September 6, 2010

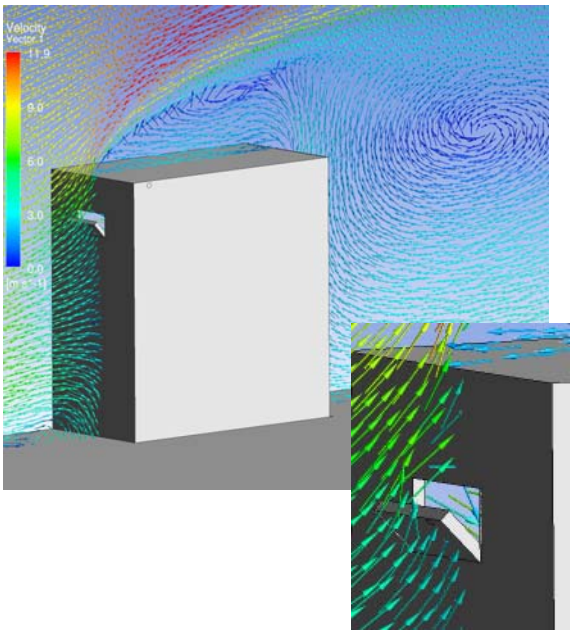
CFD-based design of natural smoke ventilation — Simulating the effect of external wind

Assessment of Fire Protection Measures

There are a number of options available when designing thermal smoke ventilation. The normal procedure when using analytical design, rather than rules of thumb, is to calculate a number of fire scenarios in the building, using more or less advanced methods ranging from easy, back-of-the-envelope calculations or some computer aided simulation tool. The scenarios evaluated are often very limited in number and external effects such as wind is ignored. Can todays easy-to-use CFD software written to simulate fire scenarios be used in a larger context, including large scale wind effects and at the same time predicting fire in a building? If so, given enough computer resources it seems that designers of smoke ventilation is using an overly crude methodology, neglecting vital parameters, and for no real reason.

The intention has been a systematic approach in verifying the capabilities of a number of different computational tools. As a rule, only free software has been included since the commercial CFD packages has been judged less used among fire safety engineers and too costly for use in simple fire engineering investigations. The chosen CFD programs has initially been evaluated by comparing computational results with experimental data from a wind exposed cube and a simple well-ventilated, single room, fire scenario. These problems are then combined using an experimental setup with a wind-exposed smoke ventilator mounted on the roof of a structure with a small pool fire below.

Two fire-specific CFD programs are included, FDS and Sofie. Furthermore, a number of applications within the open-source project OpenFOAM have been evaluated for different scenarios. Given the results from predictions compared with experimental data it is clear that, at least for rectangular geometries, the fire simulation software does very well also for general flow simulations as well as the combined simulation with external wind and buoyancy-driven flow. When dealing with complex, non-rectangular, problems it may be suitable to use applications employing unstructured meshing, such as OpenFOAM. The models within this program however, does not excel in fire simulations.



The report gives a number of recommendations when considering the effect of wind on the design of smoke ventilation.

Report

Read more in the report ”CFD-baserad dimensionering av brandgasventilation—Simulering av vindtryckets inverkan på termisk brandgasventilation” . The report can be downloaded from www.brandforsk.nu, project number 100-071.

Contact information

Jörgen Carlsson, e-mail: jorgen.carlsson@afconsult.com

BRANDFORSK
The Swedish Fire Research Board