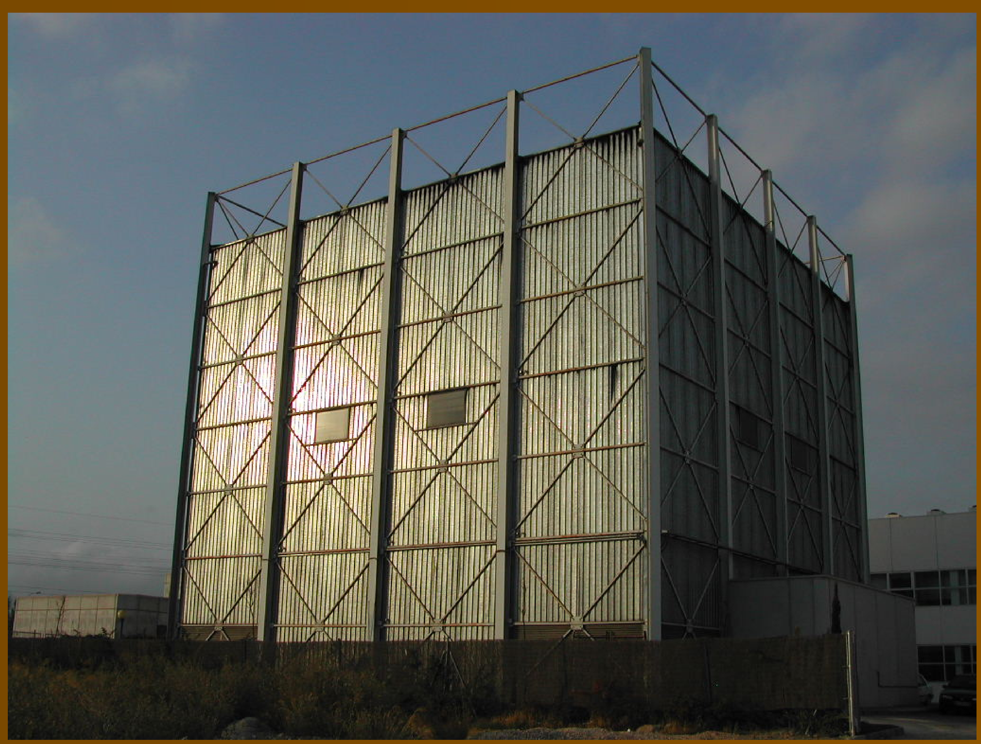
NUMERICAL AND EXPERIMENTAL STUDY OF ATRIUM ENCLOSURE FIRES IN A FULL SCALE FIRE TEST FACILITY

C. Gutiérrez-Montes*, E. Sanmiguel-Rojas, A. Viedma

Department of Thermal Engineering and Fluids. Area of Fluids Mechanics.
Antiguo Hospital de Marina, C/ Doctor Fleming s/n, 30202. Cartagena.
E-mails: candido.gutierrez@upct.es, enrique.sanmiguel@upct.es, antonio.viedma@upct.es
Technical University of Cartagena, Spain



"Fire Atrium" in Murcia, Spain.

Work Description

For the present work, a 3-D numerical model has been implemented to simulate the thermal and fluid fields induced by an enclosure fire in an atrium and for smoke exhaust system assessment. This study is focused on the 'Fire Atrium', a new full-scale fire test facility of the Technological Metal Centre in Murcia, Spain. It is an aluminium prismatic squared base building of 19.5 m x 19.5 m x 20 m with several vents arranged in its walls and four exhaust fans at the roof.

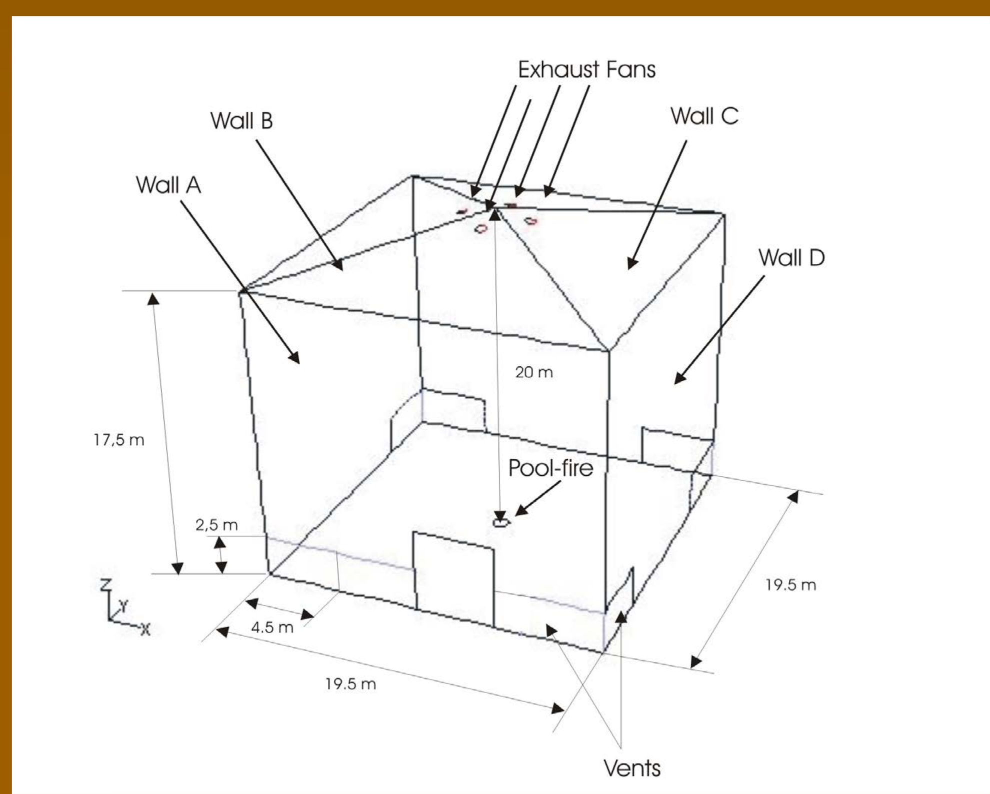
Many experimental tests have been carried out using several heptane normalized pool-fires placed at the centre of the atrium, being registered the inside air, wall and ceiling temperatures, the air through the vents inlet velocities and the exhaust fan pressure differences. The data obtained from these experiments have been used for the model validation.

Some numerical simulations have been performed. The results show good agreement with experimental data. For a fire test of 1.6 MW of average heat release power, the exhaust and ventilation system is not enough to extract the hot combustion products. There is an excessive and dangerous accumulation of hot gases at the upper part of the atrium and the exhaust capacity of the roof fan must be increased. Three different smoke exhaust systems have been compared showing that the CFD model can give the answer to that question. Also, for a 1.8 MW fire it has been predicted the smoke layer interface height showing good agreement with some other expressions from technical bibliography.

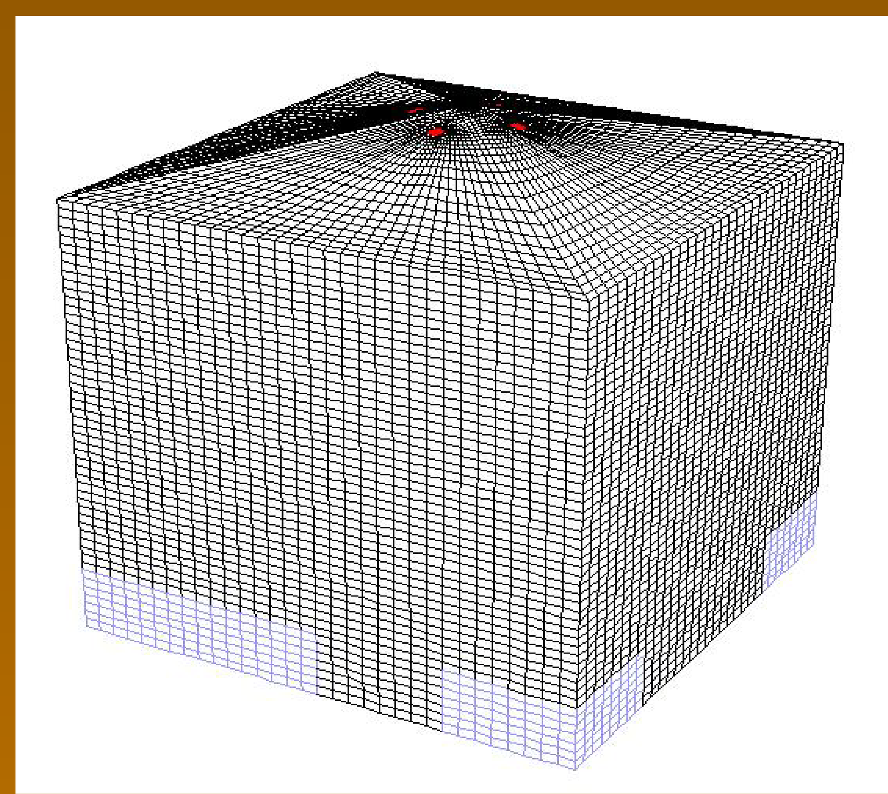
Finally, many different ways of simulating the pool-fire are being studied in order to get an accurate and relative low computational cost fire model.

Numerical model

Mesh → 3D quasi-structured non-uniform mesh
(of the entire domain and a quarter of the domain)
Frame material → Walls and roof: Aluminium
Ground: Concrete
Discretization schemes: SIMPLE
PRESTO
QUICK \ UPWIND
Radiation model → P1 Model
Turbulence model → k- ε Standard



Fire Atrium scheme



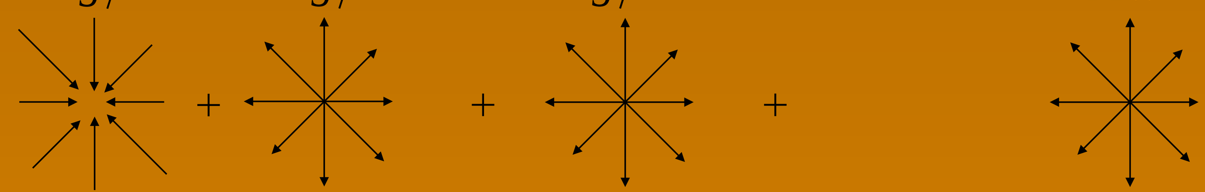
Computation domain

Ways for simulating the fire

Without combustion: → Complete combustion products injection (sources), oxygen suction (sink) and heat power source →

$$O_2 + CO_2 + H_2O + Heat\,release\,rate$$

kg/s   kg/s   kg/s   MW



Less computational cost
Less accuracy near the fire

With combustion:
(heptane injection)
More computational cost
Higher accuracy near the fire

Complete non-premixed combustion model (species transport eddy dissipation model) →

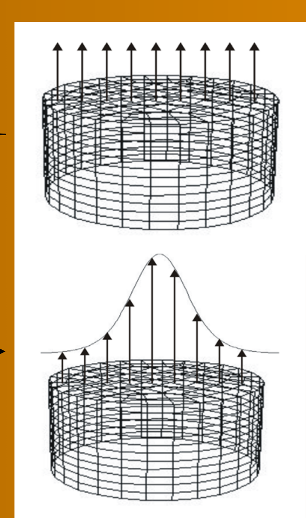
$$C_7H_{16}\left(kg/s\right) + O_2 \rightarrow CO_2 + H_2O$$

Incomplete non-premixed combustion model (mixture fraction / PDF model) →

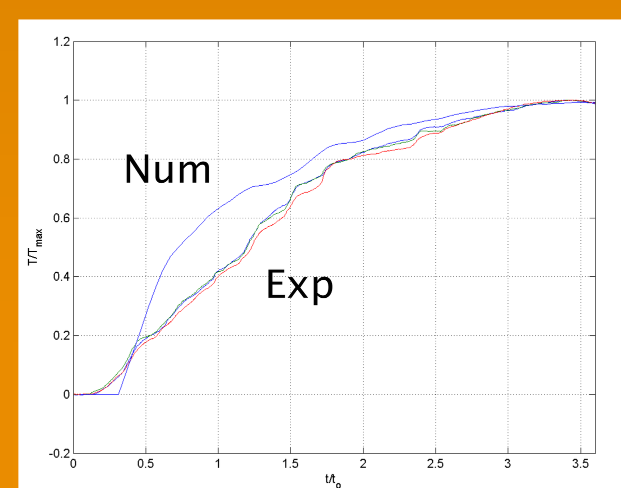
$$C_7H_{16}\left(kg/s\right) + O_2 \rightarrow CO_2 + CO + C + \ldots$$

$$\ldots + H_2O + H + H_2 + O_2 + O + OH$$

State:
Steady State → Average mass burning rate injection
Unsteady State → Constant average mass burning rate injection
Variable mass burning rate injection as a function of *r* and *t* →



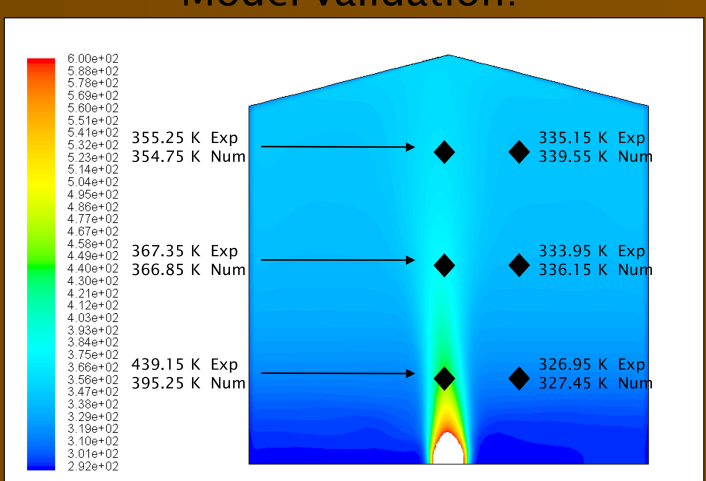
$$v_{inj} = f\left(r,t\right)?$$



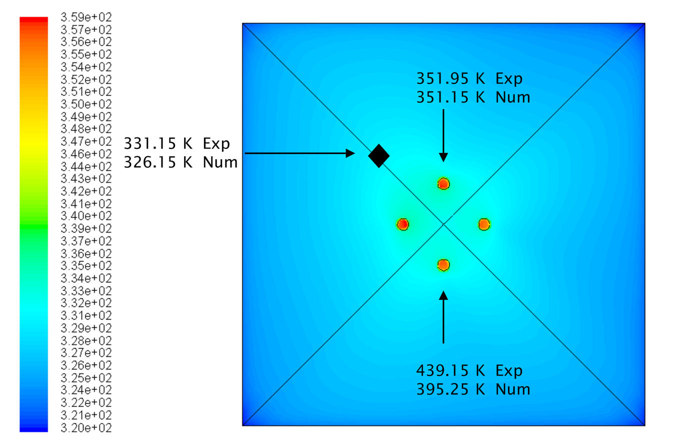
Fire tests simulations (Pool-fire D=0.92 m)

1.6 MW

Model validation:

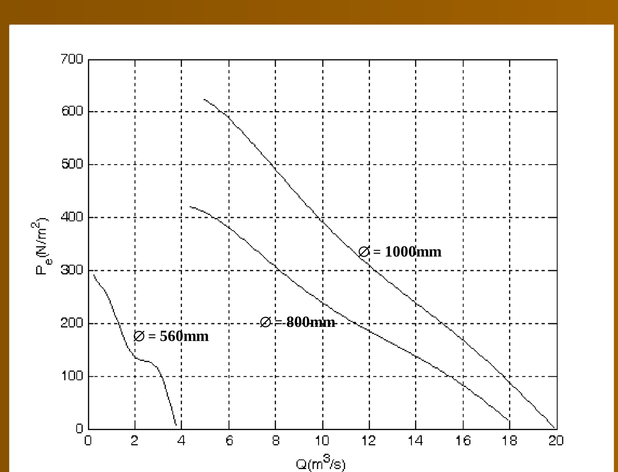


Flow temperature at a central section



Flow temperature through the exhaust fans and roof temperature

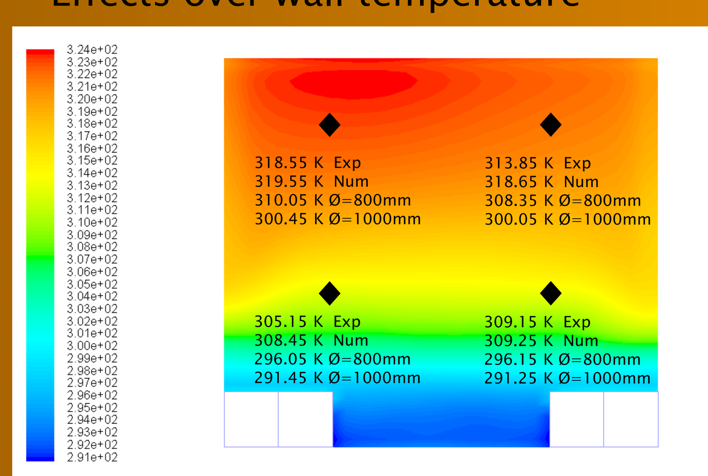
CFD application for fire safety design:
alternative smoke exhaust system study



Effects over smoke exhaust mass flow

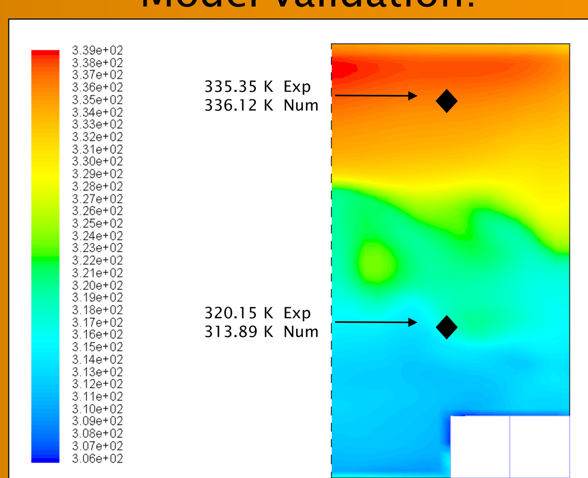
	Experimental	Numerical		
	Ø = 560mm	Ø = 560mm	Ø = 800mm	Ø = 1000mm
Exhaust fan 1	3.7	3.2	7.8	14
Exhaust fan 2	-	3.2	7.8	14
Exhaust fan 3	3.8	3.3	7.8	14
Exhaust fan 4	-	3.2	7.8	14

Effects over wall temperature

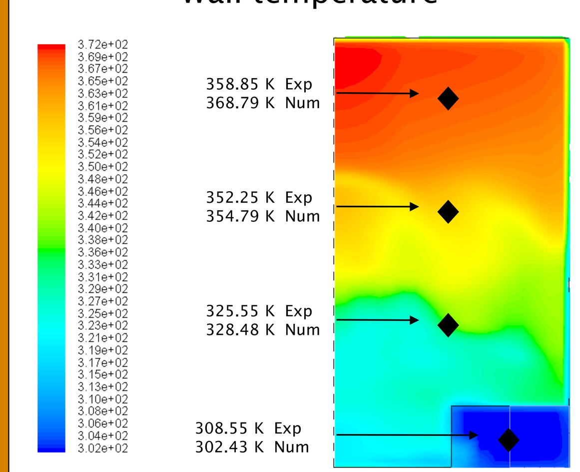


1.8 MW

Model validation:



Wall temperature



Air next to wall temperature

CFD application for fire safety design:
Smoke layer interface height prediction at two vertical lines