

Simulation of Direct Solar Dryer with ANSYS

Authors : Boukhris Lahouari

Abstract : Simulation of solar dryers with ANSYS has revolutionized the way in which drying processes are optimized and analyzed in various industries. This advanced software allows engineers and researchers to simulate the behavior of a solar dryer under different conditions, helping to improve efficiency and reduce energy consumption. This work presents a numerical study of a direct solar dryer, which uses radiation and natural convection to dry agricultural products. The simulations were made in order to determine the dynamic and thermal fields under the influence of the variation in the size of the inlet and outlet opening. The conservation equations based on the standard k- ϵ turbulence model are solved by the finite volume method using the ANSYS-Fluent commercial code.

Keywords : solar dryer, CFD, solar radiation, natural convection, turbulent flow

Conference Title : ICFMTE 2024 : International Conference on Fluid Mechanics and Thermal Engineering

Conference Location : Paris, France

Conference Dates : November 18-19, 2024