

Computational Fluid Dynamics (CFD) Modeling of Local with a Hot Temperature in Sahara

Authors : Selma Bouasria, Mahi Abdelkader, Abbès Azzi, Herouz Keltoum

Abstract : This paper reports concept was used into the computational fluid dynamics (CFD) code cfx through user-defined functions to assess ventilation efficiency inside (forced-ventilation local). CFX is a simulation tool which uses powerful computer and applied mathematics, to model fluid flow situations for the prediction of heat, mass and momentum transfer and optimal design in various heat transfer and fluid flow processes to evaluate thermal comfort in a room ventilated (highly-glazed). The quality of the solutions obtained from CFD simulations is an effective tool for predicting the behavior and performance indoor thermo-aéraulique comfort.

Keywords : ventilation, thermal comfort, CFD, indoor environment, solar air heater

Conference Title : ICFMFA 2015 : International Conference on Fluid Mechanics and Flow Analysis

Conference Location : Paris, France

Conference Dates : July 20-21, 2015