TOWARDS FIRE DYNAMICS SIMULATION IN HELYX

DANIEL DEISING[1], SALVATORE RENDA[2], EUGENE DE VILLIERS[3]
[1]*Engys GmbH, d.deising@engys.com*
[2]*Engys Ltd, s.renda@engys.com*
[3]*Engys Ltd, e.devilliers@engys.com*

Keywords: *HELYX, Fire Dynamics Simulator, Fire Modelling, HVAC*

Introduction

Over the last decade, the use of CFD methods for fire safety design has received increasing attention from the Architecture, Engineering and Construction (AEC) industry, leading to an urgent demand for accurate and efficient tools, capable to easily handle complex geometries and physics. The key ingredients for a fire simulation framework are mainly the capability of modelling in a simplified fashion the flame and smoke propagation through buildings, and a set of components such as water sprinklers for fire suppression, ventilation units, extractors and doors, whose operating conditions can be modified by a sensor logic, based on external or internal inputs, e.g. temperature at monitoring location, visibility of the area etc.
Nowadays, the standard tool for fire analysis in industry is the open-source Fire Dynamics Simulator (FDS) software, developed by the National Institute of Standards and Technology (NIST) and the VTT Technical Research Centre of Finland. Despite FDS being a widely used, well-established and validated code, to the best knowledge of the authors it exhibits several drawbacks:

- the use of simple cartesian meshes which are incapable to capture the complex geometric features of buildings
- an inherently transient solver with no possibility to run steady state analyses
- poor scalability and a non-user-friendly pre- and post-processing environment.

Furthermore, FDS is a stand-alone simulation platform, which means it cannot be easily integrated with other CFD tools and/or workflows generally used by the AEC industry. On the other hand, other commercial codes can today still be used to cope with fire-driven flow applications, however these are almost exclusively available as close source software.
Motivated by the limitations mentioned above, by the fact that HELYX already constitutes a successful framework in the ACE industry and because integrating new features for fire analysis is relatively straightforward in the existing code, we decided to carry out a research and development project on this topic which is in high demand.

In this work, we present an innovative fire dynamics modelling framework, designed for the solution of fire security problems and the development of fire control strategies. Due to the large scale of the geometries, which often represent floors, carparks or even entire buildings, the use of detailed physical models and numerical techniques is infeasible. Therefore, our proposed method is based on detail-reduced modelling approaches, i.e. averaging, loose coupling between smoke and air and a certain number of model simplifications. We remark that the proposed framework is not intended to be a replacement of other fire solvers already included into OpenFOAM such as FireFOAM, it is rather to be considered as an additional simplified modelling tool for large-scale fire dynamics simulations.

Modelling strategy

Main modelling components for a fire dynamics environment are:

- Fire source
- Smoke and visibility index
- Coolant spray
- Air humidity
- Sensor logic
- Fans
- Doors and windows

The simulation framework is based on a pressure-based, compressible, single-specie, buoyant thermal solver. Turbulence is modelled by means of RANS equations plus a two-equation turbulence model. Support for DES and LES is also included. All the heat transfer mechanisms, i.e. conduction, natural and forced convection and radiation are taken into account. Specifically, the radiation component is modelled by means of an improved version of the discrete order (DO) method, along with a constant absorption-emission model to include the interaction of radiation with fumes and air. This last aspect was found to be crucial for the accuracy of the plume development and shape. Furthermore, two additional transport equations are solved to model the smoke and humidity concentration transport. The solver can run both as steady-state or transient.

Within this modelling approach, smoke, coolant (water) and humidity are only loosely coupled to the primary air phase [1]. Smoke and humidity are solely acting as passive scalars regarding the hydrodynamics, while the coolant spray affects the air via the standard momentum exchange terms [3]. Moreover, thermal and mass coupling between air humidity, coolant spray and the primary air phase is accounted for. Within the context of fire simulation, the coupling of coolant mass and air humidity due to thermal phase change is modelled via simplified closures. The air temperature is affected by heat transfer from the coolant as well as phase change due to evaporation. Unlike common practice in other tools, the coolant spray is herein simulated as an Eulerian phase, as are all other participating phases. The fire itself is modelled as time-dependent heat and smoke release from boundary patches or sub-domains. To enable the simulation of realistic fire scenarios with fire suppression systems, the modelling of opening and closing doors, as well as the activation and deactivation of sprinklers and fans, are required. Specifically, we propose to model the opening and closing of doors by switching between boundary conditions, while fans are modelled as momentum sources with prescribed target velocity or fan curve. The control of the fire suppression systems and doors status is coordinated by means of the sensor logic which monitors solution parameter such has flow temperature and visibility index and makes decisions based on these.

Implementation

The presented modelling approach is accomplished via a newly developed class of solverObjects. These solvers are registered mesh objects for which wrappers for functionObjects and fvOptions as well as direct access functions within solvers have been created. The presented implementation utilizes the capabilities of fvOptions and adds hooks for the solverObjects to each of the fvOptions function call, which allows automatic execution of a solverObject at all fvOption calls within the top-level solver. Thus, the user can control at which stage of the top-level solver passive scalar transport equations or other solverObjects are executed. Within this framework, passive scalar transport equations can be modified to interact with the solver - and thus becoming active scalars - without modifying the source code, only by case setup. This is made possible by allowing the solverObject in turn to access any fvOption. Thus, solverObject and top-level solver or even different solverObjects can directly interact by fvOptions, e.g. source terms.
This functionality is extensively used to compute different transport quantities on demand, whenever fvOption-based source terms which need these quantities are evaluated. The primary goal of this framework is to introduce a new level of user flexibility in manipulating and adding new functionality to existing solvers without having to resort to code customisation.
Other user-interactions, like the switching of boundary conditions (e.g. opening and closing of doors) or activation and deactivation of sprinklers, are enabled via a new sensor class which allows users to perform algebraic and logical operations on fields, coupled with the available Function1 as a lookup table for the sensor output value. With this functionality, the user obtains a large amount of control over the setup of dynamically changing cases. Currently, this functionality is only available within a generic boundary condition that allows the switching between two user-specified boundary conditions based on sensor operations and the Function1 lookup table.

Results

Initial results for two test cases, a sprinkler water jet and flame development in a building, are shown to prove the capabilities of the proposed modelling framework.

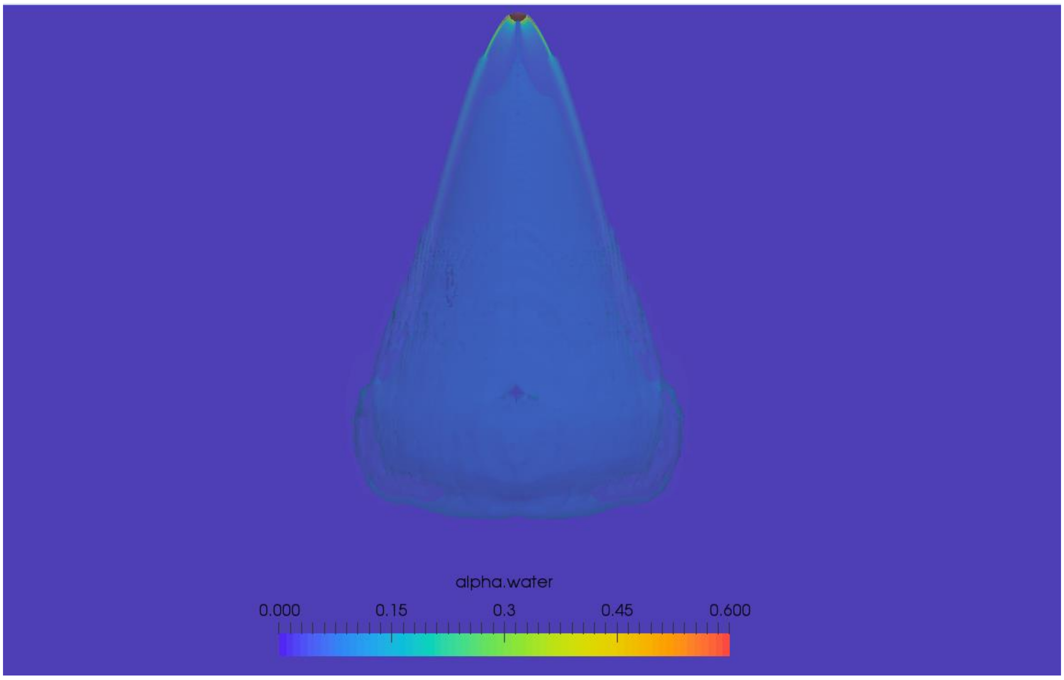

Figure 1. Water sprinkler modelled with uncoupled Eulerian spray.

Figure 1 shows contours of the water volume fraction for the sprinkler case. At the top, the sprinkler can be seen, coloured in red and modelled as a sphere sector. The aim of this test case was the verification of the droplet spray path and the cooling of the air due to heat transfer and evaporation.

Figure 2 presents the contours of the temperature field of the second test case. The case aims to reproduce experiments carried out at the Murcia fire test facility [2]. In this test case, which is specified in [2], a fire is modelled as a heat source originated in a pool located at the centre of the atrium. In the picture, the plume formation and development, with air stratification at different temperatures is displayed.

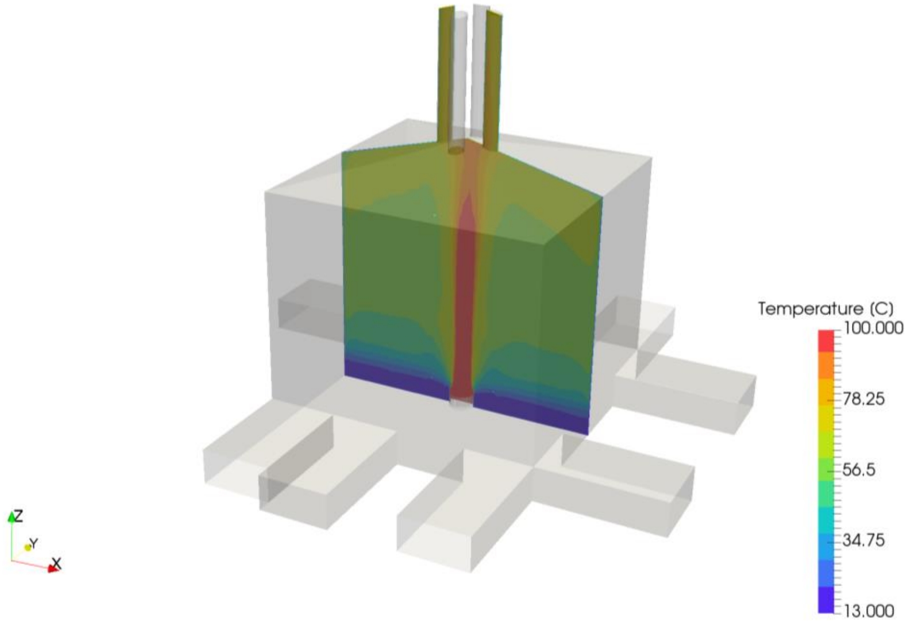

Figure 2 Temperature plume developed for the Mercia facility case

Acknowledgements

The authors thank all those involved in the organisation of OFW13 and all the contributors that will enrich this event.

References

[1] Kubilay A, Derome D, Blocken B, Carmeliet J. Wind-driven rain on two parallel wide buildings: field measurements and CFD simulations. Journal of Wind Engineering & Industrial Aerodynamics; 146:11-28, 2015

[2] Gutierrez-Montes C, Sanmiguel-Rojas E, Viedma A, Rein G, Experimental Data and Numerical Modelling of 1.3 and 2.3 MW Fires in a 20 m Cubic Atrium, Building and Environment 44, pp. 1827-1839, 2009.

[3] Wörner M, A compact introduction to the Numerical Modeling of Multiphase Flows, FZKA 6932, 2003.