Experimental and numerical study of a displacement ventilation diffuser coupled with a radiant floor

A. Fernandez-Gutierrez*, L .Parras†, I. Gonzalez-Prieto†, C. del Pino† and J.M Cejudo-Lopez*

In the last decade, there are lot of works dealing with the heating and cooling systems in large areas using displacement ventilation with radiant floors (e.g. big stores or airports). Chief attention is focussed on the measurements of the velocity [1] and temperature fields [2]. For the sake of simplicity, we present in this study the results of a scaled diffuser that uses displacement ventilation with radiant floor. Three parameters were changed: flow rate (Q), difference between the temperature of the atmosphere and the inlet cool air (ΔT_1) and the temperature between the floor and the inlet cool air (ΔT_2). Flow visualizations and measurements of the velocity field have been carried out for a laminar steady state flow. Furthermore, numerical simulations, as the temperature field depicted in Figure 1, show good agreement with the experimental data. Different areas were relevant in this study: diffuser exit, vicinity of the floor and the far field, where the boundary conditions are discussed in detail. The radial distance (r^*) at which each diffuser affects the surrounded area in terms of temperature gradient and velocity is an important parameter to design the spatial distribution of these diffusers. The main results of this work are based on the relation r^*=f(Q, ΔT_1, ΔT_2).

*Dep. Thermal Engines, ETSII, 20971, Málaga, Spain
†Dep. Fluid Mechanics, ETSII, 20971, Málaga, Spain
[1] Cehlin et al., *Int. J. of Ventilation* **4 (4)**, 349-364 (2006)
[2] Cehlin et al., *Building and Enviorement* **34**, 687-698 (2002)

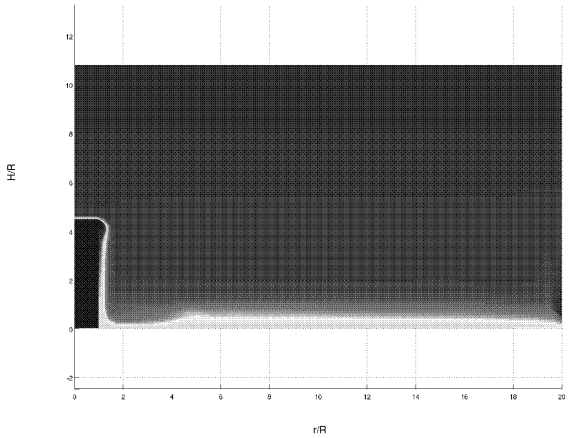

Figure 1: Cool radial jet from a diffuser discharging onto a atmosphere for Q= 20 l/min, ΔT_1= 8 K, and ΔT_2=3 K.