlled by imposing a target skewness of *0.9*, while the transition ratio was limited to *0.272* to ensure smooth variation in element size and numerical stability during solution convergence.

Near the solid boundaries, an inflation mesh was applied consisting of up to five layers, with a growth rate of *1.2*, providing enhanced resolution of the thermal boundary layer and improved prediction of conjugate heat transfer across the fluid-solid interface.

Figure 2 presents the generated computational mesh, showing local refinement near the fluid-solid interaction zone, where steep temperature gradients are expected.

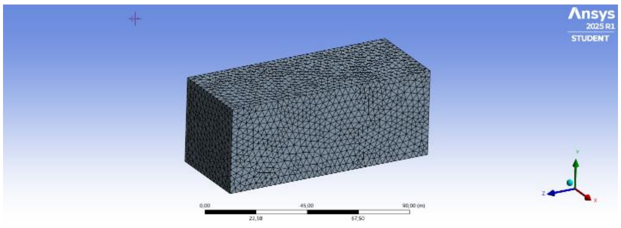

Fig. 2. Computational mesh structure

The final computational mesh contains *10298* nodes and *53335* elements, providing an appropriate balance between computational efficiency and numerical accuracy for conjugate heat transfer simulation.

The selected discretisation strategy ensured sufficient spatial resolution of temperature gradients in the interface region while maintaining computational efficiency for steady-state numerical analysis.

Numerical Procedure and Convergence Criteria

The numerical solution of the governing equations is performed in ANSYS Fluent using a steady-state solver, based on the finite volume method (FVM). Second-order discretization schemes are used, providing higher accuracy in the computation of temperature and velocity gradients. Pressure-velocity coupling is achieved using the SIMPLE algorithm.

Convergence of the solution is assessed by monitoring the scaled residuals, with the convergence criterion defined as a reduction below 10^{-6} for the energy equation and below 10^{-3} for the continuity and momentum equations. Additionally, the stabilization of temperature at characteristic points in the computational domain is monitored. The flow regime is assumed to be laminar, which is consistent with the characteristic Reynolds number of the configuration.

III. Results and Discussion

Temperature Distribution

After achieving numerical convergence, the temperature distribution in the fluid and solid domains was analyzed. Figure 3 presents the temperature field in the form of contour plots in a representative cross-section, illustrating the thermal interaction at the fluid-solid interface.

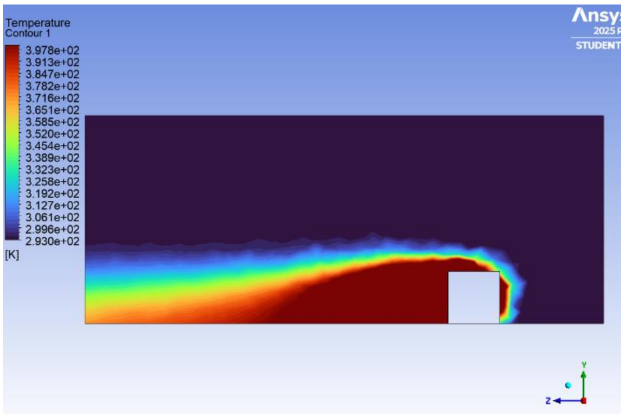

Fig. 3. Temperature contours in the fluid and solid domains in a characteristic cross-section

For a more illustrative representation of the three-dimensional nature of heat transfer, Figure 4 shows a volumetric rendering of the temperature field, providing a comprehensive view of the heat distribution within the computational domain.

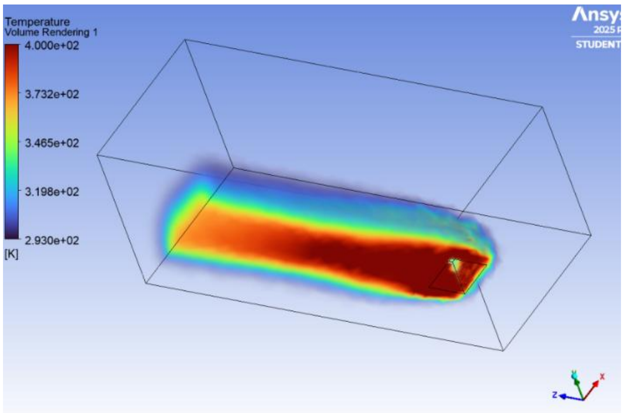

Fig. 4. Three-dimensional volumetric visualization of the temperature field in the computational domain

The three-dimensional temperature field confirms a temperature range between *293.0 K* and *397.8 K*. The highest temperature values are concentrated within the solid body and the immediate fluid-solid interaction region, while the surrounding fluid remains close to ambient temperature, indicating effective thermal dissipation through forced convection.

The maximum temperature values are observed in the central region of the solid block, reaching approximately *397.8 K*, while the minimum temperature in the surrounding fluid remained close to 293.0 K. This thermal difference of approximately 104.8 K results from limited conductive heat removal inside the solid and convective interaction with the external airflow.

In the fluid domain, the temperature gradually decreases from approximately 397.8 K near the heated solid interface to 293.0 K in the free-stream region, reflecting efficient convective heat transport. This behaviour is characteristic of conjugate heat transfer and reflects the balance between conduction in the solid and convective transport in the fluid.

The observed temperature field is consistent with the expected physical behaviour for external forced convection over a heated solid body and confirms the adequacy of the applied numerical model and boundary conditions.

Heat Transfer Intensity (Qualitative Analysis)

The intensity of heat transfer between the fluid and the solid is analyzed based on the obtained temperature field and local temperature gradients at the fluid-solid interface.

The highest heat transfer intensity is observed in the inlet region, where the temperature difference between the fluid and the solid surface is maximal and the thermal boundary layer is weakly developed. Along the length of the solid block, a gradual decrease in heat transfer intensity is observed as a result of thermal boundary layer development and the corresponding reduction in local temperature gradients at the interface.

This behavior confirms the physical consistency and numerical adequacy of the applied conjugate heat transfer model.

Engineering Analysis and Interpretation

The obtained results demonstrate that the conjugate modelling approach provides a more realistic representation of the thermal behaviour of the system compared to models in which the solid is treated as an isothermal or prescribed-temperature boundary.

The predicted thermal field spans approximately *104.8 K*, confirming strong thermal coupling between conductive heat transfer inside the solid and convective heat transport in the surrounding fluid.

The numerical results indicate stable thermal interaction at the fluid-solid interface, with clearly resolved temperature gradients across the contact region. The highest thermal gradients are concentrated in the upstream region, where the incoming fluid interacts with the heated solid surface and convective heat transfer is most intensive.

A gradual reduction in thermal intensity is observed downstream due to thermal boundary layer development, which reduces the local temperature gradient and consequently decreases the heat transfer rate along the solid surface.

The resolved temperature field clearly identifies thermally critical regions concentrated near the fluid-solid interaction zone, providing useful information for targeted optimisation of material properties, operating conditions, and system geometry.

These results demonstrate the applicability of conjugate heat transfer analysis as an effective tool for engineering thermal optimisation and design improvement.

Mesh Independence Study

A mesh independence study was performed to evaluate the sensitivity of the numerical solution to grid refinement, with particular focus on the fluid-solid interaction region where the steepest thermal gradients occur.

The variation of key output parameters, including average temperature and local thermal characteristics, remained below *2%* between successive refinement levels. This confirms grid-independent behaviour and demonstrates the numerical stability and reliability of the adopted discretisation strategy.

Validation of Results

The obtained results were qualitatively validated against classical heat transfer theory and existing literature on convective and conjugate heat transfer phenomena. The observed temperature distribution and heat transfer trends are consistent with established correlations for external forced convection, where higher heat transfer rates occur near the inlet region due to the development of a thin thermal boundary layer.

Furthermore, the results show good qualitative agreement with established numerical trends for conjugate heat transfer in comparable engineering configurations. This confirms the physical reliability of the numerical model and the correctness of the applied boundary conditions.

IV. Conclusion

The present study presents a numerical analysis of conjugate heat transfer between a fluid and a solid, implemented using ANSYS Fluent. The developed model successfully describes the interaction between convective heat transfer in the fluid domain and conductive heat transfer in the solid, ensuring continuity of temperature and heat flux at the fluid-solid interface.

The obtained results show clearly pronounced temperature gradients near the contact surface, as well as a characteristic development of the thermal boundary layer along the flow direction. It is observed that the maximum temperatures are concentrated inside the solid body, while in the fluid domain a gradual cooling occurs due to convective heat transfer.

The analysis of heat transfer intensity shows that the most effective heat transfer occurs in the inlet region of the fluid, while along the length of the body the efficiency decreases due to boundary layer development. These results are consistent with theoretical formulations and confirm the validity of the numerical model.

The developed approach demonstrates the applicability of conjugate heat transfer as a reliable tool for analysis and optimization of engineering systems. Future work may focus on turbulent flow regimes, different working fluids, and geometry optimization aimed at improving heat transfer performance.

References

[1] I. Zahan and M. A. Alim, “Effect of conjugate heat transfer on flow of nanofluid in a rectangular enclosure,” International Journal of Heat and Technology, vol. 36, no. 2, pp. 397-405, 2018, doi: https://doi.org/10.18280/ijht.360203.

[2] A. Dogan, S. Yilmaz, M. Kuzay, D.-J. Korpershoek, J. Burks, and E. Demirel, “Conjugate heat transfer modelling of a cold plate design for hybrid-cooled data centres,” Energies, vol. 16, no. 7, p. 3088, 2023, doi: https://doi.org/10.3390/en16073088.

[3] P. Renze and K. Akermann, “Simulation of conjugate heat transfer in thermal processes with open source CFD,” ChemEngineering, vol. 3, no. 2, p. 59, 2019, doi: https://doi.org/10.3390/chemengineering3020059.

[4] A. Alrashidi, “Numerical study of conjugate heat transfer for cooling the circuit board,” Journal of Electronics Cooling and Thermal Control, vol. 6, no. 3, pp. 88-97, 2016.

[5] S. Singh and K. Sørensen, “Numerical investigation of conjugate heat transfer and flow performance of a fin and tube heat exchanger with vortex generators,” in Proc. 58th SIMS Conf. Simulation and Modelling (SIMS 2017), Linköping Electronic Conference Proceedings, vol. 138, pp. 13-19, 2017, doi: https://doi.org/10.3384/ecp1713813.

[6] C. Hu, X. Feng, Z. Yuan, D. Shen, L. Zhang, and J. Zheng, “Numerical investigation of conjugate heat transfer in a heat exchanger equipped with cross-combined ellipsoidal dimple tubes,” Thermal Science, vol. 28, no. 2C, pp. 1849-1863, 2024, doi: https://doi.org/10.2298/TSCI230422226H.

[7] A. Dass and S. Gedupudi, “Fourier series based modeling of the dynamics of inclined closed loop buoyancy driven heat exchangers with conjugate effect,” arXiv preprint arXiv:2007.03878, 2020. [Online]. Available: https://arxiv.org/abs/2007.03878.

[8] ANSYS Inc., ANSYS Fluent User’s Guide, Release 2025 R2, Canonsburg, PA, USA, 2025.

[9] J. D. Anderson, Computational Fluid Dynamics: The Basics with Applications. New York, NY, USA: McGraw-Hill, 1995.

[10] H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed. Harlow, UK: Pearson, 2007.

[11] T. L. Bergman, A. S. Lavine, F. P. Incropera, and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, 7th ed. Hoboken, NJ, USA: Wiley, 2011.

[12] A. Bejan, Convection Heat Transfer, 4th ed. Hoboken, NJ, USA: Wiley, 2013.

[13] D. W. Hahn and M. N. Özisik, Heat Conduction, 3rd ed. Hoboken, NJ, USA: Wiley, 2012.

[14] ANSYS Inc., ANSYS Meshing User’s Guide, Release 2025 R1, Canonsburg, PA, USA, 2025.