Comparison between CFD-simulations and large scale experiments

ir. Xavier Deckers

Supervisors: prof. dr. ir. Bart Merci, ir. Nele Tilley, dr. ir. Pieter Rauwoens

***Abstract*: This article tries to understand the impact of different physical submodels, as used by fire safety engineers, on the accuracy of numerical simulations. Blind comparisons were made with results from 3 large scale experiments. Several CFD-packages were used in this study.**

***Keywords*: Fire Safety Engineering, CFD, Fire Modeling, Validation**

Introduction

As there is a clear trend towards more performance-based approaches in fire protection engineering regulation, CFD-models are increasingly being used as a tool in the fire safety design. In setting up the model many choices and assumptions have to be made, but the impact of these on the accuracy of the results are not always clearly understood. This article focuses on the use of different physical submodels (combustion, turbulence, radiation). Three scenarios (an atrium, different rooms with a corridor, and a tunnel) are modelled with different CFD-packages (FDS 5[1], SMARTFIRE[2] and FLUENT[3]) and compared with the experimental results after the simulations were carried out to make sure these are blind simulations.

Small experiment

In order to better understand the process of how the information is gathered from experiments, a small experiment was set up and compared with numerical simulations. The experiment concerned an ISO 9705 room corner configuration on scale 1:3 with a propane burner of 11 kW in the middle of the room and a thermocouple three. Several parameters (place burner, HRR, burner surface and door opening width) were analysed. The predicted gas temperatures from the numerical simulations were in good agreement with the experiments.

Scenario 1: atrium configuration

The first scenario concerns a fire with constant HRR (1300 kW) in the middle of a large atrium (24 x 30 x 26.3 m³). Table 1 shows the temperature evolution in the middle of the atrium for the different CFD-models. The influence of the turbulence model (RANS and LES) can clearly be seen. The assumption of modeling only the convective HRR and considering the walls as adiabatic lead to an overestimation of the gas temperatures and thus an overestimation of the smoke free height (fluent-model). The second model gave the best results. The third model, using 2 symmetry planes and an enthalpy source also gave good results. The scenario was also solved by a two-zone model where the importance of choosing appropriate boundary conditions in accordance to the large geometry was shown.

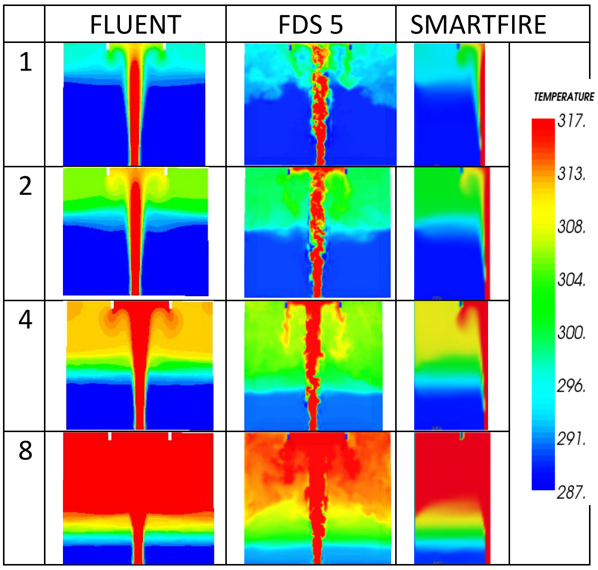

Table 1: Temperature in the middle of the atrium (cross section) after 1, 2, 4 and 8 minutes.

X. Deckers is a Project Manager at IFSET, Belgium.
E-mail: xd@ifset.com .

Scenario 2: multiple rooms and corridor

In the second scenario the smokespread and gas temperatures in multiple rooms with a corridor were predicted in good agreement with the experiment. The importance of the chosen grid was clearly shown in the FDS model by comparing the velocities from the experiment at the door where a fine grid was used and at a door with a coarser grid. The finer grid predicted the velocities accurately but the coarser grid lead to underestimations of the velocities. The simulation with SMARTFIRE slightly over-predicted the gas temperature, probably due to the use of a constant specific heat.

Scenario 3: tunnel with natural ventilation

The naturally ventilated road tunnel is 853 m long, has an inclination of 3.2% and a fire source of nominally 20 MW. With FDS 5.1 a series of parallel calculations were performed with different grid sizes, inclinations, impact of submodels (baroclinic generation of vorticity), boundary conditions. Unfortunately different results were found between the serial and parallel computations. We would like to stress the importance of checking that parallel calculations give the same results as serial calculations.

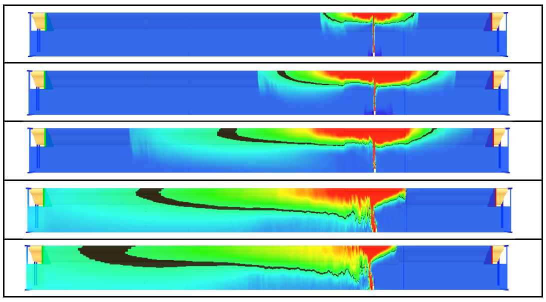

Table 2: Temperature evolution after 1, 2, 4, 12, 18 minutes with FDS (radiation model)

Table 2 shows the temperature evolution in the inclined tunnel as predicted by a serial computation with FDS. Blue stands for 41F and red for 200F or higher. The 100F temperature contour is showed in black. The destratification on distances far from the fire were not predicted accurately due to the coarse grid.

Comparing a model with a radiation submodel and a model with only the convective HRR showed similar results but different backlayering distances.

Comparing a model with a combustion model and a model with a volumetric heat source showed similar results although the heat losses to the walls were overestimated in the latter model due to a poor choice of boundary conditions by the modeler.

Conclusion

The different scenarios have given good results for the different physical submodels. The importance of choosing the right boundary conditions was shown to be really critical. Simple zone-models and hand calculations can be of help here. By choosing appropriate physical submodels in combination with a fine enough grid, good results can be expected.

When performing parallel calculations, the modeller has to check that parallel and serial calculations give the same results.

The importance of blind validation was shown in this study. To be able to compare the submodels on a more quantitative basis, a grid sensitivity study needs to be performed. The uncertainty of the measurements needs to be taken into account in order to quantitatively discuss the accuracy of the models.

Acknowledgements

The author would like to acknowledge the Large Scale Facility[5] in Lund for the use of their laboratory for the experiment, for the use of the computational facilities for the parallel calculations and Professor P. Van Hees and U. Göransson for their technical suggestions.

References

[1] http://www.fire.nist.gov/fds/
[2] http://fseg.gre.ac.uk/
[3] http://www.ansys.com/
[4] *European Research Infrastructure,* http://spartacus.forbrf.lth.se/lsf/