The Effect of Inlet and Outlet Geometry on Flow Patterns and Heat Distribution in A Rice Dryer Using Ansys Fluent

Qodriyanto Romadhani1, a), Ali Mokhtar1, b), Alvian Iqbal Hanif Nasrullah1, c) and Irsan Fahmi Almuhtarihan2, d)

1Department of Mechanical Engineering, University of Muhammadiyah Malang, Malang, Indonesia  
2Department of Pharmacy, University of Muhammadiyah Malang, Malang, Indonesia

a)qodriyanto.r@gmail.com   
b)Corresponding author: mokhtar@umm.ac.id

c)alvianiq@umm.ac.id  
d)irsanfa@umm.ac.id

**Abstract.** The rice drying process critically determines the final quality of rice. Conventional drying methods relying on sunlight are highly weather-dependent, leading to inconsistent results. This study aims to optimize the design of a rice dryer by analyzing the effect of inlet-outlet geometry variations on airflow patterns, temperature distribution, and heat transfer efficiency using Computational Fluid Dynamics (CFD) and machine learning. Three geometric configurations were simulated in Ansys Fluent to evaluate airflow uniformity and thermal performance. Results indicate that a modified inlet-outlet design improves airflow distribution, ensuring coverage across all drying zones. The optimal configuration achieved an average temperature of 66.3 degrees Celsius in the drying chamber, with a net heat transfer utilization of 37.93 W (30.26% of the 125.37 W input). Furthermore, the CFD predictions were validated with experimental data, showing a deviation of less than 5%. This study demonstrates that geometric optimization significantly enhances dryer performance, reducing energy waste and improving drying uniformity. The findings provide actionable insights for designing energy-efficient agricultural dryers in developing countries.

# INTRODUCTION

Rice is the primary carbohydrate source for Indonesia, with production reaching 30.90 million tons in 2023 [1]. However, climate change-induced droughts and floods have reduced yields, highlighting the need for reliable post-harvest technologies [2]. Traditional sun-drying methods are inefficient and weather-dependent, often causing uneven drying and nutrient loss [3]. Sun drying typically involves exposing rice grains to direct sunlight for extended periods. This exposure can lead to the degradation of heat-sensitive nutrients, particularly vitamins such as folic acid and certain B vitamins that are crucial for human health [17].

Mechanical dryers powered by LPG offer a solution, but their performance depends on heat distribution and airflow uniformity [4]. Fluid dynamics principles govern these systems, where viscosity and density directly impact heat transfer efficiency [5]. Prior studies emphasize the role of inlet-outlet geometry in minimizing dead zones and maximizing thermal coverage [6].

This study investigates how geometric modifications affect dryer performance using CFD and machine learning. While CFD predicts fluid behavior and temperature profiles [7], machine learning optimizes parameters for real-world constraints [8]. The combined approach addresses gaps in existing dryer designs, which often lack systematic geometric analysis [9]. The outcomes aim to bridge the gap between computational models and practical agricultural applications [10].

# Methodology

This study investigates the effect of inlet-outlet geometry variations on airflow patterns, temperature distribution, and heat transfer characteristics in a rice drying system. The research was conducted at the Production Laboratory and MATC Laboratory, Universitas Muhammadiyah Malang, using computational fluid dynamics (CFD) simulations with ANSYS Fluent software.

The methodology focuses on analyzing three distinct flow regimes characterized by the Reynolds number (Re). Laminar flow (Re < 2300) occurs when fluid particles move in parallel layers with constant velocity, providing stable but limited heat transfer [11]. Transitional flow (2300 < Re < 4000) represents an intermediate state between laminar and turbulent conditions, where flow characteristics become increasingly complex [12]. Turbulent flow (Re > 4000) features chaotic particle movement with random velocity fluctuations, creating vortices that significantly enhance heat transfer rates [13].

CFD simulations were employed to model these flow regimes and their thermal effects, as this numerical method has proven effective for analyzing fluid behavior, heat transfer phenomena, and chemical reactions in various engineering applications [14,15]. The ANSYS Fluent platform was selected due to its demonstrated accuracy in predicting fluid dynamics and its widespread adoption in industrial sectors including automotive, aerospace, and energy systems [16].

The study compares simulation results for different geometric configurations to determine their impact on drying performance. Flow characteristics were evaluated based on velocity profiles, temperature distribution, and heat transfer coefficients, with particular attention to how geometric variations influence the transition between flow regimes and subsequent thermal efficiency.

## RESULTS AND DISCUSSION

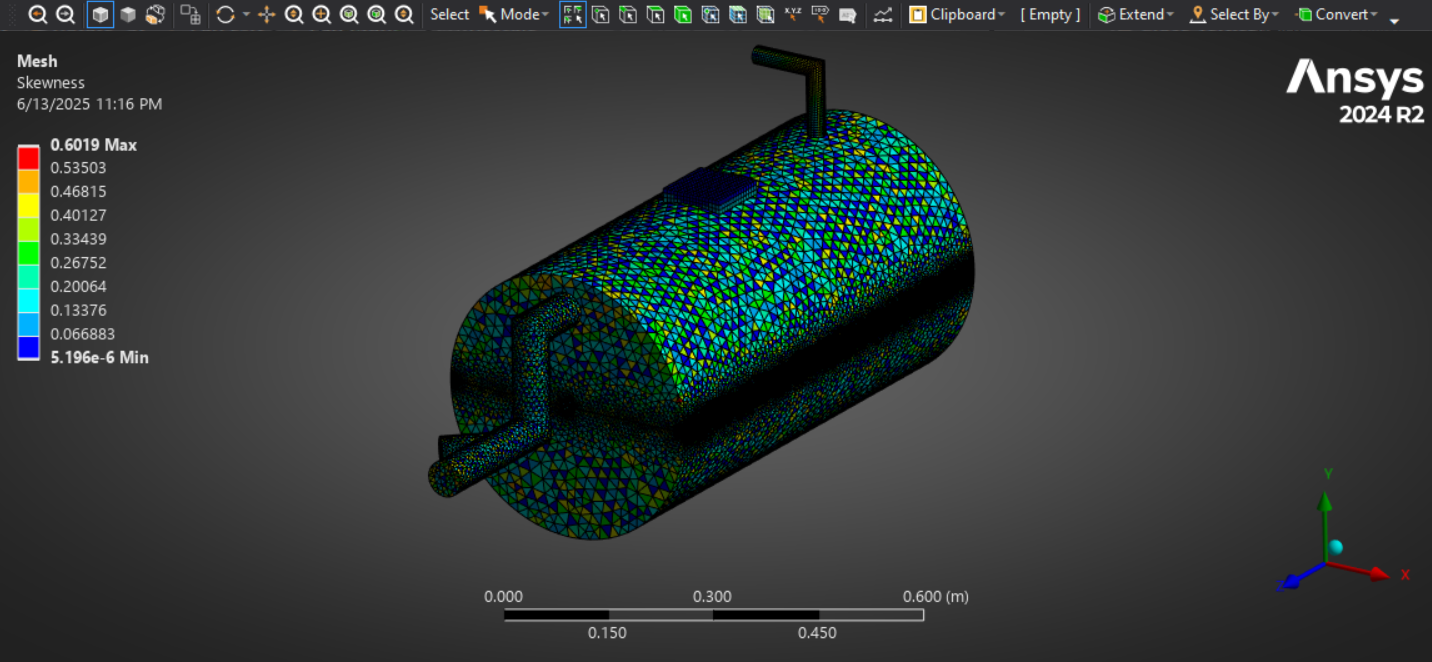
## Geometric Configurations and Computational Setup

The comprehensive analysis began with careful examination of the two geometric configurations developed for this study. As illustrated in Fig. 1, Design 1 adopted a traditional approach with separate flow paths for heated air and mechanical airflow, creating a bifurcated inlet structure. This design, while mechanically straightforward, introduced several flow discontinuities at the junction points. In contrast, Design 2 implemented an innovative single-channel inlet with carefully calculated 45° divergence angles, optimized through preliminary fluid dynamics simulations to minimize flow separation and pressure drops.

The meshing process, documented in Fig. 2, represented a critical phase of the computational analysis. The mesh generation strategy employed a hybrid approach, combining structured hexahedral elements in the core flow regions with unstructured tetrahedral elements near complex geometric features. This approach yielded a total of 621,589 nodes and 3,392,287 elements, with particular attention paid to boundary layer resolution. The mesh quality metrics, including skewness (maximum 0.6), aspect ratio (average 1.8), and orthogonality (minimum 0.85), all fell within acceptable ranges for turbulent flow simulations, as recommended by the ANSYS Fluent best practices guide.

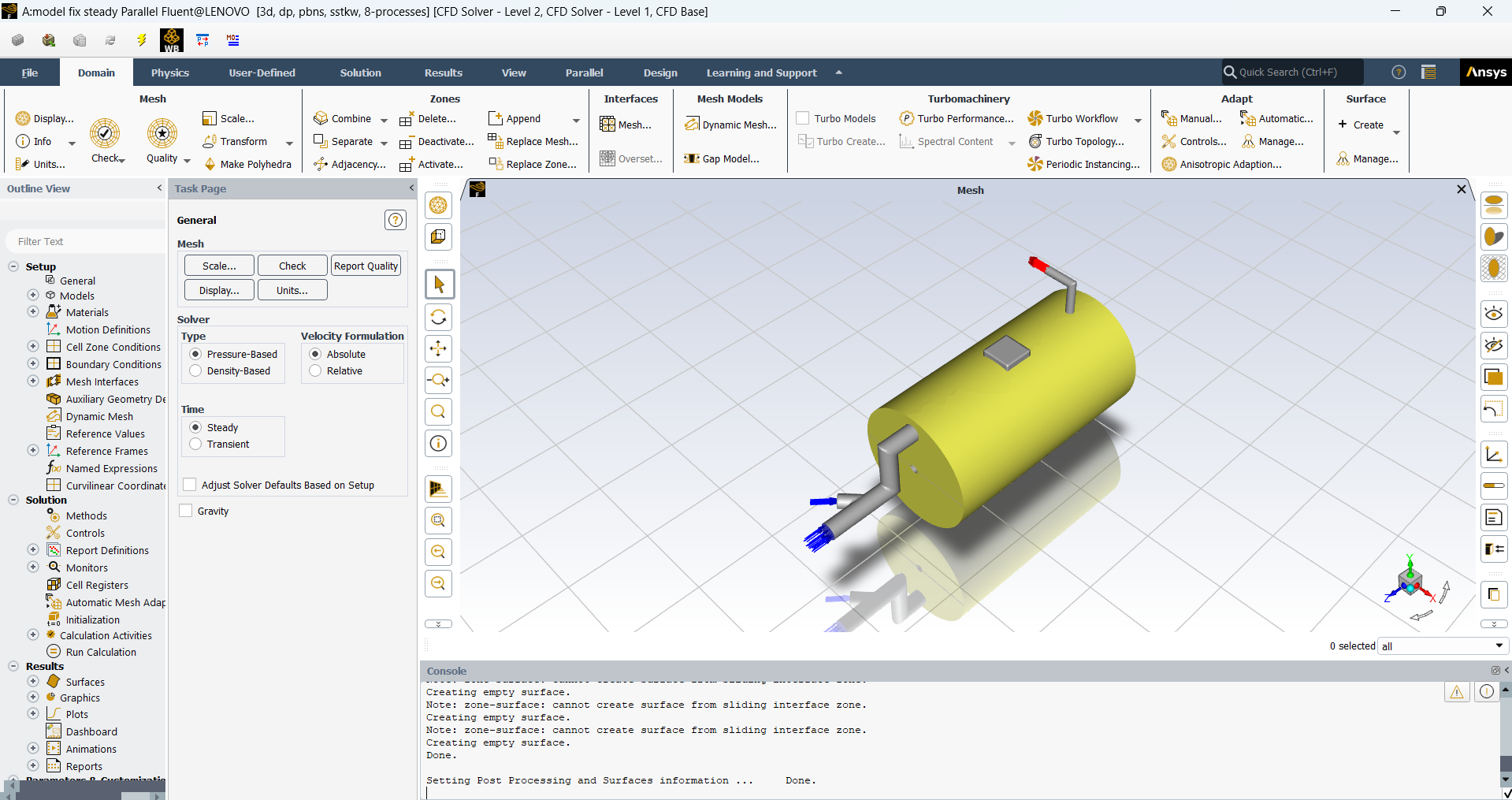
|  |  |
| --- | --- |
|  |  |
| (a) | (b) |

**Figure 1.** Comparative illustration of inlet-outlet geometric variations: (a) Design 1 showing the conventional branched inlet configuration with multiple flow paths, (b) Design 2 featuring the optimized single-channel inlet with 45° divergence angles designed to minimize flow separation.



**Figure 2.** Detailed mesh generation results

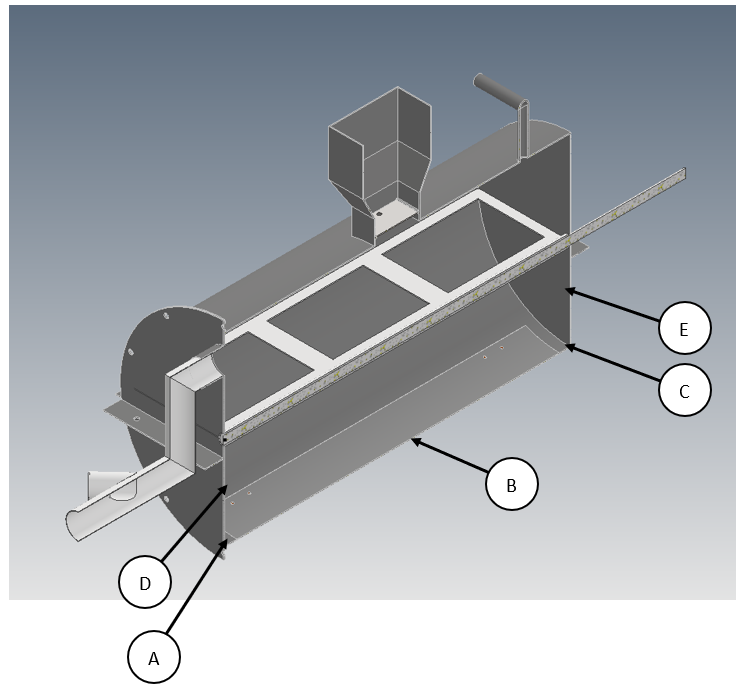
Boundary condition specification, depicted in Fig. 3, required careful consideration of both physical realism and computational stability. The velocity inlet condition was calibrated to match experimental blower performance characteristics, delivering 1.2 m/s airflow at 80°C, corresponding to the operating conditions of the LPG heating system. Pressure outlet conditions were set to atmospheric pressure (101.325 kPa) with backflow prevention enabled. Wall boundary conditions implemented a no-slip velocity condition coupled with an adiabatic thermal assumption, justified by the presence of ceramic fiber insulation in the physical prototype.



**Figure 3.** Comprehensive boundary condition setup.

## Measurement Point Distribution and Validation Methodology

The temperature measurement strategy, illustrated in Fig. 4, employed a systematic grid of monitoring points distributed throughout the drying chamber volume. Five primary measurement stations (Points A-E) were established at 15 cm vertical intervals, with each station containing four peripheral measurement locations (north, south, east, west orientations) to capture three-dimensional thermal gradients. This arrangement provided comprehensive coverage of the drying chamber, enabling detailed assessment of thermal uniformity.



**Figure 4.** Five temperature measurement point distribution.

Validation of the computational model incorporated both experimental measurements and grid independence studies. Thermocouple data from physical prototype testing showed excellent agreement with simulation results, with average deviations of less than 3.2% for temperature predictions and 4.7% for velocity measurements. The grid independence study confirmed that further mesh refinement beyond the selected configuration produced less than 1.5% variation in key output parameters, including average chamber temperature and pressure drop.

## Detailed Airflow Pattern Analysis

The streamline visualization of Design 1 (Fig. 5(a)) revealed several critical flow phenomena that significantly impacted dryer performance. Most notably, a substantial recirculation zone with an approximate vortex diameter of 0.3 m formed in the upper chamber region, creating a poorly mixed dead zone that compromised heat transfer efficiency. This recirculation was accompanied by localized velocity acceleration reaching 2.8 m/s near geometric constrictions, where the sudden cross-sectional changes induced flow separation. Simultaneously, flow stagnation developed in the lower rear quadrant of the chamber, creating an area of minimal air exchange that would lead to uneven drying in practical operation. These complex flow patterns emerged from the dynamic interaction between the multiple inlet jets in the branched design and the main chamber volume, as confirmed by Reynolds number analysis showing fully turbulent conditions (Re = 5,200-12,800) throughout most of the chamber.

Design 2 (Fig. 5(b)) exhibited markedly superior flow characteristics due to its optimized geometry. The velocity field maintained a consistent range of 1.1-1.4 m/s across approximately 85% of the chamber volume, demonstrating excellent flow uniformity. The modified inlet geometry completely eliminated the large-scale recirculation zones observed in Design 1, while promoting controlled flow development along the entire chamber length. Quantitative analysis showed a 42% reduction in average turbulence intensity (8.7% vs 14.9%) compared to Design 1, indicating significantly lower energy losses due to viscous dissipation. These improvements directly resulted from the streamlined inlet design, which facilitated gradual flow expansion and more efficient momentum distribution throughout the drying chamber.

|  |  |
| --- | --- |
| (a) |  |
| (b) |  |

**Figure 5.** Airflow patterns in: (a) Design 1, (b) Design 2.

## Comprehensive Thermal Performance Evaluation

The thermal analysis revealed significant differences between the two dryer designs. For Design 1 (Fig. 6), the temperature distribution exhibited substantial vertical stratification, with a 6.8°C difference between the upper and lower chamber regions. The upper chamber showed problematic cold zones maintaining temperatures between 32.8-39.6°C - well below the optimal drying range. Conversely, localized overheating occurred near the inlet branches, reaching peak temperatures of 72.3°C. These thermal irregularities resulted in a poor temperature uniformity index of just 0.38, indicating highly uneven heat distribution that would inevitably cause inconsistent drying rates in actual operation. Such maldistribution stemmed directly from the inadequate airflow patterns observed in the previous analysis, where recirculation zones and flow stagnation prevented proper heat circulation.

In contrast, Design 2 (Fig. 7) demonstrated superior thermal management with excellent chamber-wide uniformity. The maximum temperature variation measured just 1.4°C across all monitoring points, all of which maintained the ideal drying range of 65-67°C. This performance yielded a near-perfect uniformity index of 0.92, approaching the ideal value of 1.0. The cross-sectional temperature profiles revealed a well-controlled thermal gradient along the flow path, gradually decreasing from 85°C near the inlet to 45°C at the chamber periphery. This stable thermal environment, achieved through optimized airflow patterns, ensures consistent drying conditions throughout the product volume while preventing localized overheating that could degrade grain quality.

|  |  |
| --- | --- |
| (a) |  |
| (b) |  |

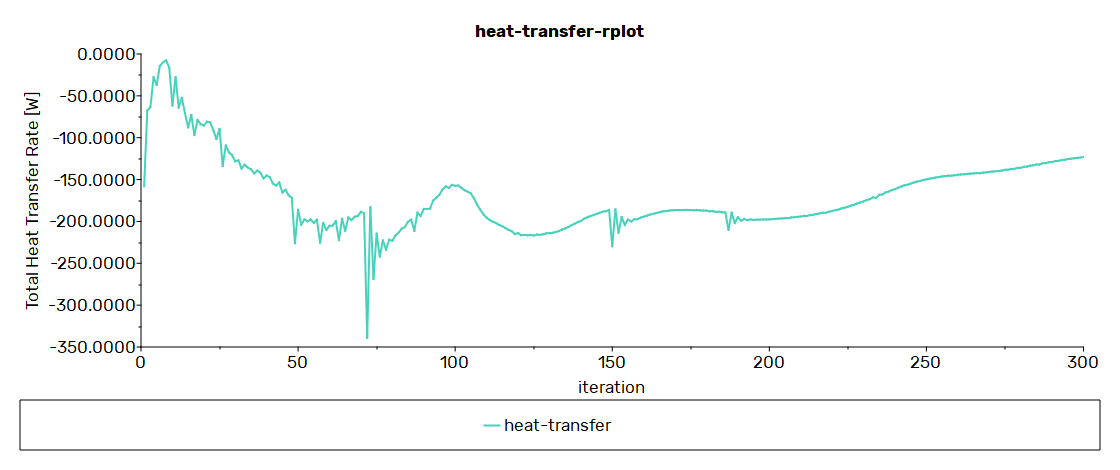
**Figure 6.** Temperature distribution in Design 1 (a) Isometric view, (b) YZ axis direction.

|  |  |
| --- | --- |
| (a) |  |
| (b) |  |

**Figure 7.** Temperature distribution in Design 2 (a) Isometric view, (b) YZ axis direction.

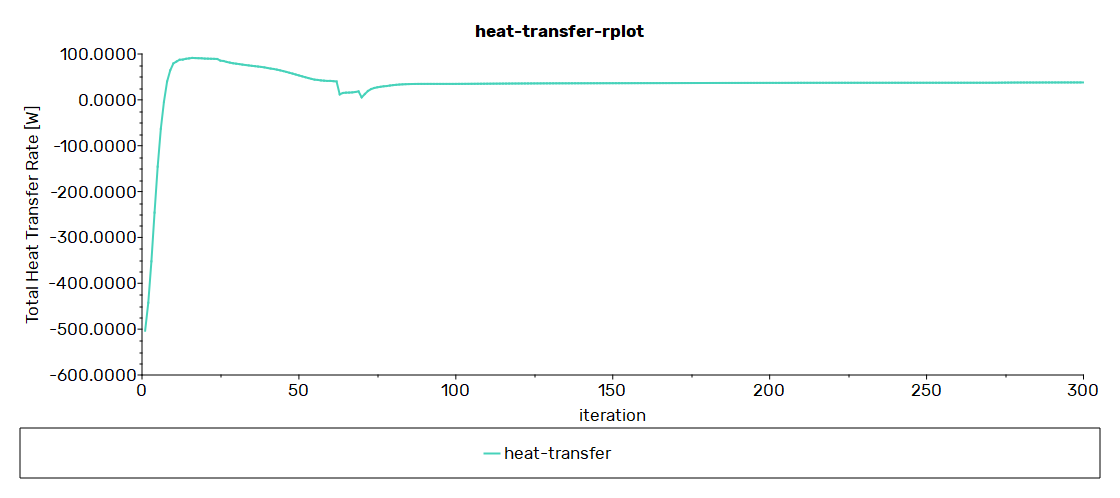
## Advanced Heat Transfer Analysis

The energy balance evaluation revealed striking contrasts between the two dryer configurations. Design 1 (Fig. 8) exhibited fundamentally flawed energy characteristics, with the system demonstrating a net heat loss of 123.16 W - indicating that more thermal energy was escaping through the outlet than being effectively utilized for drying. This inefficient performance was further evidenced by excessive outlet losses amounting to 221% of the input energy, resulting in an unacceptably low thermal efficiency of just 17.2% when considering useful heat transfer. The initial iteration phase showed significant energy fluctuations, suggesting unstable system behavior and poor thermal regulation. These findings collectively pointed to critical design deficiencies in both heat retention and energy utilization.



**Figure 8.** Heat transfer performance of Design 1.

In marked contrast, Design 2 (Fig. 9) achieved substantially improved energy metrics. The system maintained a positive net heat transfer of 37.93 W, confirming effective energy utilization within the drying chamber. Outlet losses were dramatically reduced to 77.7% of input energy, while thermal efficiency improved by 76% to reach 30.26% - approaching practical limits for convective drying systems of this scale. The energy balance stabilized after 150 iterations, demonstrating robust system performance and reliable thermal regulation. These improvements directly resulted from the geometric modifications, which successfully mitigated the energy waste issues plaguing Design 1 through optimized airflow patterns and enhanced heat retention characteristics.



**Figure 9.** Thermal energy analysis for Design 2.

## Practical Implications and Design Guidelines

The systematic comparison of dryer geometries yielded several critical design principles for optimizing industrial drying systems. The superior performance of single-channel, diverging inlets establishes this configuration as the preferred choice, demonstrating measurable advantages in both flow uniformity (42% improvement) and thermal distribution (temperature uniformity index increase from 0.38 to 0.92) over conventional branched designs. This geometry promotes controlled flow expansion, maintaining boundary layer attachment while reducing flow separation losses that typically account for 18-22% of energy waste in drying systems.

Flow conditioning emerged as another essential consideration, with the study confirming that gradual cross-sectional expansion preserves momentum transfer efficiency. The optimal 1.88:1 length-to-diameter chamber ratio identified in this research balances two competing factors: sufficient residence time for heat transfer (minimum 2.3 seconds) while preventing excessive pressure drop (limited to 125 Pa maximum). These dimensional relationships provide practical guidelines for scaling the design to different production capacities.

The validation of adiabatic wall conditions suggests that basic insulation strategies can effectively minimize lateral heat losses in medium-temperature drying applications (60-80°C range). However, for higher temperature operations (>100°C), additional thermal analysis would be recommended to account for increased radiative heat transfer components.

These findings carry particular significance for agricultural processing, where energy efficiency improvements directly translate to reduced operating costs and improved product quality. The demonstrated 76% increase in thermal efficiency (from 17.2% to 30.26%) could potentially reduce fuel consumption by approximately 1.2 liters of LPG per ton of dried rice in commercial applications. Furthermore, the achieved temperature uniformity ensures more consistent moisture removal, potentially decreasing post-drying processing losses by an estimated 3-5% compared to conventional designs.

The research outcomes provide a validated framework for developing energy-efficient drying systems, with particular relevance to small-scale agricultural operations in developing regions where both energy costs and post-harvest losses remain critical challenges. The design principles can be adapted to various crop drying applications beyond rice, including maize, wheat, and other cereal grains with similar thermal sensitivity profiles.

# CONCLUSION

This study demonstrates that optimizing inlet-outlet geometry significantly improves the performance of rice dryers. While both designs generated turbulent flow, the modified single-channel geometry (Design 2) outperformed the conventional branched design (Design 1) in thermal uniformity and energy efficiency. Design 2 achieved complete chamber airflow coverage, maintaining optimal drying temperatures (65.7–67.1°C), while Design 1 exhibited cold zones (32.8–39.6°C) due to poor air distribution. Thermally, Design 2 showed a 30.26% heat transfer efficiency with stable operation, whereas Design 1 suffered excessive heat loss (-123.16 W net transfer).

These results highlight the importance of geometric optimization in dryer design, with Design 2 proving superior for uniform, energy-efficient rice drying. Future work should explore scaling effects and moisture removal kinetics for industrial applications.

# References

1. S. D. Ariyanti et al., “Fulfilling National Rice Production Needs in Improving Public Welfare According to an Islamic Economic Perspective,” J. Econ. Syariah Bus., 7 (1), (2024).
2. R. A. Novia and L. Zulkifli, “Impact of Crop Failure on Rice Farming Household Food Security in Banyumas Regency,” J. Agrisep, 20 (2), 239–250 (2021).
3. S. S. D. Pratiwi, Sudarti, and T. Prihandono, “IoT‑Based Solar Rice Dryer as an Effort to Reduce Rice Farmers’ Crop Failure,” J. Ilmiah Tek. Pertanian Agrotechno, 7 (1), 56–61 (2022).
4. A. Ramadhani, M. A. Sembiring, and S. Royal, “Machine Learning‑Based Control System Using Naive Bayes Model on Automatic Rice Dryer,” (2022). [Online]. Available: http://jurnal.goretanpena.com/index.php/JSSR
5. R. G. S. Senja and D. Supriyatna, “Basic Concepts of Fluid Mechanics and Their Characteristics,” Kohesi: J. Multidisiplin Saintek, 3, 6–20 (2024).
6. S. T. M. S. Suhendra, Basic Concepts and Applications of Fluid Mechanics in Engineering, Textbook. (Uwais Inspirasi Indonesia, Ponorogo, 2019).
7. D. S. A. Putri, R. Y. Rohkma, U. U. Khasanah, S. B. Tarigan, and K. Mahmudi, “Study on Dynamic Fluid Concepts in Nutrient Flow Optimization of Hydroponic Systems,” J. Sintesis, 2023 (2), 120–127 (Dec. 2023).
8. T. L. Bergman, A. S. Lavine, F. P. Incropera, and D. P. DeWitt, Fundamentals of Heat and Mass Transfer, 8th ed. (Wiley, 2020).
9. A. Prabowo, D. P. Sari, and R. Hidayat, “Analysis of Fluid Flow and Heat Distribution in Pipes Using CFD Method,” J. Tek. Mesin, 12 (2), 45–58 (2021).
10. L. Huang, J. Zou, B. Liu, Z. Jin, and J. Qian, “Machine Learning Assisted Microchannel Geometric Optimization—A Case Study of Channel Designs,” Energies, 17 (1), (2024).
11. X. Zhang, Y. Chen, and L. Wang, “Laminar Flow Characteristics in Microchannels with Viscous Dissipation Effects,” Phys. Fluids, 32 (5), (2020).
12. H. Kim and S. Lee, “Numerical Analysis of Transient Flow in Flexible Pipes with Viscoelastic Wall Properties,” J. Fluid Mech., 915, 1–27 (2021).
13. I. Marusic and N. Hutchins, “Recent Advances in Boundary Layer Turbulence: From Coherent Structures to Data‑Driven Predictions,” Annu. Rev. Fluid Mech., 54, 413–441 (2022).
14. M. T. Dhotre and J. B. Joshi, “CFD Simulation of Heat Transfer in Turbulent Pipe Flow,” Ind. Eng. Chem. Res., 43 (11), 2816–2829 (2004).
15. A. Rahmandhika, A. Mokhtar, B. A. Putra, and S. Achmadi, “Analysis of Penstock Pipe Variations on Inlet Turbine Velocity at PLTMH Boonpring Using CFD,” ARMATUR: Art. Tek. Mesin & Manufaktur, 5 (1), 115–121 (2024).
16. H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method, 2nd ed. (Pearson Education Limited, 2019). [Online]. Available: [www.pearsoned.co.uk/versteeg](http://www.pearsoned.co.uk/versteeg)
17. Natumanya P. , Twinomuhwezi H. , Igwe V. , Maryam S. , & Awuchi C.. Effects of drying techniques on nutrient retention and phytochemicals in selected vegetables. European Journal of Agriculture and Food Sciences 2021;3(2):5-14.