Case Study

A Comprehensive Study with a Focus on Solar Vacuum Dryer Modeling - Optimizing Solar Dryer Design with Computational Fluid Dynamics for Enhanced Food Preservation

Dr. Oumar Drame
Cheikh Anta Diop University of Dakar/Senegal
Fluid Mechanical and Applications Laboratory

Edited by the Ansys Academic Development Team

education@ansys.com

Ansys Software Used

This case study uses Ansys Discovery™, the 3D product simulation software and Ansys Fluent®, the fluid simulation software.

Summary

This case study introduces Ansys Fluent Software, a leading Computational Fluid Dynamics (CFD) software, as a robust tool for modeling and analyzing the integration of solar energy into various aspects of daily life. Leveraging advanced physical models, Ansys Fluent tool enables precise examination of heat and mass transfer, chemical reactions, and fluid-structure interactions within solar energy systems.

Solar energy offers immense versatility with a wide range of practical applications in everyday scenarios. We explore ten key domains where solar energy plays a crucial role, from simple to complex implementations, including Solar Dryers, Solar Water Heaters, Solar Cookers, and more. Each application harnesses solar energy as a renewable and eco-friendly resource, reducing reliance on fossil fuels and combating climate change.

Through Ansys Fluent tool's sophisticated capabilities, we delve into the intricate dynamics and optimization potentials of these solar-powered solutions. Our aim is to enhance efficiency and sustainability in everyday practices, contributing to a greener future.

Table of Contents

1. Introduction

In this presentation, Ansys Fluent Software, a leading Computational Fluid Dynamics (CFD) software, serves as a powerful tool for modeling and analyzing the integration of solar energy into various aspects of daily life. By leveraging sophisticated physical models, Ansys Fluent Software enables precise examination of diverse phenomena, including heat and mass transfer, chemical reactions, and fluid-structure interactions within solar energy systems.

Solar energy emerges as an immensely adaptable resource with numerous practical applications in everyday scenarios. Here, we explore ten domains where solar energy plays a pivotal role, spanning from straightforward to intricate implementations:

- Solar Dryers
- Solar Water Heaters
- Solar Cookers
- Solar Water Pumps
- Solar Water Purification Systems
- Solar Street Lights
- Solar Chargers
- Solar Ventilation Systems
- Solar Roofs and Solar Tiles
- Solar Power Tower Plants

Each of these applications harnesses solar energy as a renewable and eco-friendly resource, reducing reliance on fossil fuels and actively contributing to the global effort to combat climate change. Through Ansys Fluent Software's advanced capabilities, we delve into the intricate dynamics and optimization potentials of these solar-powered solutions, aiming to unlock greater efficiency and sustainability in everyday practices.

2. Problem Statement

Drying is pivotal for preserving foods by moisture reduction, thereby preventing decay. Traditional methods, which can be weather-dependent and energy-intensive, may lead to uneven drying. Solar dryers, harnessing the sun's free energy, represent a sustainable alternative. These devices, featuring a heat-absorbing dark surface and a transparent cover, ensure more uniform drying while protecting against the elements.

Computational Fluid Dynamics (CFD) using Ansys Fluent software plays a vital role in designing these solar dryers by optimizing air circulation and temperature control for enhanced efficiency and improved food quality preservation. This advanced technical approach is crucial for developing energy-efficient and effective solar drying solutions.

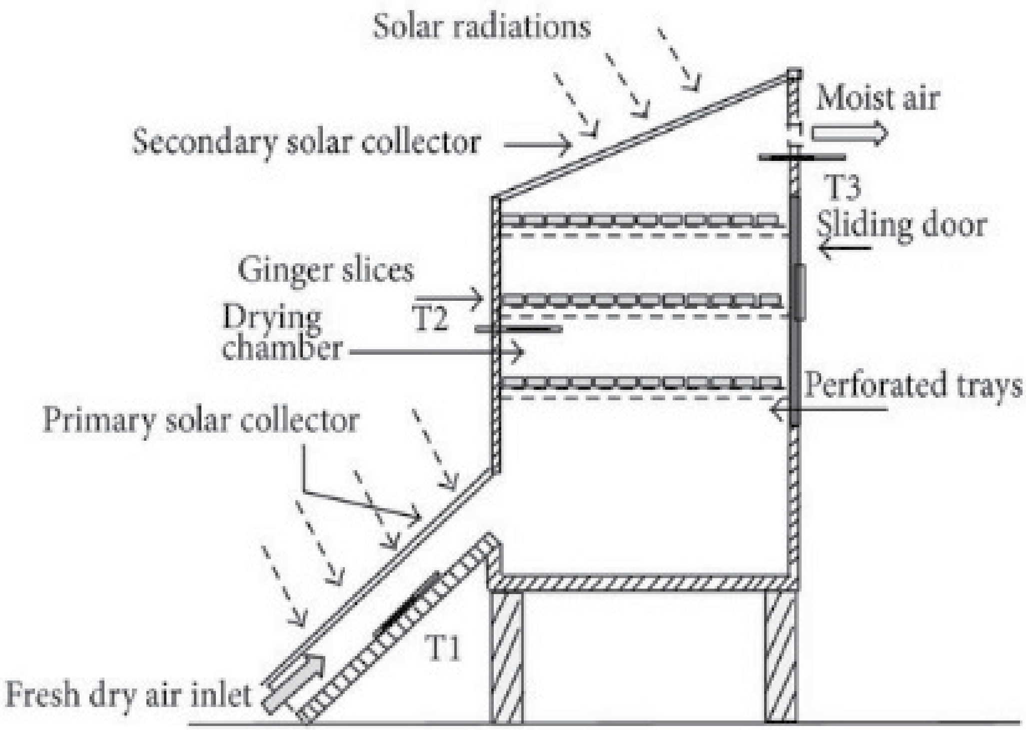

Figure 1 : Mixed Solar dryer

The principal challenge in the domain of Computational Fluid Dynamics (CFD) simulation of solar dryers is rooted in the elucidation and optimization of heat and mass transfer processes within the drying apparatus. Simulations are obligated to address the intricate interactions between the airflow, the drying surface, and the assortment of products undergoing desiccation. Foremost hurdles include the precise modeling of temperature gradients and humidity distributions within the dryer's chamber, forecasting of dehydration rates, and the reduction of drying times whilst preserving the integrity of the product's quality.

Further, the simulations are required to incorporate the variability of meteorological conditions and the distinctive characteristics of the materials constituting the dryer's structure. They must also simulate the influence of the products' geometrical configurations and placement on the drying efficiency, as well as the system's overall energy utilization efficiency. By surmounting these challenges, CFD simulations can contribute significantly to the design of more efficacious solar dryers and to the enhancement of drying operations, tailored to meet the specific demands of the end-users. This endeavor is pivotal in devising custom solutions that mediate between energy conservation and maintaining the desired caliber of the dried commodities, thereby advancing the viability and sustainability of solar drying technology across diverse sectors.

3. Physics Behind the calculation

The CFD (Computational Fluid Dynamics) calculation of the solar dryer is based on modeling the physical principles governing heat and mass transfer inside the dryer. Here is a more detailed explanation with equations:

3.1 Heat Transfer

Heat transfer in the solar dryer primarily occurs through convection and radiation. Convection involves the movement of heated air through the dryer, while radiation concerns the transfer of thermal energy by solar rays and the heated surfaces of the dryer. These processes can be mathematically described by the heat equation (1):

$$\rho C_p \frac{\partial T}{\partial t} + \rho \mathbf{u} \cdot \nabla T = k \nabla^2 T + \dot{q}_\vartheta \quad (1)$$

where ρ is the air density, C_p is the specific heat at constant pressure, T is the temperature, t is time, **u** is the air velocity, k is the thermal conductivity of air, ∇ is the gradient operator, and is the volumetric heat source density.

3.2 Mass Transfer

Mass transfer occurs as moisture evaporates from the products to be dried and is carried by the air. This process can be described by the moisture diffusion equation (2):

$$\frac{\partial \emptyset}{\partial t} + \nabla \cdot (\mathbf{u}\emptyset) = D \nabla^2 \emptyset + S \quad (2)$$

Where ϕ is the moisture molar fraction in the air, D is the moisture diffusion coefficient in air, and S is the source or sink of moisture, which depends on the evaporation rate of the products.

3.3 Navier-Stokes Equations

To model air movement inside the dryer, the Navier-Stokes equations are used. These equations describe the conservation of mass, momentum, and energy for a moving fluid. In laminar flow, the Navier-Stokes equations take the following form (3):

$$\rho \frac{\partial u}{\partial t} + \rho (\mathbf{u} \cdot \boldsymbol{\nabla}) \mathbf{u} = -\nabla p + \mu \nabla^2 \mathbf{u} + \mathrm{f} \quad (3)$$

where ρ is the air density, **u** is the air velocity, p is the pressure, μ is the dynamic viscosity of air, and f is the volumetric force resulting from pressure and density gradients.

By solving these coupled equations in a geometric domain (Figure 2) representing the real-scale of the dryer prototype (Figure 3), it is possible to accurately simulate heat and mass transfer processes and predict dryer performance, including temperatures, evaporation rates, and drying times.

4. Geometry

To conduct this simulation, we initially generated a 3D geometry of the dryer (Figure 2) using Ansys Discovery Software, with dimensions accessible in the associated Geometry file included with this case study. We precisely replicated the dimensions of a prototype dryer developed in collaboration with students (Figure 3). Subsequently, this geometrically accurate representation facilitated the parameterization of various components of the dryer, including the glass panel, absorber, and drying chamber. This parameterization enabled us to proceed with mesh generation. The meshing process is essential for capturing the intricate details of each component and ensuring that the simulation accurately replicates the physical phenomena occurring within the dryer.

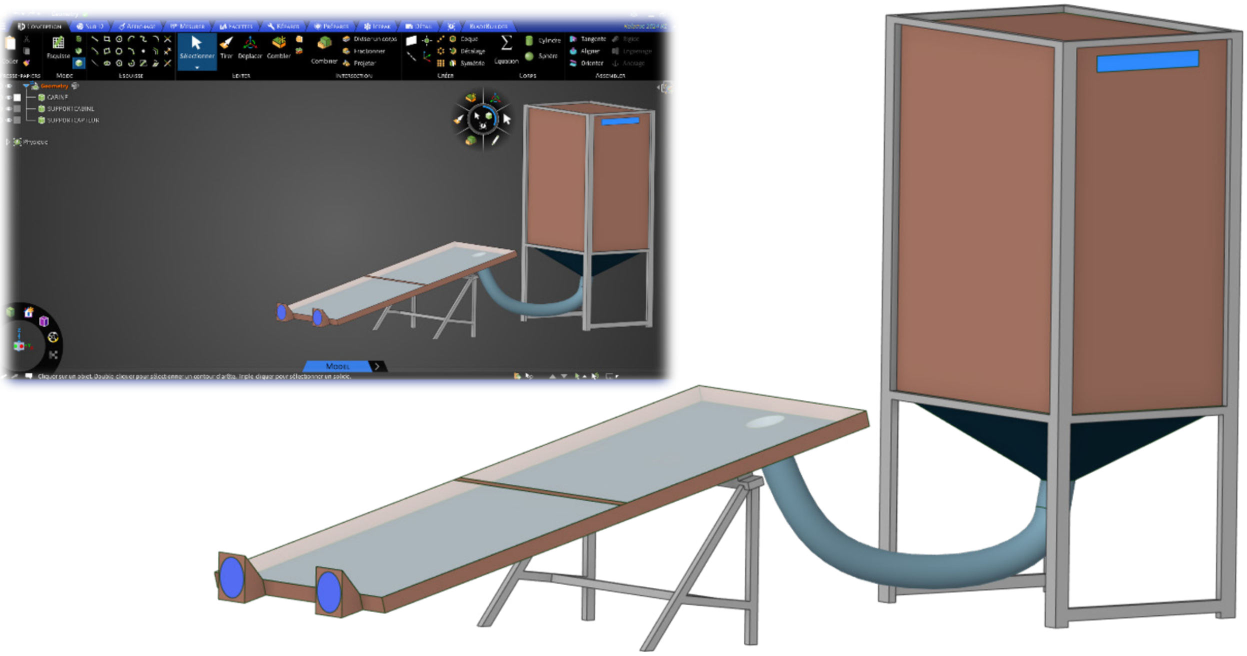

Figure 2: CFD geometry of case study dryer

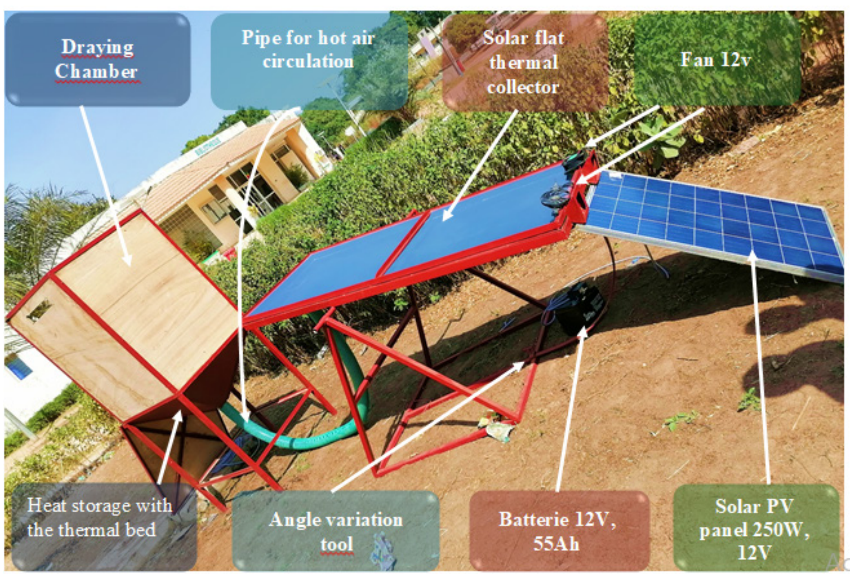

Figure 3: Real-scale of the dryer prototype

4. Meshing

To perform the simulation within the designated domain, we partitioned it into highly refined control volumes using the Ansys Fluent meshing tool. Despite the geometric intricacies, we achieved a mesh of notable quality (Figure 4), characterized by an average orthogonality quality of 0.77 and a predominant maximum orthogonality quality of 0.99. To ensure the integrity of the mesh, we conducted meticulous testing across various element numbers. Among these, the configuration that maintained result fidelity was identified at 1,307,544 elements, presenting a commendable balance between mesh precision and computational efficiency. This rigorous mesh evaluation process ensures the reliability and accuracy of the subsequent simulation results.

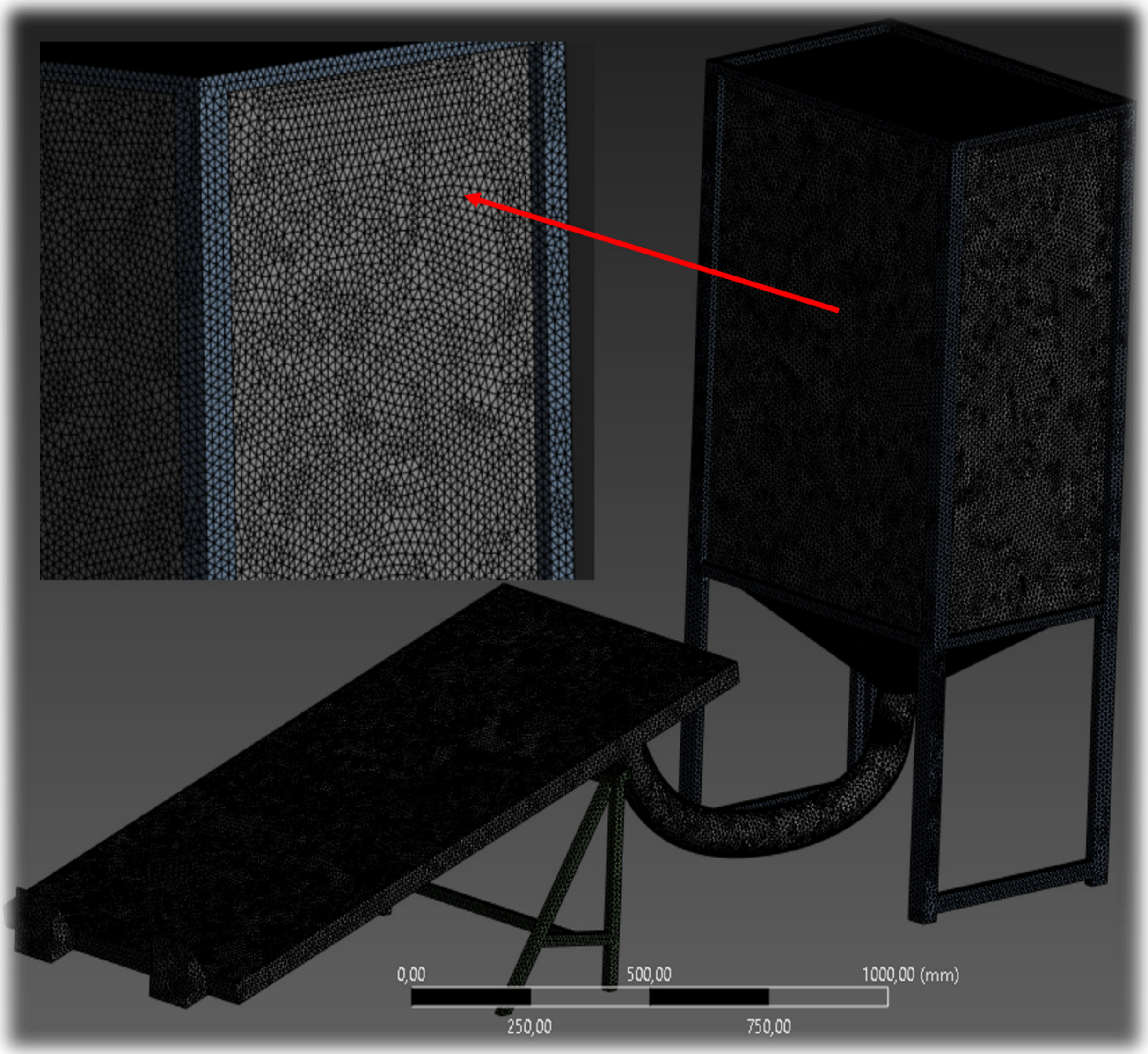

Figure 4: Mesh generated for the dryer

5. Model setup

5.1 Radiation model in Ansys Fluent Software

After activating the energy equation in the model, we carefully selected the Rosseland radiation model from among the radiation models available in Ansys Fluent Software (see Section 50.4.4. Radiation Model Dialog Box in Ansys Help). Additionally, we enabled "Solar Ray Tracing" by selecting "Use Direction Computed from Solar Calculator (Figure 5). This feature allows us to obtain the thermal radiation flux specific to the geographical location where the simulation was conducted without the need for external software.

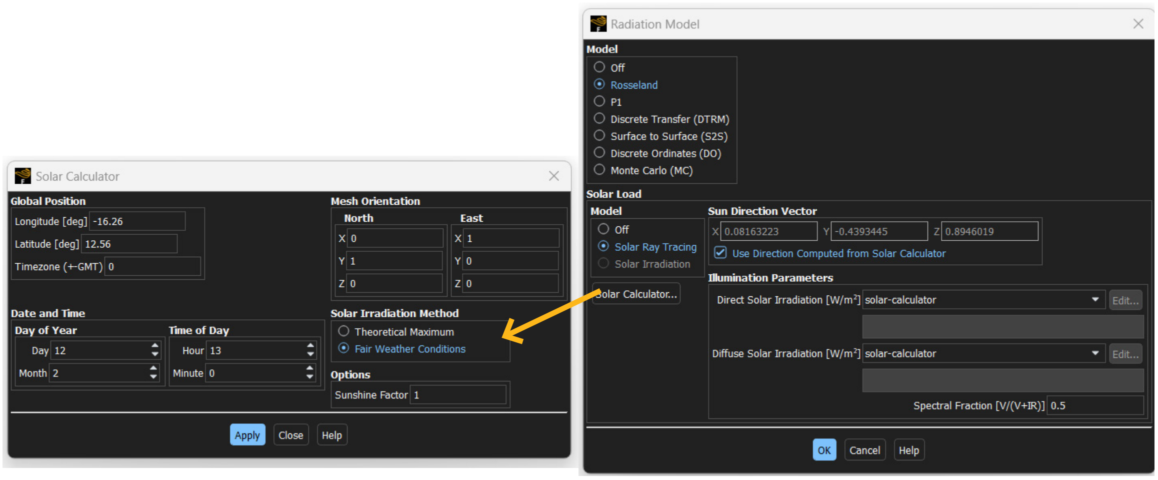

Figure 5: Radiation model definition

6. Boundary conditions

6.1 Inlet

In preparation for the upcoming simulation setup, we engaged in a meticulous and detailed crafting process, during which we precisely adjusted and fine-tuned an extensive and diverse collection of boundary conditions. These specific conditions were judiciously selected and incorporated after conducting an exhaustive and comprehensive analysis of the materials utilized within the framework of the simulation. Additionally, we took into account the prevailing conditions that were rigorously outlined and documented in the associated experimental study, by conducting the simulation using the same simulation conditions with the data from the third hour of drying.

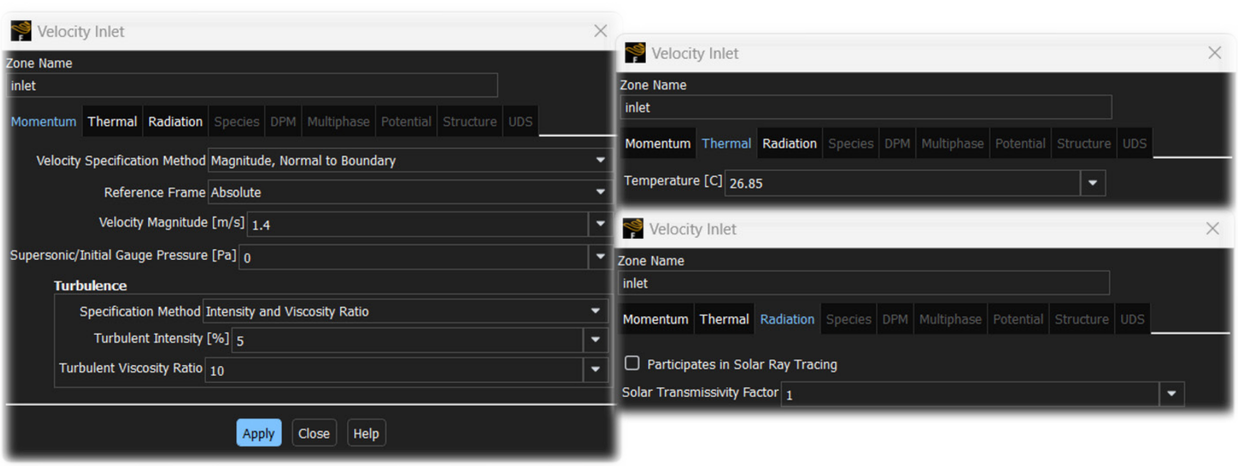

Figure 6: Velocity inlet

6.2 Outlet

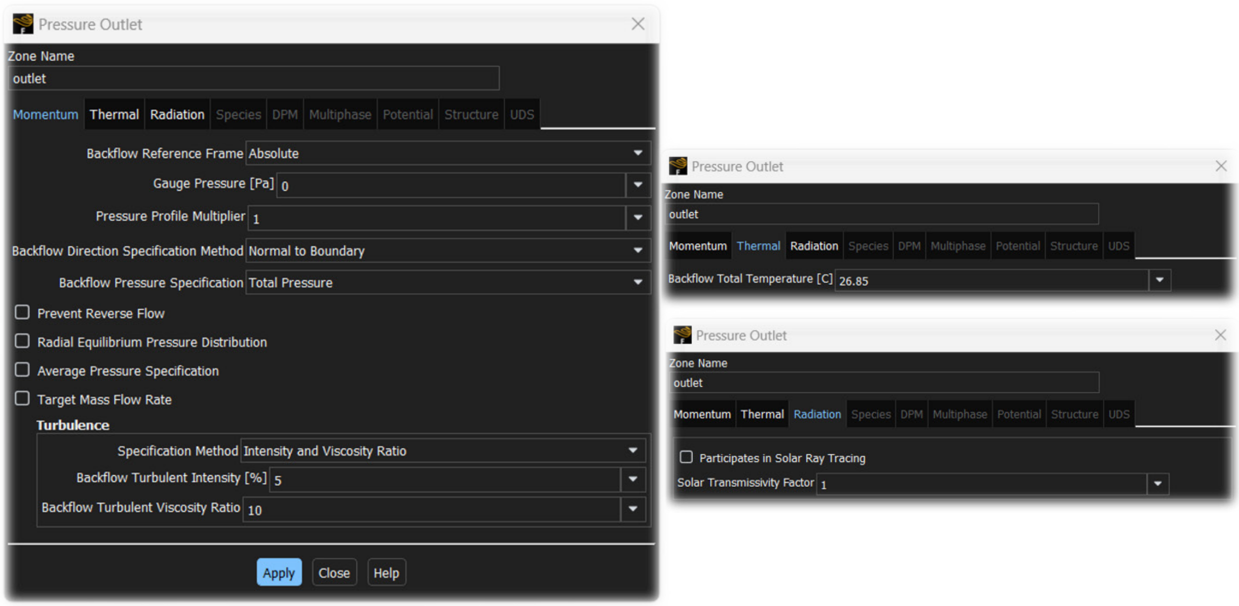

Figure 7: Pressure Outlet

6.3 Walls: Glass

For the boundary conditions applied to the walls within our setup, we operated under the assumption that all walls of the dryer are fixed. Consequently, these conditions were meticulously applied to the momentum calculations of the walls.

In terms of materials, we selected glass, characterized by its semi-transparent surface properties, with a specified thickness of 6mm. This choice enabled us to accurately define and select specific values for both absorptivity and transmissivity, which were set at 0.05 and 0.95 respectively, ensuring that our simulation accurately reflects the physical behavior of the system under study (Figure 8).

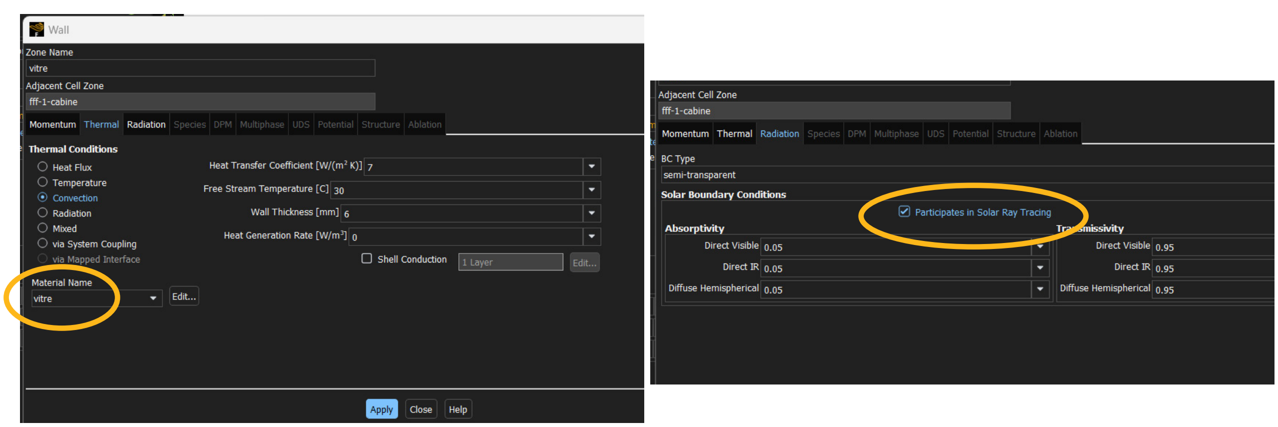

Figure 8: Wall definition for glass

6.4 Walls: Absorber

We utilized copper as the material (opaque surface) with a thickness of 2mm, allowing for the setting of absorptivity 0.9. All other walls such as those of the drying chamber, supports, etc., are considered adiabatic and do not participate to Solar Ray Tracing (Figure 9).

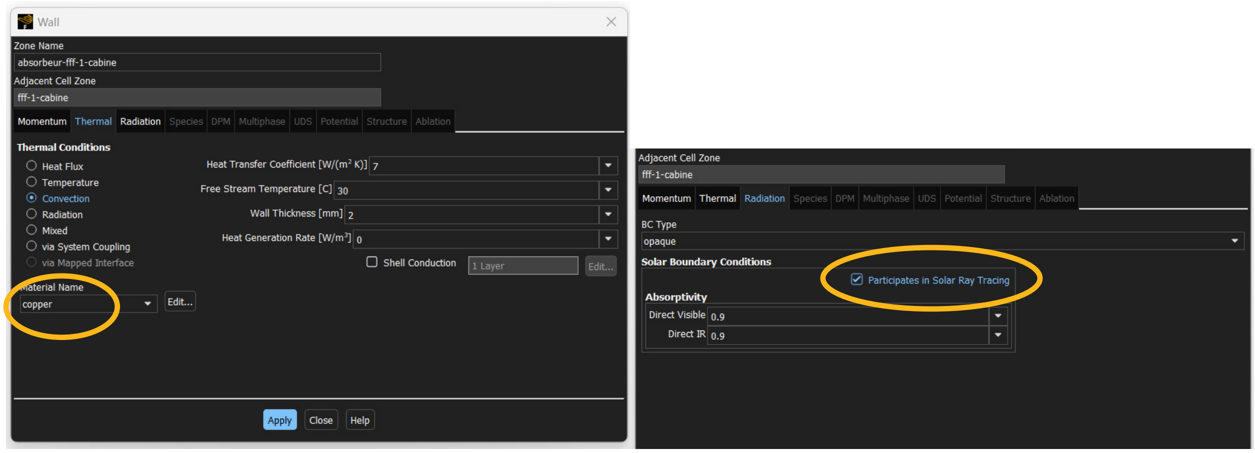

Figure 9: Wall definition for the absorber

7. Results

In the course of our investigation, we executed the simulation under steady-state conditions, and convergence was successfully achieved following 630 iterations (Figure 10 and Figure 11).

Building upon this work, it is feasible to conduct the simulation under unsteady-state conditions, which will allow for the application of the Piecewise linear model. Additionally, this approach enables the utilization of thermal flux values derived from meteorological data corresponding to various times throughout the day.

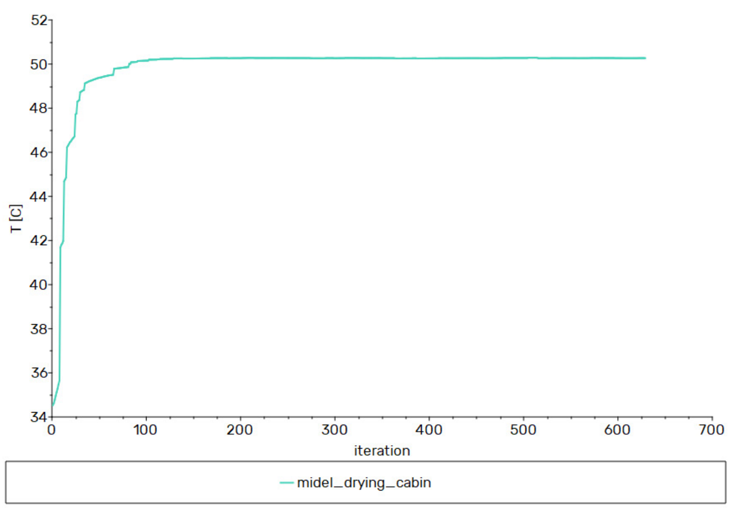

Figure 10: Convergence plot for the simulation

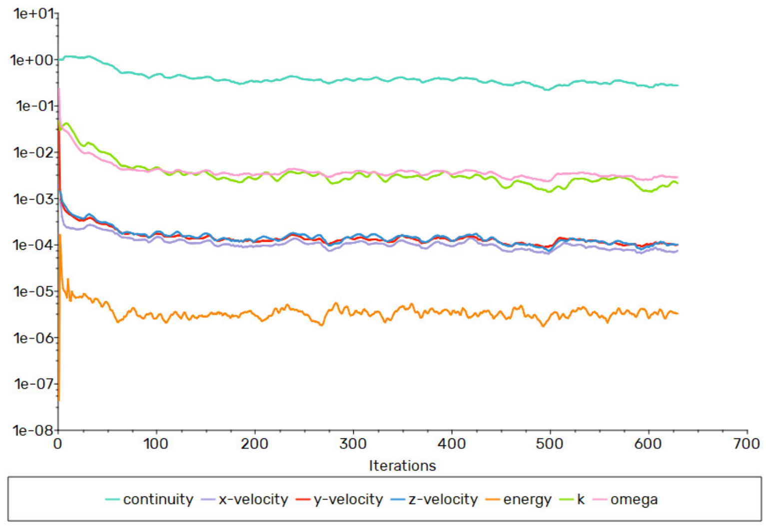

Figure 11: Scaled residuals

After conducting a comprehensive optimization study on the physical and optical parameters of the glass and absorber, it was determined that the optimal thicknesses are respectively 6 mm for the glass and 2 mm for the absorber. Following this analysis, we performed various simulations by altering the air velocities at the dryer inlet. It was observed that a velocity of 1.4 m/s is ideal for achieving efficient air distribution within the drying cabin (Figure 12). At this velocity, the air blade in the absorber has sufficient time to effectively warm up through convection, thanks to the thermal capture plate.

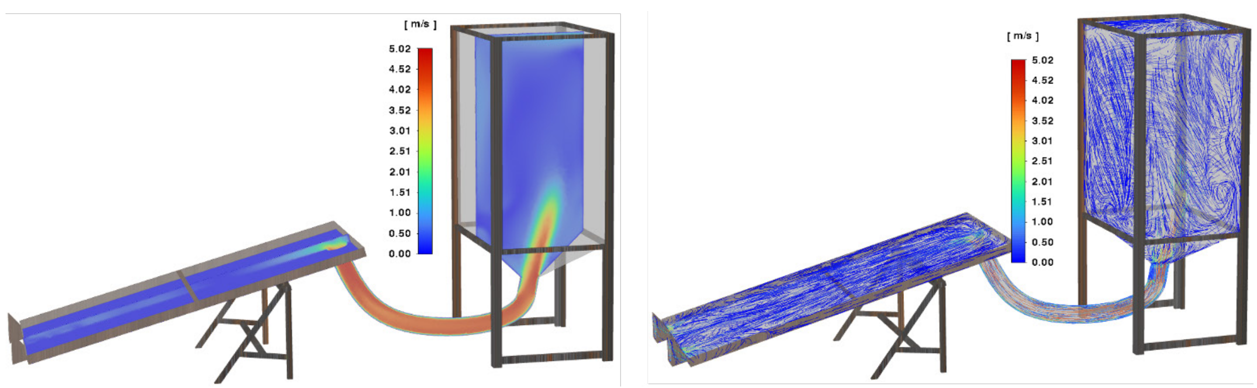

Figure 12: Evolution of Velocity Magnitude

This optimization of velocity and physical parameters of the dryer ensures a uniform and optimal temperature distribution, reaching 52°C within the drying chamber (Figure 13). This outcome is in perfect correlation with the data derived from the experimental study.

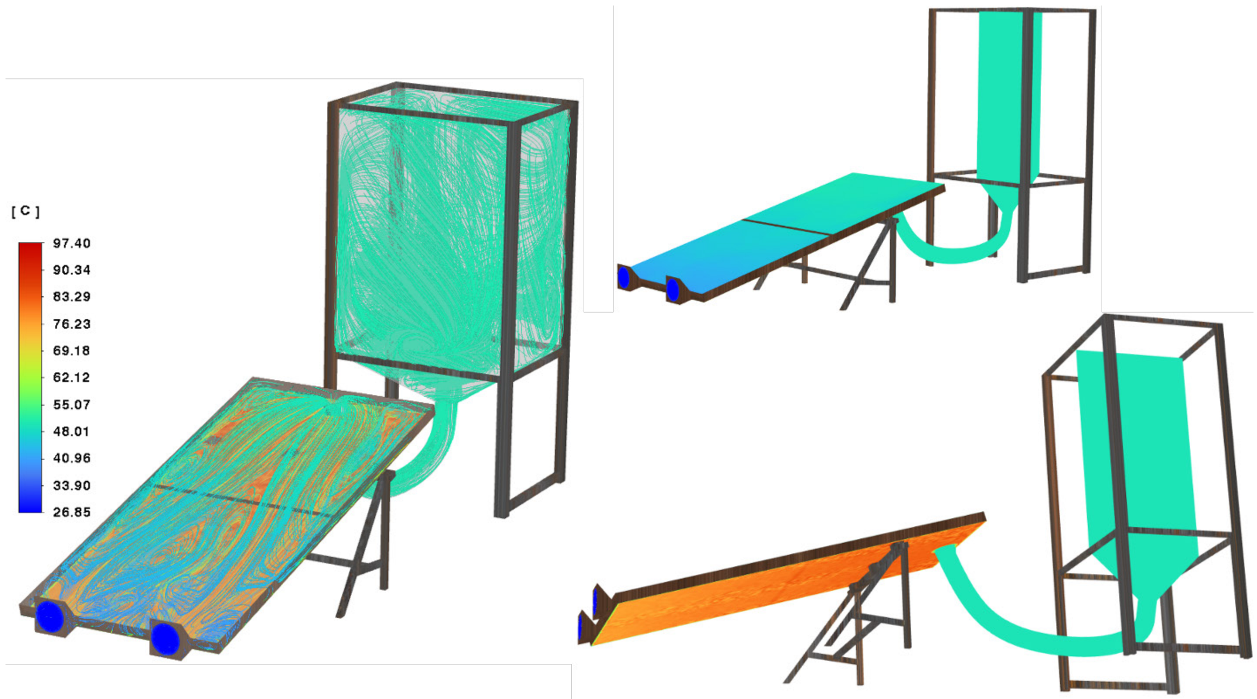

Figure 13: Evolution of Temperature

By precisely setting the geographic area where the simulation was conducted and correctly adjusting the dryer's orientation with a 15° tilt angle for the absorptive surface, we successfully replicated the experimental conditions (Figure 17). This configuration enabled us to achieve a maximum solar heat flux of 993.39, observed on the surface of the absorber (Figure 14).

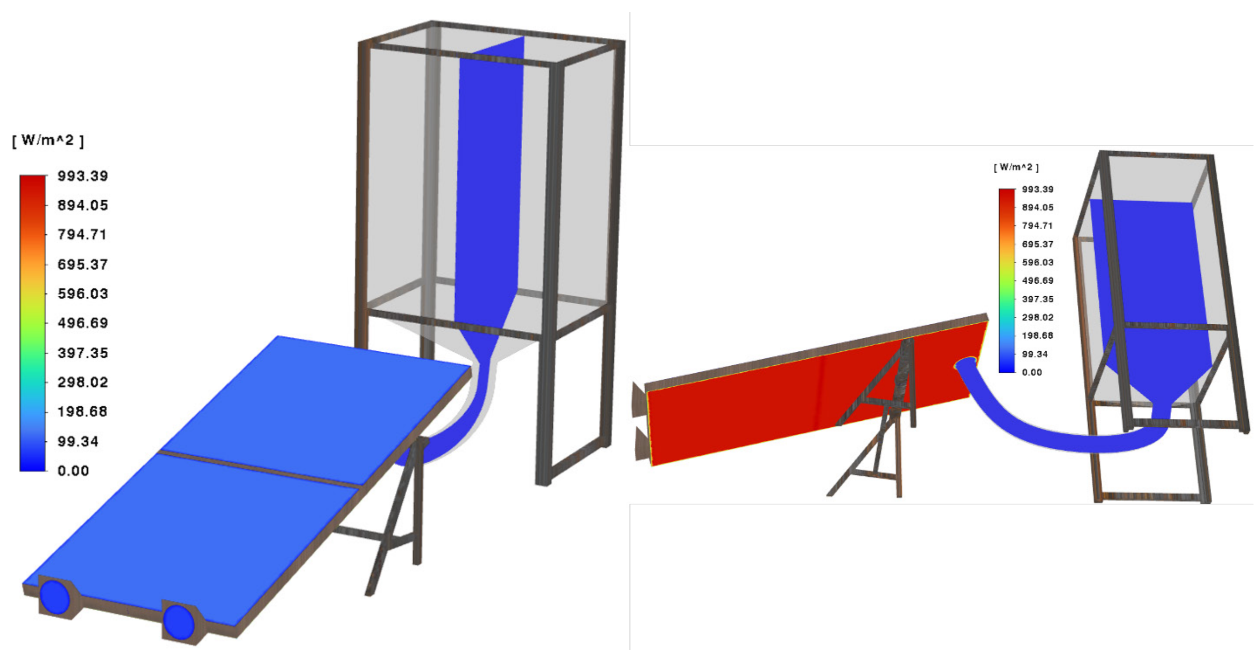

Figure 14: Solar heat flux distribution

To gain a deeper insight into the temperature dynamics within the dryer, we meticulously established points that mirror the exact positions of the temperature probes, as specified in the experimental setup. These points were strategically positioned at the entrance, middle, and exit zones of both the absorber (Figure 15) and the drying cabin (Figure 16). The temperature profiles derived from these points accurately depict the temperature variations within the dryer. As the air progresses through the absorber, a noticeable reduction in temperature fluctuations is observed. This stabilization in temperature variations can largely be attributed to the establishment of a uniform and stable flow regime, which effectively moderates and equalizes the temperatures throughout the internal environment of the dryer.

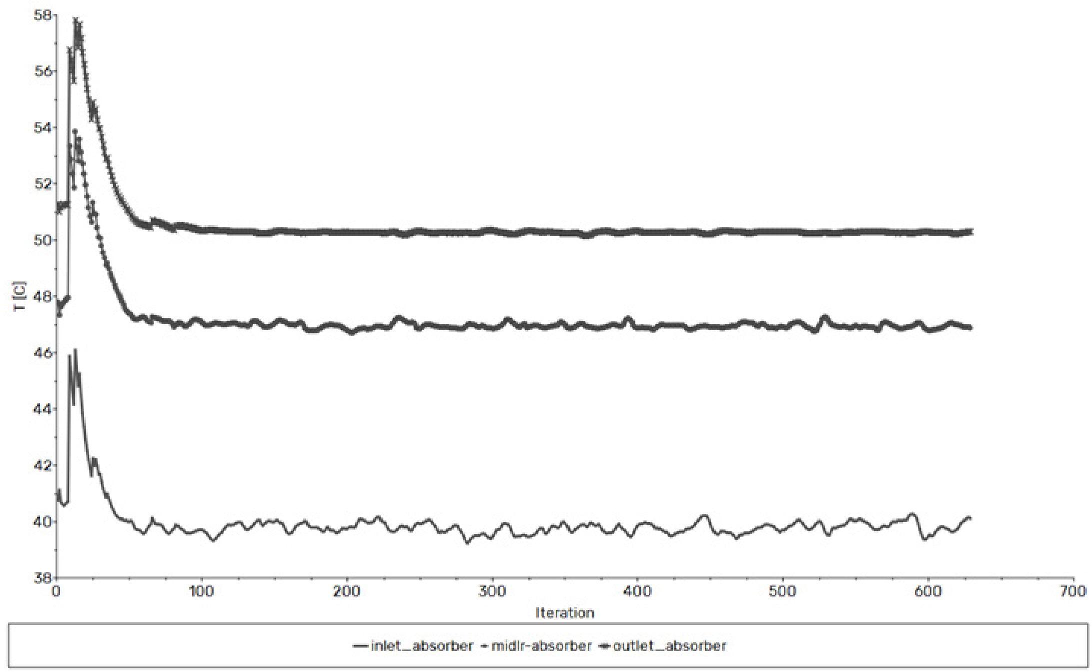

Figure 15: Temperature evolution inside absorber

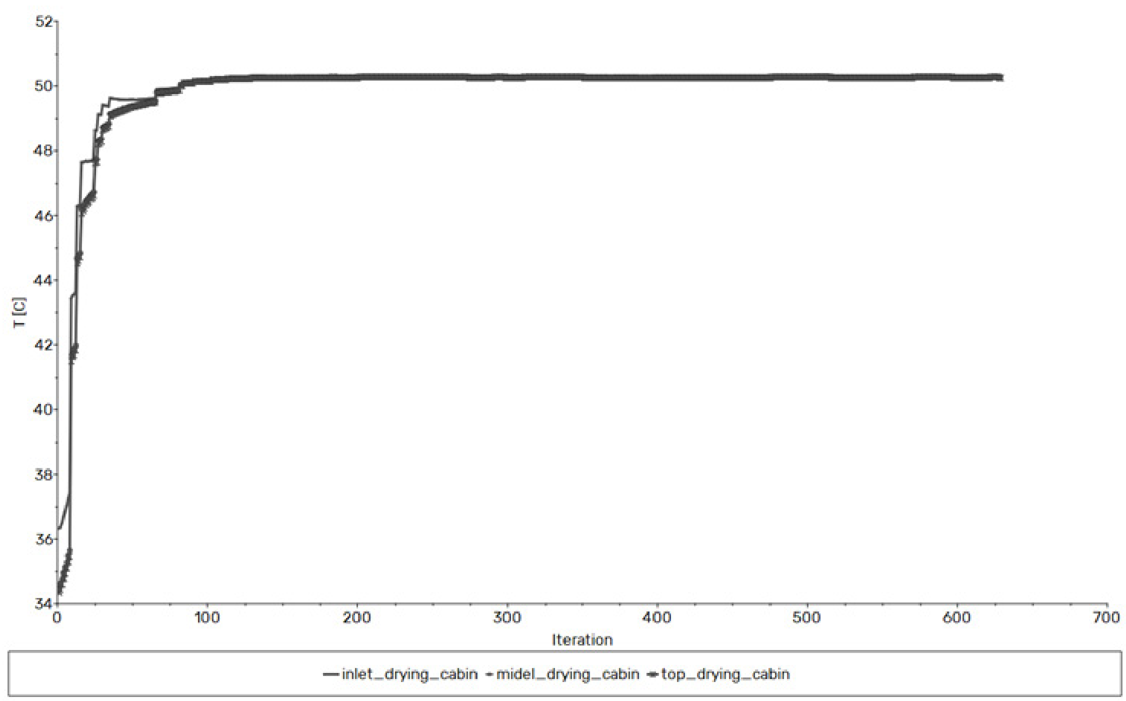

Figure 16: Temperature evolution inside the drying cabin

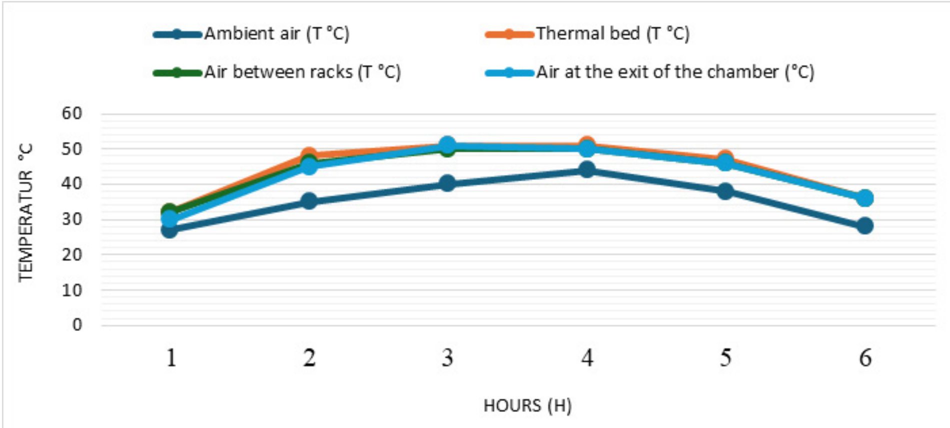

Figure 17: Temperature evolution from experimentation

8. Further Steps

This case study, which represents a simple numerical analysis of a vacuum solar dryer, yielded promising and satisfactory results. These findings pave the way for multiple avenues of research to be explored in the future. For instance, a deeper investigation into the dynamics of unsteady-state flow could be pursued, leveraging further functionalities available in Ansys Fluent software, such as the utilization of the Piecewise linear model. Incorporating this model would enable us to simulate variations in thermal radiation flux throughout the day, marking a significant advancement in solar drying process modeling. Moreover, we intend to broaden our scope by simulating the dryer loaded with food products to be dried, such as mango slices. This approach will allow us to delve deeper into our understanding of the drying process, considering the complex interactions between the heating fluid and the materials being dried. For example, we could explore phenomena like conjugate fluid-structure heat exchange, which are pivotal for realistic modeling of food drying in a solar environment. By combining these technical advancements with a thorough analysis of the dryer's performance under real-world conditions, we can gather valuable insights for future optimization of this solar drying system.

9. References

[1] P. Demissie; M. Hayelom; A. Kassaye; A. Hailesilassie; M. Gebrehiwot; M. Vanierschot: Design, development and CFD modeling of indirect solar food dryer; 10th International Conference on Applied Energy (ICAE2018), 22-25 August 2018, Hong Kong, China; *10.1016/j.egypro.2019.01.278*

[2] E. Getahun; M. A. Delele; N Gabbiye; S. W. Fanta; P. Demissie; M. Vanierschot: Importance of integrated CFD and product quality modeling of solar dryers for fruits and vegetables: A review Solar Energy 220 (2021) 88–110; *https://doi.org/10.1016/j.solener.2021.03.049*

[3] N. Tayyeb; J. Arezou; K. M. Hossein; A. Mortaza: CFD Simulation and Optimization of Factors Affecting the Performance of a Fluidized Bed Dryer; *Iran. J. Chem. Chem. Eng.; Vol. 32, No. 4, 2013 1021-9986/13/4/81*

[4] A. W. Deshmukh; M. N. Varma; C. K. Yoo; K. L. Wasewar: Investigation of Solar Drying of Ginger (Zingiber officinale): Emprical Modelling, Drying Characteristics, and Quality Study; Chinese Journal of Engineering Volume 2014, Article ID 305823, 7 pages *http://dx.doi.org/10.1155/2014/305823*

[5] Serigne Thiao, Awa Mar, Oumar Drame, Diouma Kobor, Issakha Youm Case Analysis of the Optical Parameters of Thermal Solarn Flat-Plate Collector on Heat Fluid Transfer and Pressure: International Journal of Sustainable and Green Energy 2022; 11(4): 75-83; *doi:10.11648/j.ijrse.20221104.1*

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

Here at Ansys, we rely on your feedback to ensure the educational content we create is up-to-date and fits your teaching needs.

Please click the link here out a short survey (~7 minutes) to help us continue to support academics around the world utilizing Ansys tools in the classroom.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com