Volume XXIX 2026
MBNA Publishing House Constanta 2026

Scientific Bulletin of Naval Academy

SBNA PAPER • OPEN ACCESS

Numerical Investigations of the Thermofluid Performance of a Solar Collector Tube used in Maritime Applications

To cite this article: Avram Elena-Rita, Cotorcea Alexandru, Kahraman Nafiz, Simion Dragoș-Ionuț, Pintilie Alexandru, Scientific Bulletin of Naval Academy, Vol. XXIX 2026, pg. 194-201.

Submitted: 17.05.2026
Revised: 15.08.2026
Accepted: 07.09.2026

Available online at www.anmb.ro

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-26-I1-018

Numerical Investigations of the Thermofluid Performance of a Solar Collector Tube used in Maritime Applications

Avram Elena-Rita[1]*, Cotorcea Alexandru[2], Kahraman Nafiz[3], Simion Dragoș-Ionuț[4], Pintilie Alexandru[5]

[1]"Mircea cel Bătrân" Naval Academy, Navigation and Naval Management Faculty, Naval Engineering and Port Management Department, email: rita.avram@anmb.ro
[2]"Mircea cel Bătrân" Naval Academy, Navigation and Naval Management Faculty, Naval Engineering and Port Management Department, email: alexandru.cotorcea@anmb.ro
[3] "Erciyes University, Faculty of Aeronautics and Astronautics, Astronautical Engineering Department, email: nafiz@erciyes.edu.tr
[4]"Mircea cel Bătrân" Naval Academy, Navigation and Naval Management Faculty, Naval Engineering and Port Management Department, email: dragos.simion@anmb.ro
[5]"Mircea cel Bătrân" Naval Academy, Interdisciplinary Research Center in the Maritime Field, email: alexandru.pintilie@anmb.ro
*rita.avram@anmb.ro

Abstract. The increasing demand for reducing fuel consumption and improving energy efficiency in maritime industry has intensified interest in solar thermal systems for auxiliary energy supply. In this study, a numerical model of a solar collector tube is developed in COMSOL Multiphysics. The main objective of this paper is to evaluate the thermal performance of a solar collector tube from the thermofluid behavior perspective. The device is working under conditions relevant to maritime applications. The model considered the water as the working fluid, and simulations are carried out for different working fluid velocities, as 0.005 m/s, 0.01 m/s, and 0.02 m/s, The heat flux applied to the tube wall is constant, at 800 W/m^2. The numerical results confirm the development of laminar and thermally developing flow conditions. The study demonstrates the significant influence of flow rate on temperature distribution, outlet temperature, thermal efficiency, and pressure losses. Lower flow rates result in higher temperature increases, whereas higher flow rates improve heat transfer intensity but lead to increased hydraulic resistance. The highest thermal efficiency was obtained at a fluid flow velocity of 0.02 m/s, as well as, the heat transfer coefficient of 71.7 W/m^{2}K. The results highlight the strong interaction between thermal and hydraulic performance. Also, these emphasize the importance of selecting appropriate operating conditions for efficient solar thermal system operation in maritime environments.
Keywords: solar collector tube, heat transfer, laminar flow, COMSOL Multiphysics, maritime applications.

1. Introduction

It is well known that, in the recent years, one of the major goals of maritime transport is decarbonization. Increasing environmental regulations and reducing fuel consumption and greenhouse gas emissions have become a necessity. The International Maritime Organization (IMO) has set ambitious targets for reducing greenhouse gas emissions from ships. In this context, the ship operators and port infrastructures have to adopt renewable energy-efficient and low-carbon technologies for sustainable maritime operations. Solar thermal systems used on board ships and within port facilities for providing auxiliary thermal energy represent a such technology (Marine Environment Protection Committee, 2023).

There are studies (Tay & Konovessis, 2023) that have highlighted the growing interest in solar-assisted technologies. It has been reported that the application of photovoltaic systems in maritime transport typically achieves a conversion efficiency of 15-20%. The installation costs of these are ranging from 2.60 to 3.20 USD/W.

Solar collector tubes are widely used to convert incident solar radiation into useful thermal energy for low and medium-temperature applications. These systems function as compact heat exchangers in which the absorbed solar energy is transferred to a circulating working fluid through coupled conductive and convective heat transfer mechanisms. Current research efforts are increasingly focused on improving the performance of solar collectors. The performance of solar thermal collectors strongly depends on the thermofluidic conditions of the working fluid. Parameters such as flow rate, inlet temperature and imposed heat flux directly influence the heat transfer process, but also the hydraulic behavior of the system. Lower flow rates increase the outlet fluid temperature due to longer residence times. Higher flow rates improve the heat extraction capacity, but also increase pressure losses and reduce the temperature rise. Currently, both nanofluids are used to improve the performance of solar collectors due to their increased heat transfer capacity and numerical modeling techniques. The results of the use of nanofluids and the improved heat transfer characteristics have been presented in specialized works (Sainz-Mañas et al., 2022), (Obalalu et al., 2023).

At the same time, thermofluidic modeling through simulation software, such as COMSOL Multiphysics, has become an essential tool for the analysis of solar thermal systems. These simulations allow the investigation of velocity distributions, temperature fields and pressure variations under controlled operating conditions, in order to optimize the performance of the system. In the specialized literature, thermal efficiencies of 60-70% are reported for flat plate and evacuated tube collectors (Sainz-Mañas et al., 2022). This is why they remain among the most reliable and technically suitable configurations for engineering applications. Simultaneously, efficiency improvements of 10-15% have been demonstrated in nanofluid-based direct absorption solar collectors (NDASC). Thus, theoretical experiments and numerical simulations carried out by Obalalu et al. have shown thermal performance improvements of 1.78 - 2.26%, demonstrating the potential of carbon-containing hybrid nanofluids to increase the efficiency of solar collectors in maritime applications (Obalalu et al., 2023). Furthermore, there are investigations that indicate limited economic feasibility, despite high environmental benefits (García Rodríguez et al., 2025). The same study demonstrates that providing a fire detection system is technically feasible using a single 550 Wp photovoltaic panel.

In addition, Lykas et al. performed an analysis of 63 representative studies on hybrid and multigeneration energy concepts integrating renewable sources and reported overall energy efficiencies ranging from 16% to 78%, depending on the system configuration. Also, some authors reported in the reviewed studies the improvement of the coefficient of performance by up to 15% for hybrid absorption cooling systems and reductions in electricity costs by up to 40% in residential energy management through predictive optimization (Powell & Mohammadi, 2021).

The implementation of solar collector tubes on board ships presents a number of challenges less significant in terrestrial applications. There are factors that can influence the amount of incident solar radiation, such ship motions, particularly roll and pitch. These can affect the efficiency of the collector. Also, the humidity, salinity and corrosive conditions, can accelerate the degradation of materials and reduce the durability of the system. Beside of these, the available deck area and structural integration further restrict the size and configuration of solar installations on board commercial ships (García Rodríguez et al., 2025; Paulson et al., 2024).

Although considerable progress has been made in the development of marine renewable energy systems, few investigations are reported that examine simplified collector geometries that allow a clearer interpretation of the influence of flow on operational performance. Many existing studies focus on either large-scale hybrid energy systems or complex experimental setups.

In this context, the present study investigates the thermofluidic behavior and operational performance of a heated tube. This tube is representative of a solar collector operating under conditions relevant to marine applications. A fluid model is developed in COMSOL Multiphysics.

Furthermore, the evaluation of the thermal performance the analysis focuses on the influence of the flow rate on the velocity distribution, temperature evolution, outlet temperature and pressure losses. The results are intended to support both the design and operational optimization of solar thermal systems for marine applications.

2. Materials and methods

2.1 Study area and physical model

The present study considers a simplified representation of a single straight cylindrical tube representative of the flow channels embedded within the absorber plate of a solar collector. Such configurations are frequently employed in maritime and port-related applications for supplying low-temperature thermal energy required for auxiliary onboard and shore-based services.

The numerical model was developed under steady-state conditions. It is assuming a horizontal tube subjected to uniform heating along its external surface. This simplified approach allows the investigation of the coupled heat transfer and fluid flow mechanisms while maintaining a computationally efficient framework suitable for parametric analysis.

2.2 Materials and software

A two-dimensional axisymmetric numerical model was developed in COMSOL Multiphysics 6.1. The aim is to investigate both the fluid flow and heat transfer processes occurring inside a solar collector tube representative of low-temperature solar thermal systems used in the maritime applications. The simulations were performed under steady-state conditions using a finite element approach implemented within the COMSOL Multiphysics environment.

The internal fluid domain and the surrounding solid tube wall are the computational domain. The geometry was defined by the length of 1 m for the tube, the inner radius of 0.004 m, and a wall thickness of 0.001 m. Due to the geometrical symmetry and uniform heating conditions, an axisymmetric formulation was made.

Because of its widespread use in solar thermal installations, the water was selected as the working fluid. Using the COMSOL material database, the temperature-dependent thermophysical properties of water, as density, dynamic viscosity, thermal conductivity, and specific heat capacity, were defined. The copper was used as the tube material. Heat conduction within the solid wall at the interaction between the wall and the flowing fluid was included in the model. The numerical model combined the incompressible Navier–Stokes equations for laminar flow with heat transfer in solids and fluids, and non-isothermal flow.

2.3 Mathematical model

The governing equations for fluid flow have been solved in axisymmetric cylindrical coordinates. Incompressible laminar flow is assumed. The continuity and momentum equations are expressed as:

$$\nabla \cdot \boldsymbol{u} = 0 \quad (1)$$

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

where,

ρ - fluid density $[kg/m^3]$,

u - velocity vector $[m/s]$,

p – pressure $[Pa]$,

μ - dynamic viscosity$[Pa \cdot s]$.

The heat transfer in the fluid domain was described using the energy equation (3), heat conduction in the solid tube wall was governed by equation (4).

$$\rho c_{\mathrm{p}}(\boldsymbol{u} \cdot \nabla T) = k \nabla^2 T \quad (3)$$

$$k_{\mathrm{s}} \nabla^2 T = 0 \quad (4)$$

where,

u - velocity field $[m/s]$,

p – pressure $[Pa]$,

T – temperature $[K]$,

ρ – density $[kg/m^3]$,

μ – dynamic viscosity $[Pa \cdot s]$,

c_{p}– specific heat at constant pressure $[J/(kgK)]$,

k – fluid thermal conductivity $[W/(mK]$,

ks – solid thermal conductivity $[W/(mK]$.

The heat transfer flow modeling was performed by introducing the convective term into the energy equation. Thermal continuity conditions were imposed at the fluid–solid interface. These conditions ensure equal temperature and heat flux on both sides of the wall, which allows for a correct description of the heat exchange between the fluid and the solid material.

2.4 Boundary conditions and numerical procedure

A constant heat flux of 800 W/m^2 was applied to the outer surface of the tube wall to simulate the thermal energy absorbed from solar radiation. The tube inlet temperature is 293 K. The tube inlet velocities are between 0.005 m/s and 0.02 m/s. At the outlet, a zero pressure condition was imposed, allowing the flow to develop naturally along the tube. No-slip boundary conditions were applied at the fluid-solid interface.

The operating conditions investigated corresponded to Reynolds numbers below the laminar transition threshold for flow inside the pipe. A mesh with local refinement was generated near the tube wall to accurately capture the velocity and thermal gradients that develop in the region near the wall. The governing equations were solved using the stationary finite element solver available in COMSOL Multiphysics, and numerical convergence was achieved by appropriate solver settings and mesh refinement.

Post-processing of the numerical results included the analysis of velocity profiles, temperature distributions, outlet temperature variation, and pressure drop along the tube. The useful thermal energy transferred to the fluid was evaluated using equation (5):

$$Q_{\mathrm{u}} = \dot{m}c_{\mathrm{p}}(T_{\mathrm{out}} - T_{\mathrm{in}}) \tag{5}$$

while the hydraulic performance of the system was assessed through the pressure drop:

$$\Delta p = p_{\mathrm{in}} - p_{\mathrm{out}} \tag{6}$$

These performance indicators were used to evaluate both heat transfer efficiency and pressure losses under different operating conditions in marine environments.

3. Results and discussions

To evaluate the thermofluid behavior of the solar collector tube, a set of steady-state simulations was performed for three inlet velocities: 0.005 m/s, 0.01 m/s, and 0.02 m/s, while all other parameters were kept constant. The results are presented in terms of velocity field, temperature distribution, outlet temperature, and pressure drop.

The velocity distribution obtained from the numerical simulations confirms the development of a laminar flow regime inside the tube. The results show a characteristic parabolic velocity profile, with maximum velocity along the axis of symmetry and zero velocity at the wall due to the no-slip boundary condition.

The simulations indicate that the flow becomes fully developed over a relatively short entrance region, approximately 1.6–6.4 cm from the entrance, depending on the imposed velocity. After which the velocity profile remains unchanged along the axial direction. This behavior is consistent for all investigated inlet velocities (0.005 m/s, 0.01 m/s, and 0.02 m/s), confirming that the flow remains within the laminar regime for the entire computational domain.

The preservation of the parabolic profile across all cases demonstrates that inertial effects remain moderate and that viscous forces dominate the flow behavior, as expected for low Reynolds number conditions.

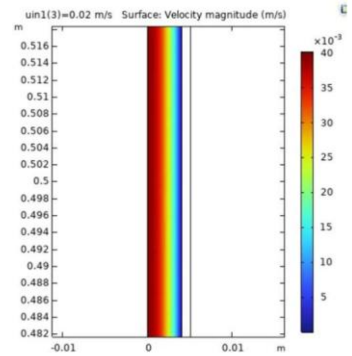

Figure 1. Velocity magnitude distribution inside the tube for laminar flow conditions.

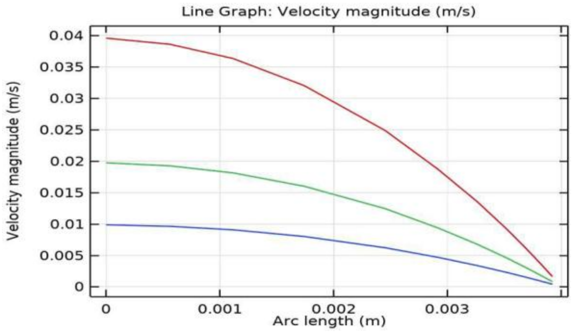

Figure 2. Radial velocity distribution inside the tube for different inlet velocities

For the three analyzed velocities, the thermal entrance length increases with the fluid velocity. At a velocity of 0.005 m/s, the temperature profile becomes fully developed after approximately 11.2 cm from the tube entrance. For a velocity of 0.01 m/s, the thermal entrance length increases to approximately 22.3 cm. At the maximum analyzed velocity of 0.02 m/s, complete thermal development occurs after approximately 44.7 cm. The results show that, as the fluid velocity increases, a longer distance is required to stabilize the temperature profile. For all cases, the thermal entrance length is less than the total tube length of 1 m. For the velocity of 0.02 m/s, the thermal entrance zone occupies almost half of the tube length, so the effects of the entrance region cannot be neglected.

The temperature field shows a gradual increase along the tube length due to the constant heat flux applied at the external wall. Heat transfer occurs through conduction in the solid wall and convection within the flowing fluid. The temperature contours indicate higher temperatures near the wall and lower values towards the tube centerline, reflecting the development of the thermal boundary layer and the continuous transfer of heat to the fluid.

The simulations also show that the outlet temperature decreases as the inlet velocity increases. For an inlet velocity of 0.005 m/s, the outlet temperature reached 318.5 K, while for 0.01 m/s and 0.02 m/s the outlet temperatures decreased to 310.2 K and 304.8 K, respectively. This behavior is associated with the shorter residence time of the fluid inside the heated tube at higher flow rates, which reduces the amount of heat absorbed during the flow process.

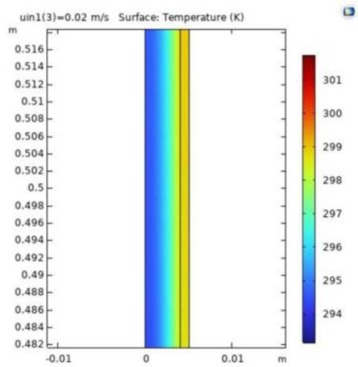

Figure 3. Temperature contours inside the tube under non-isothermal laminar flow conditions

The temperature field shows a gradual increase from the tube centerline towards the heated wall, reflecting conductive and convective heat transfer processes within the fluid and solid domains.

The pressure drop increased significantly with inlet velocity, rising from 12 Pa at 0.005 m/s to 45 Pa at 0.01 m/s and 170 Pa at 0.02 m/s. This behavior is characteristic of laminar flow conditions, where viscous effects dominate the hydraulic resistance. The pressure drop increased markedly with inlet velocity, indicating higher hydraulic resistance at larger flow rates.

The thermofluidic analysis shows that increasing the inlet velocity leads to lower outlet temperatures and significantly pressure losses. Lower flow velocities favor higher temperature rise and reduced hydraulic resistance, while higher velocities improve heat transfer intensity but increase pressure drop.

The Reynolds number was calculated for all simulated inlet velocities in order to verify the flow regime. The obtained values were approximately 40, 80, and 160 for inlet velocities of 0.005 m/s, 0.01 m/s, and 0.02 m/s, respectively. Since all Reynolds numbers remain well below the laminar transition limit of Re=2300, the simulations confirm the existence of a stable laminar flow regime, consistent with the parabolic velocity profiles obtained in COMSOL.

A theoretical validation of the pressure drop was also performed using the analytical expression for fully developed laminar flow in circular tubes. The comparison showed that the numerical pressure losses are higher than the theoretical predictions. For example, at an inlet velocity of 0.01 m/s, the theoretical pressure drop was approximately 5 Pa, while the numerical simulation yielded about 45 Pa. This discrepancy can be attributed to entrance effects, thermal gradients, and numerical modeling assumptions, which are not fully represented in the simplified analytical formulation.

Based on the numerical simulations, additional heat transfer performance indicators were evaluated. These include: the average fluid temperature, pressure drop, convective heat transfer coefficient, Nusselt number, and thermal efficiency. The average fluid temperature decreased from 305.4 K at an inlet velocity of 0.005 m/s to 299.64 K at 0.01 m/s and 296.69 K at 0.02 m/s, indicating a progressive reduction in fluid heating as the flow rate increased. At the same time, the effective pressure drop increased from approximately 12 Pa to 45 Pa and 170 Pa, confirming the stronger hydraulic resistance associated with higher velocities.

The convective heat transfer coefficient increased with inlet velocity, from 54.8 W/m²K at 0.005 m/s to 66.9 W/m²K at 0.01 m/s and 71.7 W/m²K at 0.02 m/s, reflecting the enhancement of convective heat transfer at higher flow rates. The evolution of the estimated Nusselt number suggests a gradual intensification of convective heat transfer as the inlet velocity increases. Although these values remain below the theoretical value of 4.36 corresponding to fully developed laminar flow under constant heat flux conditions, the results suggest that the thermal field remains partially developing along the tube length due to the relatively short geometry and the influence of axial thermal gradients.

The thermal efficiency also increased significantly with flow rate, reaching 31.2% at 0.005 m/s, 48.7% at 0.01 m/s, and 65.3% at 0.02 m/s. This behavior indicates that higher inlet velocities improve the overall heat transfer rate despite producing lower outlet temperature rise. Consequently, the results highlight the strong coupling between thermal and hydraulic performance and demonstrate that system efficiency cannot be evaluated solely based on outlet temperature.

The numerical simulations performed in COMSOL Multiphysics provided an accurate description of the fluid flow with heat transfer processes occurring inside the solar collector tube. The results confirmed that the flow remained laminar for all investigated inlet velocities, with Reynolds numbers ranging from approximately 40 to 160 and characteristic parabolic velocity distributions across the tube section.

The temperature field showed a progressive increase along the axial direction of the tube due to the imposed constant wall heat flux. The outlet temperature decreased from 318.5 K at an inlet velocity of 0.005 m/s to 304.8 K at 0.02 m/s, demonstrating the influence of residence time on thermal behavior. At the same time, the pressure drop increased significantly with velocity, reaching values from approximately 12 Pa to 170 Pa, indicating increased hydraulic resistance at higher flow rates.

The analysis of additional heat transfer indicators revealed that the convective heat transfer coefficient increased from 54.8 W/m²K to 71.7 W/m²K as the inlet velocity increased, while the estimated Nusselt numbers remained below the theoretical fully developed laminar value, suggesting the presence of a thermally developing flow regime inside the tube.

The thermal efficiency also increased with inlet velocity, from 31.2% to 65.3%, showing that higher flow rates improve the overall heat transfer rate despite reducing the outlet temperature rise. Overall, the simulations highlighted the existence of a clear trade-off between thermal performance and hydraulic losses, emphasizing the importance of selecting appropriate operating conditions for efficient solar thermal system operation in maritime applications.

Conclusions

This study provides a simplified numerical framework that allows for the direct evaluation of the influence of flow rate on heat transfer and pressure drop under controlled conditions. Numerical simulations confirmed that the inlet flow rate influences both thermal performance and pressure drop.

Lower flow rates produced higher outlet temperatures due to longer residence time. Higher flow rates improved heat transfer rate and thermal efficiency, but generated higher pressure drop.

The results demonstrated that the system operated under laminar flow conditions and with thermal development for all cases investigated. Reynolds numbers remained well below the laminar transition limit. Velocity distributions retained the characteristic parabolic profile associated with laminar flow. The analysis also highlighted a clear trade-off between temperature rise and hydraulic resistance, underlining the importance of selecting appropriate operating conditions for efficient system performance.

The main limitations of the study are related to the simplifying assumptions adopted in the numerical model, including steady-state operation, simplified tube geometry and uniform heat flux conditions. Also, environmental effects specific to maritime applications, such as transient solar radiation and ship motion, were not taken into account. The proposed numerical approach can serve as a useful preliminary design and optimization tool for solar thermal systems for ships and port facilities. The results of the study contribute to improving energy efficiency.

Acknowledgements
You must ensure that anyone named in the acknowledgments agrees to being named.

Author contributions: for research article with many authors a short paragraph specifying their contributions should be provided.
Examples:

- Conceptualization: A.E.R., C.A., K.N., S.D.I., P.A.
- Methodology: A.E.R.
- Writing original draft: A.E.R.
- Writing review and editing: S.D.I., P.A.
- Supervision: C.A., K.N.

Funding: This research received no external funding.

Conflict of interest: The Authors declare no Conflict of interest.

References

1. Gaber, M., El-banna, S. H., Hamad, M. S., & El-Dabah, M. (2021). An intelligent energy management system for ship hybrid power system based on renewable energy resources. Journal of Al-Azhar University Engineering Sector, 16(60), pp.712–723.
2. García Rodríguez, L., Castro-Santos, L., & Lamas Galdo, M. I. (2025). Feasibility and limitations of solar energy integration in merchant ships: A case study on fire detection systems. Journal of Marine Science and Engineering, 13(5), pp. 991. https://doi.org/10.3390/jmse13050991
3. Guo, X., Lang, X., Yuan, Y., Tong, L., Shen, B., Long, T., & Mao, W. (2024). Energy management system for hybrid ship: Status and perspectives. Ocean Engineering, 310 (Part 1), 118638. https://doi.org/10.1016/j.oceaneng.2024.118638
4. Inal, O. B., Charpentier, J.-F., & Deniz, C. (2022). Hybrid power and propulsion systems for ships: Current status and future challenges. Renewable and Sustainable Energy Reviews, 156, 111965. https://doi.org/10.1016/j.rser.2021.111965
5. Lykas, P., Georgousis, N., Bellos, E., & Tzivanidis, C. (2023). A comprehensive review of solar-driven multigeneration systems with hydrogen production. International Journal of Hydrogen Energy, 48(2), pp. 437–477. https://doi.org/10.1016/j.ijhydene.2022.09.226
6. Mushtaq, R., Iqbal, M., Khaliq, A., & Iqbal, J. (2025). Optimal design of a hybrid ship energy management system under various sea conditions using model predictive control. PLOS ONE, 20(7), e0326969. https://doi.org/10.1371/journal.pone.0326969
7. Obalalu, A. M., Memon, M. A., Olayemi, O. A., et al. (2023). Enhancing heat transfer in solar-powered ships: A study on hybrid nanofluids with carbon nanotubes and their application in parabolic trough solar collectors with electromagnetic controls. Scientific Reports, 13, pp. 9476. https://doi.org/10.1038/s41598-023-36716-x

8. Paulson, M, Babu, S., Chacko, M., (2024). Leveraging an installed standalone photovoltaic system for eco-friendly shipping. Engineering Review, 44, pp. 83–98. https://doi.org/10.30765/er.2318
9. Powell, K. M., & Mohammadi, K. (2021). Modeling, control, and optimization of multi-generation and hybrid energy systems. Processes, 9(7), pp. 1125. https://doi.org/10.3390/pr9071125
10. Sainz-Mañas, M., Bataille, F., Caliot, C., Vossier, A., & Flamant, G. (2022). Direct absorption nanofluid-based solar collectors for low and medium temperatures: A review. Energy, 260, 124916. https://doi.org/10.1016/j.energy.2022.124916
11. Santos, C. H., Baptista, P., & Neto, R. C. (2025). Decarbonising the maritime sector: A multi-perspective techno-economic analysis. Journal of Physics: Conference Series, 3195, 012008. https://doi.org/10.1088/1742-6596/3195/1/012008
12. Tay, Z. Y., & Konovessis, D. (2023). Sustainable energy propulsion system for sea transport to achieve United Nations sustainable development goals: A review. Discover Sustainability, 4, pp. 20. https://doi.org/10.1007/s43621-023-00132-y
13. The Marine Environment Protection Committee. (2023, July 7). 2023 IMO strategy on reduction of GHG emissions from ships
14. Wang, X., Zhu, H., Luo, X., Chang, S., & Guan, X. (2024). An energy dispatch optimization for hybrid power ship system based on improved genetic algorithm. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 238(2). https://doi.org/10.1177/09576509231205342
15. Zhao, L., Sun, C., & Lin, H. (2025). A review of hybrid energy management systems for ships. International Journal of e-Navigation and Maritime Economy, 24, pp. 57–71.