Jurnal Riset Rumpun Ilmu Teknik
Volume 5, Nomor 2, Agustus 2026
e-ISSN: 2829-016X; p-ISSN: 2829-0178, Hal. 141-152
DOI: https://doi.org/10.55606/jurritek.v5i2.8602
Tersedia: https://prin.or.id/index.php/JURRITEK

The Effect of Thermal Conditions on Airflow and Heat Transfer in a Test Object: An Experimental Study

Hilmala Nurmualimah[1*], Nur Rohmat[2], Alvian Harris Gita Purnama[3]
[1-3] Program Studi Teknik Mesin, Universitas Pamulang, Indonesia
Email: hilmalanurmualimah1@gmail.com [1], dosen000597@unpam.ac.id [2], alvianharris4@gmail.com [3]*

**Penulis Korespondensi: hilmalanurmualimah1@gmail.com [1]*

***Abstract**. This research aims to analyze the influence of thermal conditions; specifically the temperature difference between the test object and the environment; on the characteristics of air flow and heat transfer around it. The object of this study is a test piece subjected to free air flow under various temperature conditions; focusing on the convection heat transfer phenomenon. The main problem addressed is how temperature variations affect the convection heat transfer coefficient; heat transfer rate; and heat flux; as well as changes in air velocity and pressure profiles. Therefore; the objective of this research is to quantitatively compare and assess these thermal and fluid parameters through an experimental study approach and Computational Fluid Dynamics (CFD) simulation. The methodology involves direct measurement of temperature and pressure parameters under low and high-temperature conditions; which are then processed to determine the convection coefficient (h); heat transfer rate (q); and heat flux (q''). The main findings indicate that at low-temperature conditions; the heat transfer coefficient (h) was found to be 53.26 W/m^2K; the heat transfer rate (q) was 24.99 W; and the heat flux (q'') was 537.87 W/m^2K; with a pressure drop of 0.86 Pa. In conclusion; thermal conditions play a crucial role in determining the dynamics of air flow and the efficiency of heat transfer; the greater the temperature difference (ΔT); the higher the potential heat transfer rate; establishing a strong correlation between thermal conditions and the convection phenomenon.*

***Keywords**: Air Flow; CFD; Convection; Heat Transfer; Thermal Conditions.*

1. INTRODUCTION

Energy recovery from waste into productive energy input constitutes a vital aspect of industrial sustainability. This study specifically analyzes the optimization of waste air from exhaust systems by regulating the flow; with the aim of enhancing its utilization. Furthermore; production efficiency encompasses another critical element: heat transfer; which is closely related to effective airflow control. By analyzing the airflow generated by the exhaust fan; this airflow can also be leveraged to drive a wind turbine; owing to its high velocity and consistency (Kusuma Wardhana et al., 2021).

To optimize the airflow; a wind tunnel is constructed with the goal of focusing the air stream. Given the dispersive characteristic of airflow; efforts are made to create a channel that concentrates the flow onto a single point to achieve optimal airflow (Aji Saputra et al., 2021). Research conducted by Chen et al. (2021) and Ikram et al. (2021) emphasizes the presence of both stable and unstable flow rates when coupled with heat transfer. Consequently; there remains a necessity for in-depth research regarding how airflow characteristics change when a temperature variation occurs.

Artikel Masuk: 06 Januari 2026; Revisi: 10 Februari 2026; Diterima: 07 Maret 2026; Tersedia: 09 Maret 2026

Therefore; this research aims to address the issue of heat transfer control by analyzing the impact of temperature on airflow characteristics through the utilization of exhaust waste energy. By leveraging this previously wasted energy; the overall efficiency of the production process is increased.

2. LITERATURE REVIEW

Theoretical Foundations of Fluid Mechanics and Convection Heat Transfer

Fluid Mechanics is an applied branch of science that studies the behavior of fluids in both static and dynamic conditions (Mukhlisin et al.; 2022). The flow analysis in this research is based on the classification of Laminar and Turbulent flow; where the primary determining criterion is the Reynolds Number (Re). (Liu et al.; 2022)

$$\mathrm{Re} = \frac{\rho v \infty C}{\mu} \tag{1}$$

The flow investigated in this study is in the turbulent regime (Re>5000); which is a complex flow event characterized by fluctuations in velocity; pressure; and multi-scale vortices (Cafiero et al.; 2021; Asim et al.; 2023). Heat transfer in fluids primarily occurs through the convection mechanism. The analysis performed focuses on the forced convection process. This forced convection is driven by an external source which facilitates the energy transfer process. The calculation source for this convective heat transfer is Newton's Law of Cooling:

$$q = hA\,(T_s - T_\infty) \tag{2}$$

In the heat transfer efficiency process; the Nusselt Number (Nu) is typically employed for measurement. The Nusselt Number is a dimensionless quantity that describes the ratio of convective heat transfer to fluid conductive heat transfer. (Cheilytko.A.; et al 2024)

$$\mathrm{Nu} = \frac{h.L_c}{k} \tag{3}$$

The Reynolds Number (Re) and the Nusselt Number (Nu) share a close correlation; as an increase in the Reynolds Number directly affects or leads to an increase in the Nusselt Number (Han et al.; 2025). Beyond the Nusselt Number; other critical parameters for analyzing thermal characteristics and mixing scales are the Prandtl Number (Pr) and the Peclet Number (Pe). (Petropoulos et al.; 2024; Lawson; 2021)

Fractal Geometry as a Turbulence Enhancer and Research Gap

In heat transfer engineering; structural strategies to promote fluid mixing and turbulence intensity are highly needed. Fractal Grids have been proven as a state-of-the-art approach superior to conventional grids. The self-similar geometry of the fractal grid can generate stronger turbulence and vortices at various (multi-scale) levels simultaneously; which is proven to enhance convective heat transfer. (Lee et al.; 2021) For instance; a study by Lee et al. (2021) confirmed that the use of a fractal grid significantly increases flow fluctuations compared to a regular grid; making it a promising solution for heat dissipation applications. Although the potential for turbulence generation by fractal geometry has been widely validated; most previous research has focused on flow characterization under isothermal conditions (cold flow/without heat load).

Research Gap: Investigation into the effectiveness of heat transfer on Fractal Geometry under non-isothermal conditions (with heat load) and the specific role of the generated turbulence in increasing the convection coefficient remains limited. A critical analysis is needed to understand whether the dominant turbulent flow generated by the fractal structure directly correlates with increased thermal efficiency; or whether a decoupling phenomenon occurs between the increase in turbulence and the heat transfer itself. This research aims to fill this gap by experimentally analyzing (and supported by CFD simulation) the Impact of Thermal Conditions on a fractal geometry object; specifically by testing the phenomenon of thermal invariance that occurs when forced convection is applied.

Computational Fluid Dynamics (CFD) as an Analytical Tool

Computational Fluid Dynamics (CFD) is an essential numerical analysis tool for modeling and visualizing complex fluid dynamics and intricate heat transfer phenomena (Li et al.; 2025) In this context; CFD is utilized to evaluate flow characteristics around the fractal geometry; including pressure distribution; turbulence intensity; and vortex patterns that are difficult to measure precisely through experiments alone (Mahon et al.; 2022). The use of CFD also facilitates the analysis of thermal boundary conditions and the validation of experimental results to gain a deeper mechanistic understanding.

3. PROPOSED METHOD

This research employs a Research Methodology involving two complementary main approaches: (1) Experimental Study using a wind tunnel for field data acquisition; and (2) Numerical Analysis using Computational Fluid Dynamics (CFD) for model visualization and validation.

Algorithm

Subsection

This research process involves two distinct phases: an experimental phase in the wind tunnel and a numerical phase using CFD. The sequence of structured steps is presented as follows:

INPUTS: Fractal Geometry (Test Object); Free Stream Velocity (V); Heater Power (Pheater). OUTPUTS: Velocity Distribution; Temperature Difference (ΔT); Nusselt Number (Nu).

Step-by-Step Procedure:

a. Design and Preparation of the Data Acquisition System: Prepare the measuring instruments (K-Type Thermocouples; Anemometer) and assemble the Data Acquisition System (DAQ) based on the Arduino Uno Microcontroller. The system is configured for real-time recording of temperature (T_s) and flow velocity (T_∞).

b. Experimental Test Configuration: Install the Fractal Geometry within the Wind Tunnel Test Section. Install Thermocouples at designated points for measuring the surface temperature (T_s) and free stream temperature (T_∞).

c. Isothermal Flow Characterization: Conduct initial data collection for air velocity and pressure drop (ΔP) under no thermal load condition ($\mathrm{Pheater} = 0$) to verify that the flow is operating in the target turbulent flow regime.

d. Non-Isothermal Thermal Testing: Collect velocity and temperature data across three variations of thermal load conditions to measure the temperature differential (ΔT): a. Isothermal/Ambient Condition (Room Temperature). b. Medium Heat Condition (Medium Power setting). c. High Heat Condition (Maximum Power setting).

e. Numerical Simulation (CFD): Perform analysis using CFD software; applying a suitable Turbulence Model (e.g.; k-epsilon or k-omega) and Boundary Conditions that accurately replicate the experimental conditions (including variations in temperature/heater power).

f. Data Verification and Confirmation (Repeatability): Repeat the experimental data collection runs and confirm the simulation results to ensure repeatability and minimize random error.

g. Data Processing and Analysis: Calculate the dimensionless thermofluid parameters (Reynolds Number (Re); Convection Heat Transfer Coefficient (h); and Nusselt Number (Nu) from the experimental data. Compare the numerical results with the experimental results for Model Validation.

h. Conclusion Drawing: Analyze the processed data; particularly the phenomenon of Thermal Invariance (if observed); and draw conclusions based on the fulfilment of the research objectives.

i. Result Dissemination: Prepare and submit the scientific article for publication.

Fractal geometry spesification

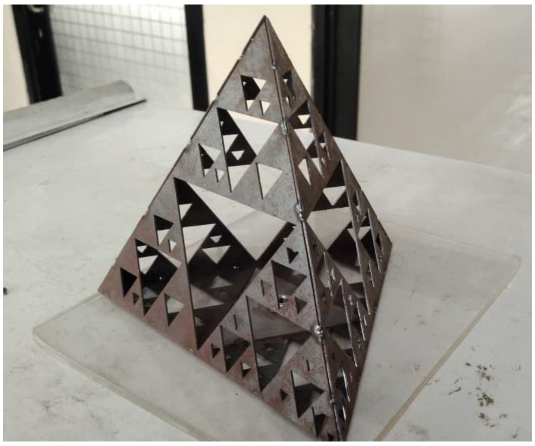


Figure 1. Fractal Geometry.

a. Material conductor using an iron plate

b. Total Surface Area: A total surface area of 464.61

c. Dimensions of the Base Triangle: The base length of the fundamental triangle is 16 and the side length is 14.

Mathematical Component Formatting

In this section; we present the definitions of the various thermodynamics and fluid parameters used in this analysis.

Temperature Difference (ΔT)

The equation used to calculate the Temperature Difference is:

$\Delta \mathrm{T} = \mathrm{T}_{\mathrm{out}} - \mathrm{T}_{\mathrm{in}}$ (1)

Eq. (1) is fundamental in determining the transfer of energy to or from a fluid.

Convective Heat Transfer Rate (q)

To calculate the heat transfer rate; the computation utilizes the formula derived from Newton's Law of Cooling; which is defined as follows:

$\mathrm{q} = \mathrm{h}.\mathrm{A}.(\mathrm{T}_{\mathrm{s}} - \mathrm{T}_{\infty})$(2)

Equation (2) is utilized when calculating the convective heat transfer rate between a solid surface and a fluid.

Heat Flux (q'')

Heat Flux is understood as the heat transfer rate divided by the unit area; which is calculated by dividing the heat transfer originating from Equation (2) by the surface area.

Reynolds Number (Re)

The Reynolds Number (Re) is a Dimensionless number that predicts the flow pattern of a fluid; whether specifically the flow is Laminar or Turbulent.

Nusselt Number (Nu)

The Nusselt Number (Nu) is defined as a dimensionless quantity that represents the enhancement of heat transfer (due to Convection relative to pure Conduction). The Nusselt Number can be calculated using the following formula:

$NuL = 0;037\mathrm{Re}^{0;6}\mathrm{Pr}^{1/3}$(3)

Eq. (3) is valid for specific turbulent flow conditions; and it allows for the calculation of the convective heat transfer coefficient (h).

Pressure (Pa)

Pressure (P) It is measured in units of Pascal (Pa); which is defined as force per unit area (N/m2).

4. RESULTS AND DISCUSSION

This section presents the findings from the testing conducted in accordance with the methodology outlined in Section 3. These results encompass the measured data for temperature and air velocity; as well as the analysis and calculation of heat transfer performance and dimensionless numbers. Subsequently; these results will be discussed and analyzed in relation to the research hypothesis and objectives.

General Description of the Research

Figure 2. Wind Tunnel.

The objective of this research is to analyse the characteristics inherent in a fractal test object placed within the test section of a wind tunnel. The core investigation focuses on characterizing air flow dynamics and heat transfer performance across this object. This analysis is conducted using a combination of experimental methodology and numerical simulation via Computational Fluid Dynamics (CFD) software. The primary focus of this study is to perform a comparative analysis of the fluid flow behaviour and temperature distribution between a non-heated condition (isothermal/ambient) and a heated condition provided by a cartridge heater.

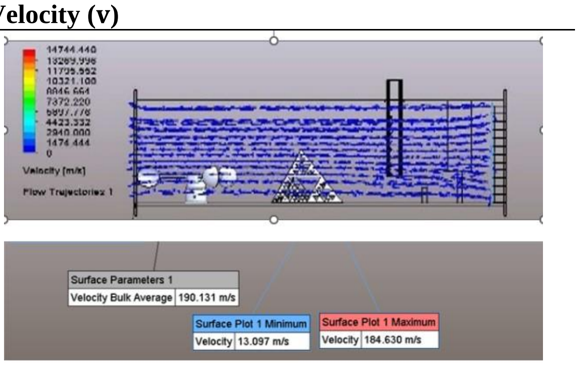
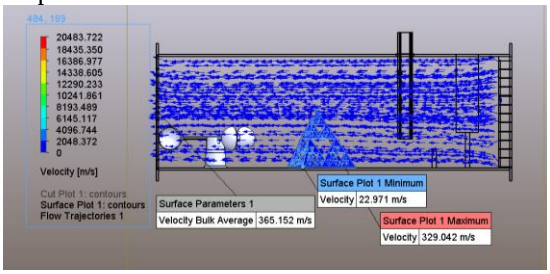
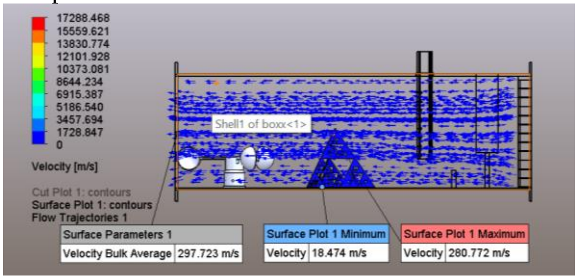
Figure 3. Test Section.

Figures and Tables

Computational Fluid Dynamics (CFD) Simulation Results

The following presents the visual data (images) and discussion derived from the computational fluid dynamics (CFD) simulation results within the test section.

Table 1. Velocity In CFD Simulation.

Comparison Aspect	Velocity (v)
Room Temperature	 Surface Parameters 1 Velocity Bulk Average 190.131 m/s Surface Plot 1 Minimum Velocity 13.097 m/s Surface Plot 1 Maximum Velocity 184.630 m/s **Figure 4.** CFD Velocity Simulation At Room Temperature
Medium Temperature (**50**℃)	 Surface Parameters 1 Velocity Bulk Average 365.152 m/s Surface Plot 1 Minimum Velocity 22.971 m/s Surface Plot 1 Maximum Velocity 329.042 m/s **Figure 5.** CFD Velocity Simulation At Room Temperature
High Temperature (**70**℃)	 Surface Parameters 1 Velocity Bulk Average 297.723 m/s Surface Plot 1 Minimum Velocity 18.474 m/s Surface Plot 1 Maximum Velocity 280.772 m/s **Figure 6.** CFD Simulation Of Velocity At High Temperature

In every CFD simulation process conducted under different applied temperatures; the fluid flow characteristics remained largely consistent. This is evidenced by the flow convergence (density) observed at the sharp edges of the test object; which signifies a significant increase in flow velocity and the immediate onset of high turbulence in those regions.

Table 2. Cfd Simulation Calculation Results.

Point of Comparison	Room Temperature	Medium Temperature 50℃	High Temperature 70℃
Temperature Difference (ΔT)	3; 43℃	10; 1℃	7; 99℃
Heat Transfer (q)	5; 74W	24; 99W	11; 57W
Heat Flux (q")	123; 54W/m^2	537;87W/m^2	249; 03W/m^2
Reynolds Number (Re)	1; 39x10^6 $\frac{\text{kg}}{\text{m.s}}$ Re > 5.000	2.337.551 $\frac{\text{kg}}{\text{m.s}}$ Re > 5.000	1.726.314 $\frac{\text{kg}}{\text{m.s}}$ Re > 5.000
Nusselt Number (Nu)	160; 34	218; 54	123;55
Pressure (P)	0; 48Pa	0; 86Pa	0; 27Pa
Velocity (V)	190;131m/s	365;152m/s	297;723m/s

In addition to the data obtained from the CFD simulation process; quantitative data was acquired through direct measurement using physical instruments. This measurement was performed directly using the K-Type thermocouple and the anemometer. The following data was obtained:

Table 3. Calculation Results From Experiments.

Point of comparison	Room Temperature	Medium Temperature 50℃	High Temperature 70℃
Temperature Difference (ΔT)	1; 5℃	1; 75℃	1; 75℃
Heat Transfer (q)	0;18699W	0;12265704W	0;305481075
Heat Flux (q")	4;024W/m^2	2;64W/m^2	6;575 W/m^2
Reynolds Number (Re)	13.391;54 Re > 5.000	16.629;9 Re > 5.000	16.484;9 Re > 5.000
Nusselt Number (Nu)	9;86	11;224	11;15
Pressure (P)	2;07936N/m^2	3; 2400368N/m^2	2; 885625N/m^2
Velocity (V)	1;9m/s	2;38m/s	2;25m/s

Comparative Analysis of Flow Characteristics and Thermal Impact

Comparison of Experimental and Numerical Flow Visualization

The following section presents the comparative data derived from the direct experimental observation (using smoke media) and the numerical simulation using CFD.

The research findings demonstrate that the implementation of the Sierpinski fractal geometry (featuring a repeating triangular configuration) on the test object significantly influences the fluid flow characteristics; primarily marked by a pronounced increase in turbulence intensity.

This phenomenon can be mechanistically explained based on the principles of fractal geometry. Fractal geometries are characterized by the property of a high surface area within a limited volume and a repeating structure across multiple scales. When the fluid flows past the test object arranged with this fractal pattern; the fluid is forced to navigate through a complex; tortuous path. The combination of these factors results in localized acceleration of the fluid flow. As explained in fluid mechanics theory; when flow passes through sections with complex or multiplied cross-sectional areas; the fluid flow velocity increases. Visually; the Sierpinski fractal geometry acts as a static mixer; constantly breaking up the boundary layer and regenerating fresh; high-intensity turbulence.

Effect of Heating: Heat Transfer Efficiency and Temperature Distribution

Increased Heat Transfer Efficiency The testing results indicate that the fractal geometry enhances the heat transfer efficiency from the test object to the fluid. This improvement is a direct consequence of the high turbulence induced by the fractal structure; which promotes vigorous mixing between the hot surface layer and the cooler bulk fluid.

Temperature Profile Analysis A crucial finding in this analysis is the significant temperature change observed in the fluid as it passes the test object. The fluid experiences a temperature rise upon interaction with the fractal surface; followed by a subsequent temperature drop after traversing the test object zone. This demonstrates the effectiveness of heat transfer concentrated in the fractal area.

Limitation of Temperature Increase in the Test Section (Thermal Invariance) Although heat transfer efficiency increased; the testing also recorded that the actual temperature in the test section did not deviate significantly from the ambient temperature; even when medium or high power was applied. This phenomenon is attributed to the material's ability to maintain its thermal properties (thermal mass) and the massive heat transfer capacity of the fractal structure. The massive heat transfer from the test object to the fluid meant that the surface temperature did not have time to surge (thermal inertia); because the supplied thermal energy was immediately channeled to the moving fluid. Consequently; the surface temperature tended to retain the environmental temperature; indicating a state of thermal invariance under forced convection.

5. CONCLUSION

In conclusion; the investigation into the influence of thermal conditions on air flow and heat transfer around a fractal geometry test object yields two primary findings:

Consistent Flow Regime:

The air flow around the fractal geometry test object is consistently dominated by the turbulent regime under both isothermal and non-isothermal conditions. Under isothermal conditions; the fractal geometry effectively creates high local velocity concentrations at the apex points of the structure; confirming its function as a turbulence enhancer.

Thermal Invariance and Low Efficiency:

Under non-isothermal conditions with forced convection; the system exhibited a strong thermal invariance phenomenon. Consequently; the measured temperature increase was minimal (ΔTremained very low). This low ΔT renders the convective heat transfer process ineffective in terms of thermal gradient; although the consistent turbulent dominance implies a high inherent potential for heat dissipation that warrants future investigation under different thermal boundary conditions.

DAFTAR REFERENSI

Agustin, D. E., Wawan, & Heryadi, Y. (2022). Analisa peredam panas glasswool pada alat pembakar sampah (insinerator) portabel 2 in 1. *Jurnal Teknologika*, *12*(1), 1–12.

Ansyah, P. R., & Ramadhan, M. N. (2018). *Termodinamika teknik I*.

Asim, S. B., Noviani, E., & Helmi, H. (2023). Pemodelan aliran fluida bidang miring pada lapisan tipis. *Jurnal Matematika Integratif*, *19*(1), 127–136. https://doi.org/10.24198/jmi.v19.n1.40855.125-136

Cafiero, G., Castrillo, G., & Astarita, T. (2021). Turbulence properties in jets with fractal grid turbulence. *Journal of Fluid Mechanics*, *915*, 1–21. https://doi.org/10.1017/jfm.2021.46

Diantoro, I., Darmanto, A., & Tim Mesin Pertahanan Akademi Militer Magelang. (2022). Pengaruh lokasi ketebalan maksimum airfoil simetrikal terhadap coefficient drag dan coefficient lift pada main rotor helikopter Bell 412. *Jurnal Mekanikasista*, *10*(1), 36–48. https://ojs.akmil.ac.id/index.php/jurnalmekanikasista/article/view/23

Efendi, R., Darwin, D., Badia, B. A., Tando, A., Herlina, H., & Padang, W. L. (2023). Rancang bangun termokopel data logger berbasis Arduino Mega 2560 skala laboratorium. *Machine: Jurnal Teknik Mesin*, *9*(2), 15–19. https://doi.org/10.33019/jm.v9i2.3803

Fatiatun, F., Pratiwi, A. D., Wirdati, A. C., & Avifatun, N. (2022). Penerapan termodinamika heating dan cooling pada dispenser. *Jurnal Penelitian dan Pengabdian kepada Masyarakat UNSIQ*, *9*(2), 146–150. https://doi.org/10.32699/ppkm.v9i2.2658

Firman, M., Hartadi, B., Arifin, J., Irfansyah, M., & Program Studi Teknik Mesin. (2024). Perancangan terowongan angin rangkaian terbuka dengan sistem PIV (particle image velocimetry). *Journal of Energy, Environment, and Mechanical Engineering Management*, *8*(1), 57–63. https://doi.org/10.30588/jeemm.v8i1.1781

Fredrick, M., Telaumbanua, E., & Zebua, R. K. (2024). Analisis aplikasi mekanika fluida dalam industri penerbangan. *Identik: Jurnal Ilmiah Teknologi Informasi*, *1*(2), 63–69. https://doi.org/10.70134/identik.v1i2.93

Han, T., Li, Q., Shang, L., Chen, X., Zhou, F., & Li, W. (2025). Study on the influence of Reynolds number on heat exchange performance and Nusselt number of spray coil heat exchanger. *Processes*, *13*(2), 1–16. https://doi.org/10.3390/pr13020588

Junaidin, B., Kusumaningrum, A., Prayogih, W., & Reo, Y. (2022). Rancang bangun sensor kecepatan angin berbasis Arduino untuk terowongan angin low subsonic. *AVITEC, 4*(2), 187–194. https://doi.org/10.28989/avitec.v4i2.1295

Lawson, J. M. (2021). Mass transfer to freely suspended particles at high Péclet number. *Journal of Fluid Mechanics, 913*, 1–25. https://doi.org/10.1017/jfm.2020.1177

Lee, T. S., Ooi, E. H., Chang, W. S., & Foo, J. J. (2021). Fractal grid-induced turbulence strength characterization via piezoelectric thin-film flapping velocimetry. *Scientific Reports, 11*(1), 1–19. https://doi.org/10.1038/s41598-021-02680-7

Li, D., Sang, Y., Lv, Z., Wu, K., & Lai, Z. (2025). Dynamic response analysis of wind turbine tower with high aspect ratio: Wind tunnel tests and CFD simulation. *Thin-Walled Structures, 211*, 113113. https://doi.org/10.1016/j.tws.2025.113113

Liu, Q., Miao, W., Bashir, M., Xu, Z., Yu, N., Luo, S., & Li, C. (2022). Aerodynamic and aeroacoustic performance assessment of a vertical axis wind turbine by synergistic effect of blowing and suction. *Energy Conversion and Management, 271*, 116289. https://doi.org/10.1016/j.enconman.2022.116289

Mahon, H., Friedrich, D., & Hughes, B. (2022). Wind tunnel test and numerical study of a multi-sided wind tower with horizontal heat pipes. *Energy, 260*, 125118. https://doi.org/10.1016/j.energy.2022.125118

Mufidah, I. Z., Trisnowati, E., Salsabila, K. M., Muniroh, C., Risqi, F. D. M., & Kurniawan, R. (2023). Analisis hukum termodinamika pada pembuatan batu bata di Magelang, Jawa Tengah. *Jurnal Pendidikan MIPA, 13*(3), 784–789. https://doi.org/10.37630/jpm.v13i3.1053

Mukhlisin, A., Erwin, E., & Wiyono, S. (2022). Rancang bangun smoke generator pada kecepatan angin rendah dengan wind tunnel rangkaian terbuka. *Jurnal Asiimetrik: Jurnal Ilmiah Rekayasa dan Inovasi, 4*(1), 81–88. https://doi.org/10.35814/asiimetrik.v4i1.2944

Niulai, J., & Muskitta, N. D. (2022). Pengaruh bentuk benda uji terhadap pola aliran angin di ruang uji wind tunnel. *Journal of Vocational Engineering and Education, 7*(1), 37–46. https://doi.org/10.32531/jvoe.v7i1.486

Petropoulos, N., Couchman, M. M. P., Mashayek, A., de Bruyn Kops, S. M., & Caulfield, C. C. P. (2024). Prandtl number effects on extreme mixing events in forced stratified turbulence. *Journal of Fluid Mechanics, 983*, 1–14. https://doi.org/10.1017/jfm.2024.110

Ratna, O., Titisari, D., & Hamzah, T. (2021). Kalibrator suhu dengan thermocouple dilengkapi dengan tampilan grafik. *Jurnal Teknokes, 14*(1), 28–35. https://doi.org/10.35882/teknokes.v14i1.5

Santi, D., Yuliati, S., Meidinariasty, A., & Program Studi Teknologi Kimia Industri Politeknik Negeri Sriwijaya. (2023). Analisis laju perpindahan panas pada alat tray dryer tenaga surya dalam proses pengeringan ikan asin. *Jurnal Pendidikan Tambusai, 7*(3), 21537–21543.

Trisnowati, E., Putri, D. R., Annisa Qurrota, S. S., Nikmah, F. K., & Mulyaningrum, D. (2023). Analisis konsep termodinamika pada produksi kerupuk sebagai bentuk kearifan lokal di Magelang, Jawa Tengah. *Jurnal Pendidikan MIPA, 13*(1), 268–273. https://doi.org/10.37630/jpm.v13i1.795

Zaki, M. F. G., Ghofur, A., & Studi Teknik, P. (2020). Analisa efektivitas heat transfer pada PLTU dengan metode ASME 4.0 dan 4.3. *Jurnal Rotary Teknik Mesin, 2*(1). https://doi.org/10.20527/jtam_rotary.v2i1.2003