CALIBRATION OF A NUMERICAL JET FAN MODEL FOR SIMULATING SMOKE CONTROL IN UNDERGROUND CAR PARK

E. Didier*, B. Henriques*, R. Brás†

* DEMI, FCT-UNL
Campus de Caparica, 2829-516, Monte de Caparica, Portugal
e-mail: deric@fct.unl.pt; bm_henriques@hotmail.com

†France Air Portugal Lda
Av. Casal da Serra Lote I-4, Escr. 3, 2625-085, Póvoa de Santa Iria, Portugal
e-mail: Ricardo.bras@france-air.com

ABSTRACT

Jet fans, suspended below the ceiling, are actually currently used in enclosed car park in order to control the smoke flow in case of fire, i.e. forcing the smoke flow in direction of natural or mechanical outlets. This technique was originally used in road tunnels to control smoke and pollution and recently applied to car park [1]. The technique allows avoiding subdivisions of the space and simplifying the ventilation duct network. However, very complex flow patterns may be generated by the jet fans and the general shape of the car park and, so that, may produce the spread of the smoke.
Computational Fluid Dynamics – CFD – is a helpful tool to predict the expected flow patterns and to verify if spread of smoke occurs. However, in term of engineering applications, it is impossible simulating directly the jet fans. A solution consists developing a jet fan model, simple but representing the typical flow pattern produce by the fan. The model consists to a rectangular box, with the dimensions of the jet fan, and where boundary conditions are imposed to the upwind and downwind circular sections (figure 1). Classically, only the normal velocity at the fan face is considered (defined by the volume flow rate). In the present study, the jet fan model includes a tangential component of velocity imposed at the outflow face (swirl). A simple variation of this component is chose: the value of the tangential velocity is defined as a linear function of the distance from the center of the face, r, with a zero value at the center (r=0) and a maximum value when r=R, with R the rayon of the outflow face of the fan. Calibration of the jet fan is carried out by comparing velocity intensity obtained by numerical simulations with experimental data measured in the horizontal symmetric plane of the jet fan (figure 2). Numerical modeling is performed for an isothermal flow using the commercial code FLUENT of ANSYS.
Finally, a CFD application of smoke control is performed for an underground car park with a fire source of 4MW (corresponding to a small van in fire), using the present numerical jet fan. Comparison with a more classical jet fan (without swirl) is done.

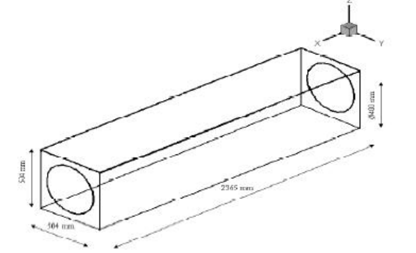

Fig 1. Numerical jet fan model: rectangular box with circular upwind and downwind section.

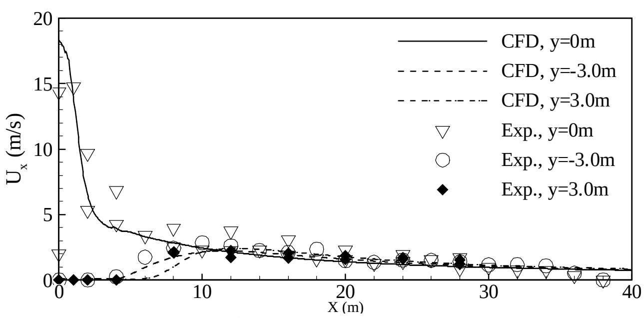

Fig 2. Calibration of the numerical jet fan model with experimental measurements.

References

[1] J.C. Viegas and J.G. Saraiva, CFD study of smoke control incide enclosed car parking. *Interflam 2001, 9th Int. FIRE Science & Engineering Conference*, Edinburgh: Interscience communications (2001).