CFD-based Optimization of Stove Designs for Sustainable Heating in Mongolian Gers

Battulga Nasanjargal[1*] and Gantuya Ganbat[1]

[1]Faculty of Mechanical and Electrical Engineering, German-Mongolian Institute for Resources and Technology, Ulaanbaatar, Mongolia
[2]Faculty of Raw Materials and Environmental Engineering, German-Mongolian Institute for Resources and Technology, Ulaanbaatar, Mongolia
battulga@gmit.edu.mn

Abstract. Efficient and safe heating is essential for maintaining thermal comfort and reducing indoor air pollution in traditional Mongolian gers, particularly during the heating season. This study presents a computational fluid dynamics (CFD) investigation of thermal performance in various stove geometries commonly used in gers. Multiple stove designs were modeled to analyze combustion-driven heat transfer and air circulation patterns within the confined space of a stove.

The simulations focused on evaluating temperature distributions, flow dynamics, and heat flux across the stove body and surrounding area under fuel combustion conditions. To ensure model reliability, simulated stove wall and inner stove temperatures were validated against experimental measurements obtained during controlled fuel ignition tests conducted in the field. The comparison revealed strong agreement, confirming the validity of the CFD approach.

Results demonstrated that stove geometry significantly influences the performance of the stove. These findings highlight the importance of stove design optimization for both energy efficiency and occupant health in ger households.

The study offers a foundation for the development of improved heating solutions tailored to Mongolia's unique housing and climatic conditions. The integration of CFD tools with empirical field data presents a promising approach for evaluating and enhancing traditional stoves, contributing to sustainable and low-emission heating technologies in cold-climate settings.

Keywords: CFD simulation, Ger stove, Stove geometry, Heat transfer.

1 Introduction

Efficient and clean domestic heating remains a critical issue in cold-climate regions such as Mongolia, where average winter temperature is approximately −20°C [1]. In traditional Mongolian dwellings (*gers*), coal-fired stoves are the primary source of heat. While these stoves are indispensable for maintaining indoor thermal comfort, they are often characterized by poor combustion efficiency and high emissions of harmful pollutants, including fine particulate matter (PM2.5), carbon monoxide, and sulfur dioxide [2]. These emissions contribute to severe winter air pollution,

A. Lkhamsuren et al. (eds.), *Proceedings of the 3rd International Conference Resources and Technology (RESAT 2025)*, Advances in Engineering Research 284,
https://doi.org/10.2991/978-94-6463-928-5_17

particularly in urban areas such as Ulaanbaatar [3], and have been linked to respiratory and cardiovascular health issues.

Given the urgent need to improve energy efficiency and reduce emissions, significant attention has been directed toward the design optimization of domestic stoves. Improved stove geometry and airflow management can enhance heat transfer, reduce fuel consumption, and lower pollutant output [4]. However, experimental testing of stove prototypes is time-consuming, costly, and sometimes impractical. As an alternative, Computational Fluid Dynamics (CFD) modeling offers a robust and cost-effective approach to simulate the thermal and fluid behavior of stoves under varying design and operating conditions [5]. CFD modeling has been previously applied to evaluate indoor environment of gers, particularly simulating the dispersion of carbon monoxide emissions from coal-fired stoves [6].

CFD allows researchers to analyze complex heat transfer mechanisms, including conduction in solids and natural convection in fluids, and to investigate the impact of structural modifications such as baffles, internal tubes, and insulation layers on overall stove performance [7]. The use of CFD in stove design has been increasingly adopted in recent years, particularly in regions where indoor air pollution poses a major health risk and clean fuel alternatives remain economically inaccessible.

In the context of Mongolian gers, this study investigates the use of CFD modeling to simulate and improve the heat distribution efficiency of a conventional coal-burning stove. Particular emphasis is placed on the impact of an internal vertical tube array—a passive heat exchanger mechanism—on enhancing thermal performance. The simulation was conducted using COMSOL Multiphysics®, which solves time-dependent non-linear partial differential equations governing heat transfer and airflow within the stove's three-dimensional geometry.

The objective of this research is twofold: (1) to demonstrate the feasibility of using validated CFD models to improve stove design for ger households, and (2) to inform the development of cleaner and more efficient heating solutions in cold-climate, low-income settings. The findings offer significant implications for reducing coal consumption, improving indoor thermal comfort, and lowering the health burden associated with household air pollution in Mongolia.

2 Methodology

2.1 Stove wall temperature measurement

The measurement was conducted in Mongolian traditional ger to assess the heat transfer characteristics of a household coal-burning stove (Figure 1) and to monitor the thermal behavior of its external surfaces during combustion.

The measurement was initiated at an ambient temperature of 14±1°C, and a 1kg sample of dry coal was ignited manually using a match and small kindling material. No forced ventilation was applied, simulating typical household conditions. The coal was allowed to burn under natural conditions. The measurement continued until the stove top temperature reached approximately 120°C, at which point the combustion process was deemed to have reached steady-state surface heating.

To evaluate the stove's thermal output and its potential implications for indoor air quality and energy efficiency, the outer wall temperature of the stove was measured at 5-minute intervals using a non-contact infrared thermometer (DT8012T, 2°C accuracy). Throughout the experiment the ventilation conditions were kept constant. No additional fuel was added during the test to ensure uniformity in combustion behavior. All safety precautions were strictly followed to minimize risk during high-temperature operation. The recorded temperature data provide insight into the stove's heat retention and external surface temperature.

Fig. 1. Typical household stove in Mongolian traditional ger.

2.2 Numerical simulations

In this study, numerical simulations were conducted using the COMSOL Multiphysics® Computational Fluid Dynamics (CFD) to model the heat transfer dynamics of a domestic coal-burning stove. The objective was to simulate the development of temperature fields within the stove over time, accounting for realistic physical and geometric parameters observed during the experimental setup.

Simulation framework. The simulation employed a system of non-linear, unsteady-state partial differential equations to capture the thermal behavior and energy distribution across key components of the stove, including thermal cavities, air inlets, and the influence of gravity. [7, 8]. The dominant heat transfer mechanisms, which are forced and natural convection within air domains, were both incorporated into the models. [9].

A time-dependent simulation approach was adopted to replicate the duration of the standard experimental energy test, which spans approximately 1 hour.

Geometry and Meshing. The main body (40x30x60cm) was modeled in three-dimensional geometry, capturing all major components essential to heat generation and dissipation. These elements include:

- Heating elements (coal).
- A front door with 3 air inlets and one outlet.
- The stove frame and air cavities.

A non-uniform meshing was employed, with denser meshes near high-gradient regions such as the heating zone, inlets and output (Fig. 2). Finite element types and sizes were selected based on the complexity and thermal sensitivity of each domain. The mesh configuration, totaling approximately 56k elements, was optimized to achieve a balance between computational efficiency and simulation accuracy.

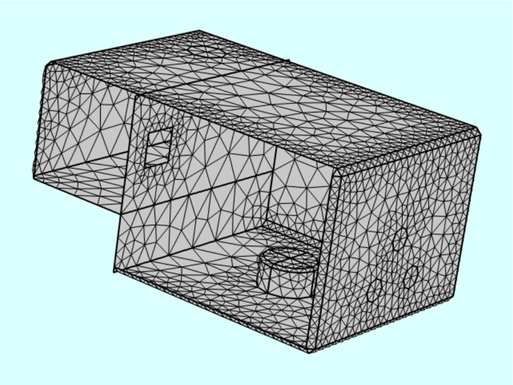

Fig. 2. Three-dimensional view of the finite element model geometry and mesh used in the CFD simulations.

Governing equations. The heat transfer analysis was governed by the energy conservation equation, which accounts for transient heat conduction and internal heat generation:

$$\rho C_p\left(\frac{\partial T}{\partial t}\right) + \nabla \bullet (-k\nabla T) = Q$$

where: T is temperature (K), ρ is density (kg/m^3), C_p is specific heat capacity at constant pressure (J/kg·K), k is thermal conductivity (W/K·m), and Q is the heat source (W).

In addition to thermal transport, species transport was modeled to simulate the behavior of gaseous byproducts and consumption-related diffusion. This was expressed through the species conservation equation:

$$\frac{\partial c}{\partial t} = D\nabla^2 c + R$$

where, c is the species concentration (mol/m^3), D is the diffusion coefficient (m^2/s), and R is the volumetric reaction rate (mol/s·m^3).

These equations were numerically solved within the COMSOL Multiphysics using time-dependent solver.

Boundary conditions. All components in the model were considered as domains with solid surfaces, where heat transfer in solid was governed primarily by conduction. Initial conditions and experimental parameters were included as model inputs:

- The initial temperature was set at 14±1°C, consistent with the experimental measurement setup.
- The initial temperature of the coal and surrounding air were set based on measured value.
- The stove operation mode was defined as static, with a surface temperature target of ca. 120°C.

Tube array configuration. To enhance the heating efficiency of the stove, vertical tubes in a regular grid pattern were integrated into the rear section of the stove. Several configurations with varying numbers of tubes were simulated in COMSOL to determine the optimal arrangement for maximizing thermal performance. Specifically, configurations with 5×4, 5×6, 5×8, and 5×12 tube arrays were tested and compared based on their heat efficiency characteristics.

3 Results and Discussions

The Finite Elements Model provides detailed insights into the transient thermal behavior of all major components of the stove. In this study, the evolution of temperature over time was analyzed to assess the impact of internal structural modifications, specifically, variations in the vertical tube array configuration, on heat transfer and overall thermal performance.

To evaluate the effectiveness of different tube configurations, simulations were conducted using various tube array arrangements: Original (no tubes), 5×4, 5×6, 5×8, and 5×12. The simulation tracked temperature and velocity profiles over a 60-minute period, corresponding to the duration of the experimental measurement test.

Table 1 presents the average top plate temperature at 60 minutes for each configuration.

Table 1. Simulated top plate average temperature that corresponds to different tube arrangements at 60 minutes.

Tube array arrangement	Original (without tubes)	5×4 tubes	5×6 tubes	5×12 tubes	5×8 tubes
Average temperature of top plate at 60 minutes (°C)	85	89	97	89	87

As shown in Table 1, the 5×6 tube configuration resulted in the highest average top plate temperature (97°C) at the end of the simulation period. This indicates that the inclusion of a well-optimized vertical tube array significantly enhances the internal heat transfer, leading to more efficient distribution of thermal energy throughout the stove. These findings suggest that design modifications incorporating specific internal geometries, such as the 5×6 tube array, can improve the thermal efficiency of the stove. This may have practical implications for energy-saving designs in domestic heating systems.

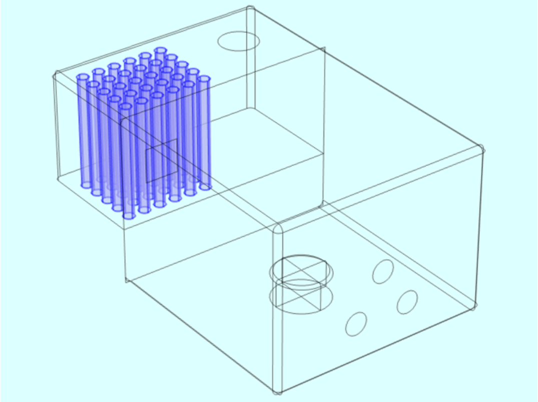

Fig. 3. CFD model of the stove with a 5×6 tube array configuration for enhanced heat transfer.

Figure 3 presents a three-dimensional wireframe representation of the CFD model of the stove. The most prominent feature is the internal 5×6 tube array, consisting of 30 individual vertical tubes arranged in a regular grid pattern. This tube bank is strategically positioned within the stove's heat exchange section to maximize the surface area available for heat transfer from the hot combustion gases to the surrounding air. The vertical arrangement of tubes enhances natural convective heat transfer, allowing hot combustion gases to flow efficiently around and through the tube bundle. The design of the array introduces geometric complexity that encourages the formation of turbulent flow structures, which further improve thermal exchange by disrupting laminar flow and increasing the heat transfer coefficient.

The incorporation of this internal structure reflects a targeted approach to optimize heat extraction from the combustion process. By simulating various tube array configurations, the model provides valuable insights into their impact on overall heat transfer performance, offering a cost-effective means of design evaluation prior to the fabrication of physical prototypes.

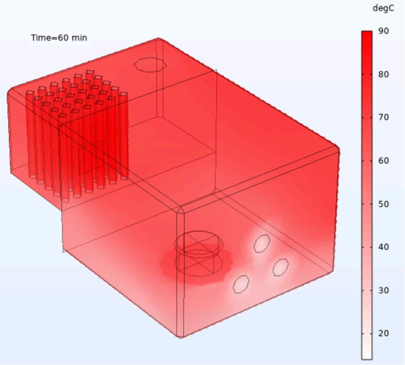

Fig. 4. Simulated temperature distribution within the stove at 60 minutes, illustrating the temperature development.

Figure 4 the simulated temperature distribution within the stove after 60 minutes of operation, as generated by the CFD model. This visualization utilizes a color gradient scale, a standard method in CFD analyses, to represent temperature variations across the stove's internal and external components. The image clearly illustrates the spatial distribution of heat, with the highest temperatures concentrated in the combustion chamber and heat exchange zones, as expected. Gradual temperature decreases are observed toward the stove’s outer surfaces, indicating the directional flow of thermal energy through conduction and convection mechanisms. This temperature field offers valuable insights into the thermal efficiency and effectiveness of the stove’s design. Specifically, regions with uniform temperature distribution suggest efficient heat transfer and minimal energy loss, while areas with sharp gradients may highlight design limitations or thermal bottlenecks. The simulation results, therefore, serve as a basis for evaluating and optimizing stove geometry to achieve more effective heat utilization and improved overall performance.

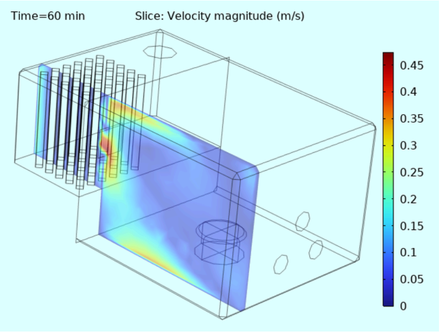

Fig. 5. Slice plot of the simulated velocity magnitude within the stove, used to define boundary conditions for single-tube simulations.

Figure 5 presents a slice plot of the simulated velocity magnitude within the stove chamber. In CFD, a "slice plot" is a cross-sectional representation that illustrates the spatial distribution of a variable (in this case, velocity magnitude) within a specific plane of the 3D domain. This type of visualization provides a detailed view of internal flow dynamics and allows for the identification of high- and low-velocity zones. The figure depicts the flow behavior of air and combustion gases as they circulate through and around the internal components of the stove. Variations in velocity magnitude are represented by a continuous color gradient, with warmer colors typically indicating higher flow speeds. This visualization reveals important features such as the directionality and turbulence of the flow, recirculation zones, and regions of stagnant or slow-moving air.

Accurate definition of boundary conditions is critical in CFD modeling, as these parameters determine the physical constraints of the simulated environment. In this study, the observed flow patterns shown in Figure 5 were used to guide the refined simulation of flow behavior around a single vertical tube, one of the key elements in the heat exchange structure. The slice plot highlights how flow accelerates between tubes and slows in confined areas, offering valuable insight into how geometric design influences convective heat transfer.

Overall, the velocity distribution data provide a deeper understanding of the stove's internal aerodynamics, which directly affect heat transfer performance and combustion efficiency. These results contribute to the broader goal of optimizing stove design for improved thermal utilization and reduced emissions. In CFD, boundary conditions are essential inputs that define the physical behavior of the system being simulated. In this case, the overall flow pattern within the stove (shown in Figure 5) was used to inform the simulation of flow around a single tube.

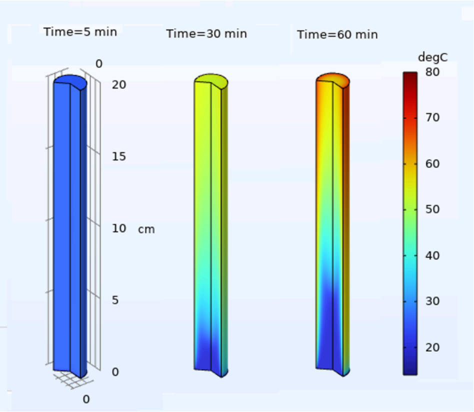

Fig. 6. Simulated time dependent temperature distribution along a single tube at 5 min, 30 min, and 60 min.

Figure 6 illustrates the simulated temperature distribution along the length of a single tube within the stove's heat exchanger. This detailed visualization provides a critical insight into the thermal conditions experienced by the tube surface and interior, highlighting the spatial variation in temperature along its vertical axis. Analyzing the internal temperature profile of the tube is essential for evaluating its heat conduction performance and the overall efficiency of heat transfer from the combustion to the surrounding air. Regions exhibiting steep temperature gradients indicate zones of high thermal flux, which may be crucial for optimizing energy extraction.

Furthermore, understanding the temperature distribution aids in material selection and structural design by identifying potential areas of thermal stress or degradation. This information supports the engineering of heat exchanger components that are both thermally efficient and mechanically robust, ensuring long-term operational stability and improved stove performance.

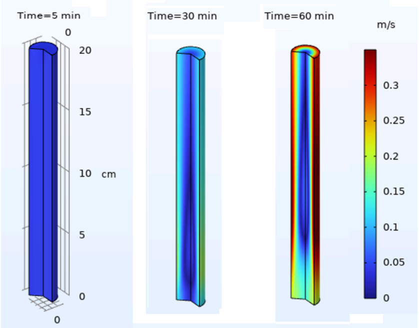

Fig. 7. Simulated time dependent velocity distribution along a single tube at 5 min, 30 min, and 60 min.

Figure 7 presents the simulated time-dependent evolution of the velocity distribution along a single heat exchanger tube within the stove. The figure displays three snapshots of the velocity profile at 5 minutes, 30 minutes, and 60 minutes of simulation, offering insight into the development of internal flow dynamics over time. At the initial stage (5 minutes), the velocity within the tube remains low (<0.05m/s) and relatively uniform. This suggests that the fluid is in a quiescent state with minimal convective motion, reflecting the early phase of the heating process before significant temperature gradients are established.

By 30 minutes, the emergence of pronounced flow patterns becomes evident. Higher velocities begin to appear along the central axis and upper regions of the tube. This marks the onset of natural convection, as the thermal energy from the combustion process generates buoyancy-driven upward flow.

By 60 minutes, the velocity profile is further developed and intensified, with the central core of the tube exhibiting the highest velocities. The well-developed upward movement of heated gases reflects a fully established convective current, which plays a central role in enhancing heat transfer from the tube walls to the surrounding air.

This time-resolved analysis highlights the importance of natural convection mechanisms within the heat exchanger tube. The progression from near-static conditions to dynamic, temperature-driven flow contributes significantly to the overall thermal efficiency of the stove. These findings underscore the relevance of CFD-based diagnostics in understanding the transient performance of heat exchange components in solid fuel combustion systems.

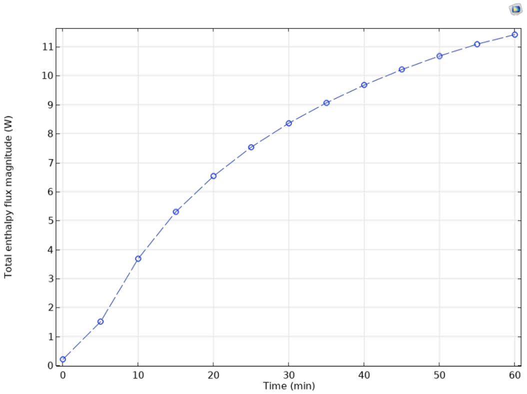

Fig. 8. Simulated total enthalpy flux magnitude over time for single tube.

Figure 8 depicts the simulated total enthalpy flux magnitude for a single heat exchanger tube as it evolves over a 60-minute period. The y-axis represents the magnitude of the total enthalpy flux through the top surface of the tube, while the x-axis represents time in minutes. The curve shows a clear trend of increasing enthalpy flux over time. After 60 min it reaches value of ca. 10W. The overall shape of the curve highlights the dynamic nature of the heat transfer process during the initial heating phase of the stove operation. The increasing enthalpy flux signifies a greater capacity of the fluid to carry thermal energy, which is a direct consequence of the developing temperature and velocity fields within the tube driven by natural convection, as visualized in Figure 7.

4 Conclusions

This study has demonstrated the effectiveness of using a validated CFD model to analyze and optimize the thermal performance of domestic coal-burning stoves. The simulations valuable insights into the influence of stove geometry, particularly the design of internal heat exchanger, on heat transfer efficiency and air circulation patterns. The results indicate that targeted design modifications, particularly through the incorporation of efficient tube of a 5×6 vertical tube array configurations as identified in this study, can significantly enhance thermal comfort while contributing to a reduction in harmful emissions. These improvements are especially relevant for residents of Mongolian ger, where reliable and efficient heating systems are critical during the long, harsh winter months. Beyond immediate performance benefits, the findings have broader implications for public health, indoor air quality, and energy efficiency in cold-climate regions. This research provides a robust foundation for future development of more efficient stove designs tailored to the specific needs of low-resource and off-grid households.

Furthermore, this study provides a strong foundation for future research and development efforts focused on creating more efficiently burning stoves for use in Mongolian gers. Future work could refine the models, investigate different fuel types, and explore advanced design concepts to maximize heat transfer and minimize emissions, ultimately translating research into practical solutions for sustainable heating in cold-climate regions.

References

1. Ganbat, G., Han, JY., Ryu, YH. et al.: Characteristics of the urban heat island in a high-altitude metropolitan city, Ulaanbaatar, Mongolia. Asia-Pacific Journal of Atmospheric Sciences **49**, 535–541 (2013). https://doi.org/10.1007/s13143-013-0047-5
2. Mongolia - Air pollution in Ulaanbaatar: initial assessment of current situation and effects of abatement measures (English). Sustainable development - East Asia and Pacific Region discussion papers Washington, DC: World Bank. http://documents.worldbank.org/curated/en/866561468274261208
3. Ganbat, G., Lee, H., Jo, HW. et al.: Assessment of COVID-19 Impacts on Air Quality in Ulaanbaatar, Mongolia, Based on Terrestrial and Sentinel-5P TROPOMI Data. Aerosol and Air Quality Research **22**, 220196 (2022). https://doi.org/10.4209/aaqr.220196
4. Zhang, J., He, X., Chen, Y., and Gao, Y.: Numerical analysis of heat transfer enhancement in household stoves with internal fins. Energy Reports, **5**, 671–678 (2019)
5. Kumar, R., and Ghosh, D.: CFD simulation of heat and mass transfer in biomass cookstoves. Renewable Energy, **147**, 2115–2125 (2020).
6. Byambakhorol, B., Ganbat, G., B. Nasanjargal: Measurement and 3-dimensional Simulations of Carbon Monoxide (CO) Distribution in Mongolian Ger. Proceedings of the second International Conference on Resources and Technology (RESAT 2023). Atlantis Press. 117-131 (2023). https://doi.org/10.2991/978-94-6463-318-4_10
7. Afolabi, A. S., & Sadiku, R.: CFD modeling of improved wood-burning stoves for rural applications. Energy for Sustainable Development, **33**, 173–181 (2016).
8. Rek, Z., Rudolf, M., and Zun, I.: Application of CFD Simulation in the Development of a New Generation Heating Oven. Journal of Mechanical Engineering **58**(2), 134-144 (2012)
9. Ait-taleb, T., Abdelbaki, A., and Zrikem, Z.: Numerical simulation of coupled heat transfers by conduction, natural convection and radiation in hollow structures heated from below or above. International Journal of Thermal Sciences **47**, 378-387 (2008)