Cooling of a smoke layer by a sprinkler spray

Author
ir. N.A.J. (Nick) Tenbült

Supervisors
Prof. ir. W. (Wim) Zeiler
R.A.P (Ruud) van Herpen Msc, FIFireE
ir. N.J. (Jur) van Oerle

INTRODUCTION

CFD models are increasingly used in building design. In the context of fire safety engineering CFD models have proven to be useful for the prediction of smoke transport in buildings in case of a fire. In a fire scenario where sprinklers are activated the temperature and movement of the smoke layer is subject to the sprinkler spray. Fire suppression by a sprinkler spray can be distinguished in three regions, namely the interference of water droplets with the fire plume (flame), smoke plume and smoke layer. This graduation project focused on the interference with the smoke layer.

By spraying water directly into a smoke layer it may cause diffusing and descending of the smoke. This phenomenon is called smoke-logging and was introduced by Bullen in 1974. According to Bullen, the stability of the smoke layer depends on the ratio between the drag force (D) and buoyancy force (B) on the smoke layer. Smoke logging will occur when D>B, otherwise the smoke layer will remain stable [1]. Smoke logging can potentially result in a decrease of the efficiency of a smoke extraction system [2] and compromised egress routes.

In the past, the effects of water droplets on a smoke layer has been studied with numerical models and experiments. The volumetric flow rate of smoke going upwards decreases under sprinkler spray due to the cooling effect of the water droplets [3]. So far, numerical simulations are performed with an evenly distributed water mass and velocity within the spray envelope. However, separate studies by Sheppard and van Venrooij indicate irregular water droplet distributions within the spray envelope for both elevation angle and azimuth angle, which is strongly dependent on the nozzle's geometry [4], [5]. Further development of the CFD-models and more experimental data is required to validate the CFD-models.

The Fire Dynamics Simulator (FDS, v6.6.0) is developed to model low-speed, thermally-driven flows with an emphasis on smoke and heat transport caused by fires unlike other CFD software packages such as ANSYS Fluent and Phoenics [6]. However, the results obtained from this simulations need to be treated carefully since the reliability of the outcome can be uncertain. A careful validation is necessary before applying the results to (non-)academic engineering problems.

The main research objective of this study is to gain insight in the cooling effects of a sprinkler spray on a smoke layer. Subordinate to the main objective numerical simulations in FDS are attempted to be validated by acquiring experimental data. In addition, the study aims to gain insight into the influence of different sprinkler spray patterns by varying the water flow rate.

METHODOLOGY

EXPERIMENTS

The experimental set-up (Figure 1) is based on earlier conducted studies with similar research objectives [3, 7, 8]. The set-up consists of two connected cabinets. In the combustion cabinet smoke is generated by a fire. The smoke flows into the smoke cabinet where a smoke layer is formed. The smoke is extracted by a mechanical fan. In the exhaust duct the smoke is analysed to determine the heat release rate (HRR) of the fire. When a stable smoke layer is formed the sprinkler spray is activated to cool the smoke layer.

Smoke and heat are generated by pool fires. In total nine experiments were performed, with heptane as a fuel, for two different pool sizes (0.25 m2 and 0.35 m2).

In the middle of smoke cabinet, a pendent sprinkler nozzle is placed at a height of 2.9 m. The sprinkler with an orifice diameter of 11.1 mm has a K-factor of 80.6 L/min√bar and a 25 mm deflector plate diameter. The sprinkler spray is activated manually and the operating pressure at the sprinkler nozzle is controlled by

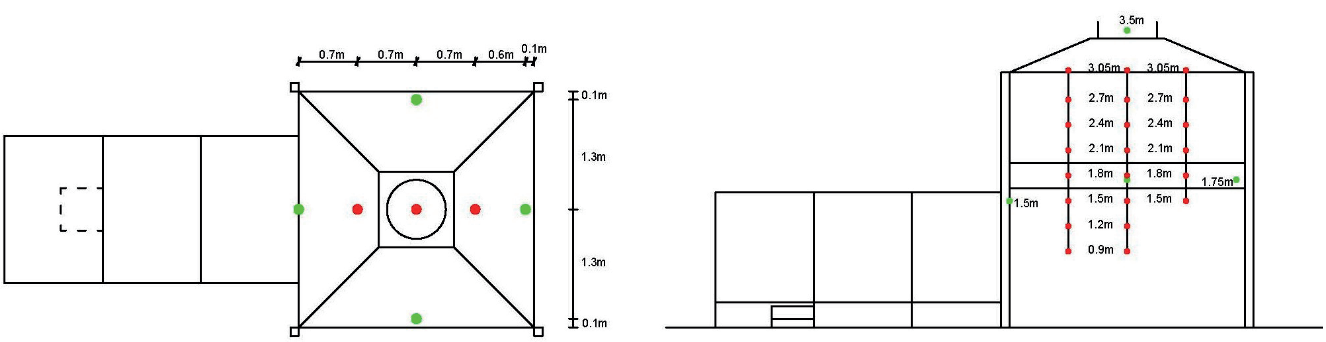

Figure 1. Positions of thermocouples; top view (left) and side view (right)

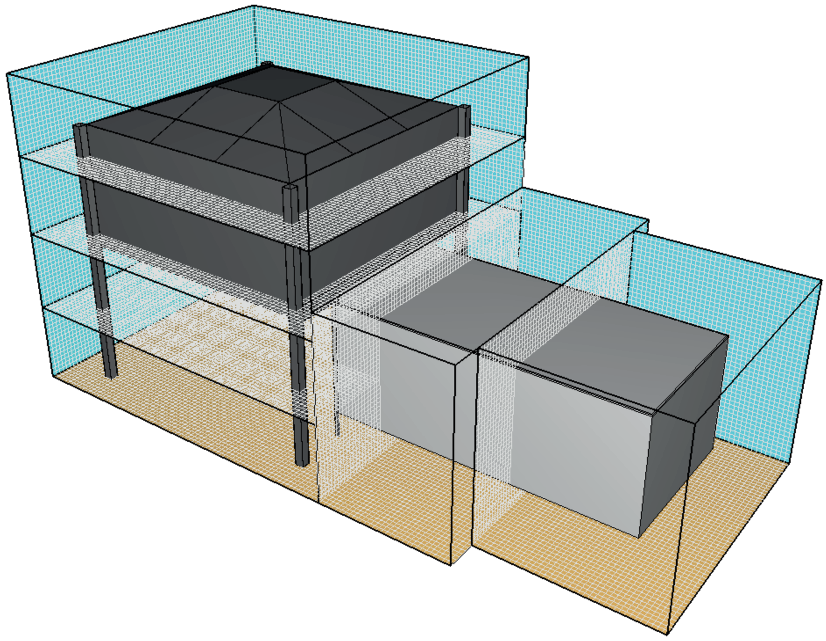

Figure 2 - Computational domain divided in sub-grids for MPI processing

pre-set pressures at the pump's frequency controller.

The smoke cabinet is equipped with a thermocouple array. To prevent the thermocouples from wetting, aluminium conical shields were placed above the thermocouples.

The combustion products are analysed in the exhaust duct to determine the HRR with the Oxygen Consumption Calorimetry (OCC) method. This method is similar to the described method in NEN-EN 13823+A1:2014 (Reaction to fire tests for building products) [9]. The following quantities are measured in the exhaust duct: temperature, differential pressure, O2-concentration and CO2-concentration. Humidity and atmospheric pressure are measured at the start of the experiment. With these quantities the volumetric flow rate, the oxygen depletion factor, the ambient mole fraction of oxygen in dry air and the HRR have been calculated.

NUMERICAL MODEL

The flow of a fluid can be described by the Navier-Stokes equations, a system of partial differential equations. For the modelling of turbulence FDS uses the Large Eddy Simulation model (LES-model). In this approach transport equations are solved for the large eddies and an eddy viscosity model (turbulence model) is used to model small eddies.

The mesh is restricted to rectangular Cartesian grids in FDS. The modelled physical space is divided into a uniform grid with approx. 600,000 cubic cells of 5.0 cm to solve the low Mach number equations. It is assumed that within each cell quantities as the gas velocity, temperature, pressure etc. are uniform and only change in time.

The HRR is calculated from the exhaust flow measurements and used as non-stationary input for the CFD model. The default 'simple chemistry' combustion model was used to determine the reaction products. This single-step, mixing controlled chemical reaction contains three lumped species, namely air, fuel and products. A lumped specie is a group of primitive species, e.g. air consists of oxygen, nitrogen and insignificant amounts of water vapour and carbon dioxide. The model requires the number of carbon, hydrogen, oxygen and nitrogen atoms, along with the soot yield and carbon monoxide yield to determine the reaction products. [6]

The spray pattern of a sprinkler can be characterized by characteristic diameters and statistical size distributions [8]. In numerical simulations, the water droplets are assumed to be spherical. However, in practice water droplets are not fully spherical, therefore the volume diameter can be described as the diameter of a sphere having the same volume as a droplet. The volume median diameter separates the higher half of the volume diameters from the lower half. The volume median diameter differs for different types of sprinklers and water pressures. [4]

When a sprinkler nozzle is activated and the water flow hits the deflector the water volume is scattered into small droplets, sprinkler atomization. To avoid the simulation of the complex atomization phenomena sprinkler droplets are introduced in the model at a spherical surface at a fixed distance from the sprinkler nozzle. The trajectory of a water droplet after injection at the spherical surface is calculated with the Lagrangian approach.

The centre of the sphere represents the sprinkler nozzle. The injection surface is divided into smaller surfaces by defining multiple elevation angles and azimuth angles. For every injection surface the velocity and mass fraction are inserted to model a realistic spray pattern. In numerical simulations, it is impractical to follow the motion of every single droplet in the sprinkler spray, therefore a particle injection rate (Np) is prescribed. A large group of real droplets is then represented by a computational Lagrangian particle [10].

Bucket tests in an open space are performed to model the sprinkler pattern. A mathematical model is used to translate the water collection at the floor into the above-mentioned injection properties for the spherical injection surface. The 'spray table' is implemented in the FDS-model to model the sprinkler spray. The results of the bucket tests are compared with the FDS predictions and subsequently the spray pattern table is improved by 'trial-and-error'. The spray table of the best-fitted results is used in the final simulations.

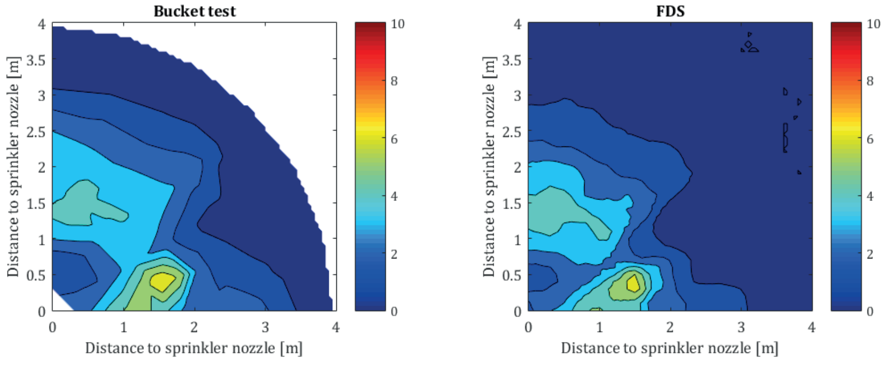

Figure 3. Water collection at floor (lpm/m2), Bucket test (left), FDS (right)

RESULTS

EXPERIMENTS

A smoke layer with an average temperature of approximately 150 - 160°C is reduced with 50°C, 70°C and 90°C for water flow rates of 56 l/min, 71 l/min and 93 l/min. In Figure 4 the average smoke layer temperature is shown of experiment SH2 (56 l/min). The sprinkler is manually activated, this is indicated by the dashed lines. The expected temperature curve without sprinkler activation is shown by the blue, dashed line. During all experiments, the HRR of the fire keeps slowly increasing during sprinkler activation, resulting in a small temperature increase of the smoke layer during sprinkler activation. It takes around 50 seconds for the smoke layer to reach its 'minimum' temperature and at this point the smoke layer is cooled down with 45 – 50 °C compared to the expected temperature without sprinkler activation. Once the sprinkler is deactivated the temperature starts to increase again till the fire is terminated.

SIMULATIONS

To examine cooling of the smoke layer in FDS the CFD-models were simulated two times. The first run includes the sprinkler spray. In the second run the same simulation is performed but without the sprinkler spray. In Figure 5 and 6 the simulation results corresponding to experiment SH2 and SH3 are shown.

In de development phase of the fire FDS predictions and the experimental results of SH2 show good agreement, thereafter the average temperature in the simulations is increasing faster and at sprinkler activation the temperature is approximately 20°C higher. The blue surface in Figure 5 shows the cooling of the smoke layer by the sprinkler spray in FDS. After 15 seconds of cooling the temperature starts rising again with the sprinkler still active. This effect is caused by the HRR which is increasing. During sprinkler cooling in FDS similar trends for temperature decrease and increase can be seen between the sprinkler model and model without a sprinkler. The temperature difference between those curves remains constant with an average cooling of 26°C. Where the temperature remains rather constant after its minimum is reached in the experiment, the predicted temperature in the model is higher at sprinkler deactivation than it was at activation.

To study the influence of cell size, for SH3, a coarse grid is created in the sprinkler region with cells of 10x10x10 cm. The cell size in the burner region is maintained equal to the previous models with a fine grid size of 5 cm. The results show that the underprediction of the average smoke layer cooling is larger than for the model with a fine mesh with cell sizes of 5cm, however this is no significant difference.

Table 1. Energy transferred to water particles and temperature decrease

Case: SH3	$Q_{particles}$	Avg. cooling (ΔT)
Fine grid + Complex pattern (reference)	102 kW (-)	59 K (-)
Coarse grid + Complex pattern	107 kW (+5)	53 K (-6)
Fine grid+ Simple pattern	95 kW (-7)	57 K (-2)
Coarse Grid + Simple pattern	108 kW (+6)	48 K (-9)

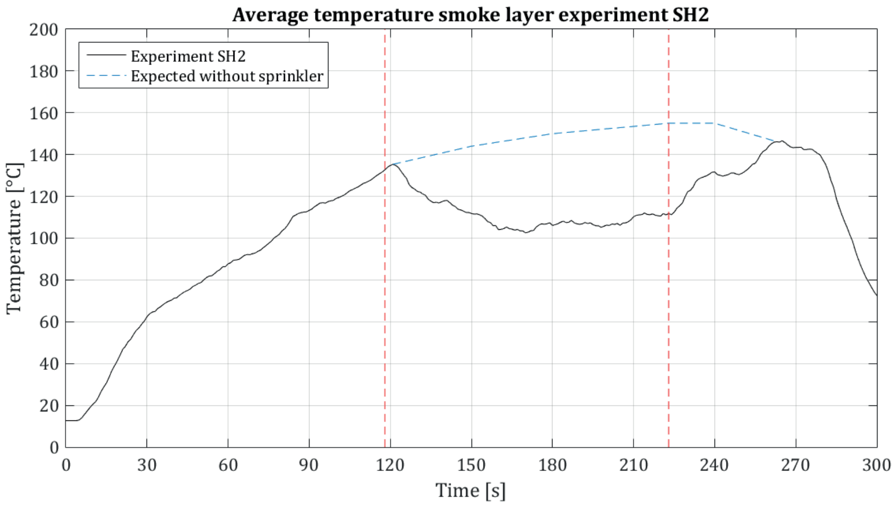

Figure 4. Average smoke layer temperature sprinkler test heptane 2 (SH2)

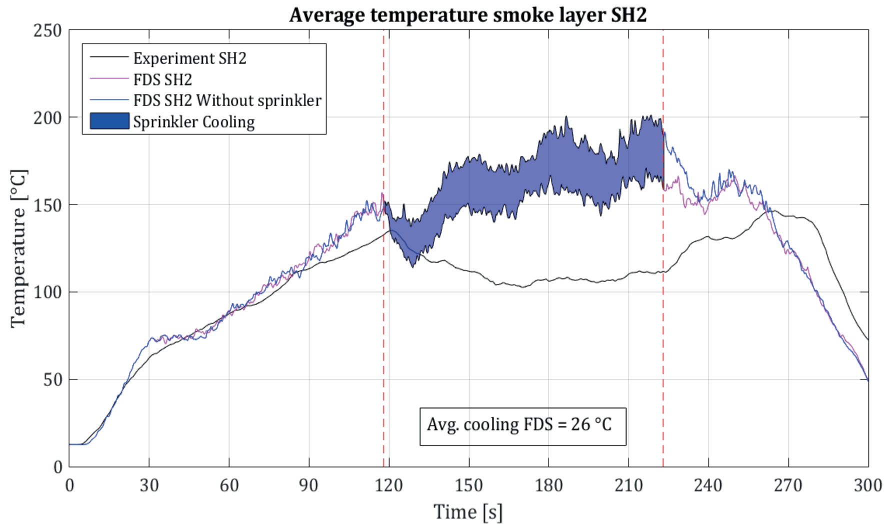

Figure 5. FDS predictions of average smoke layer temperature SH2

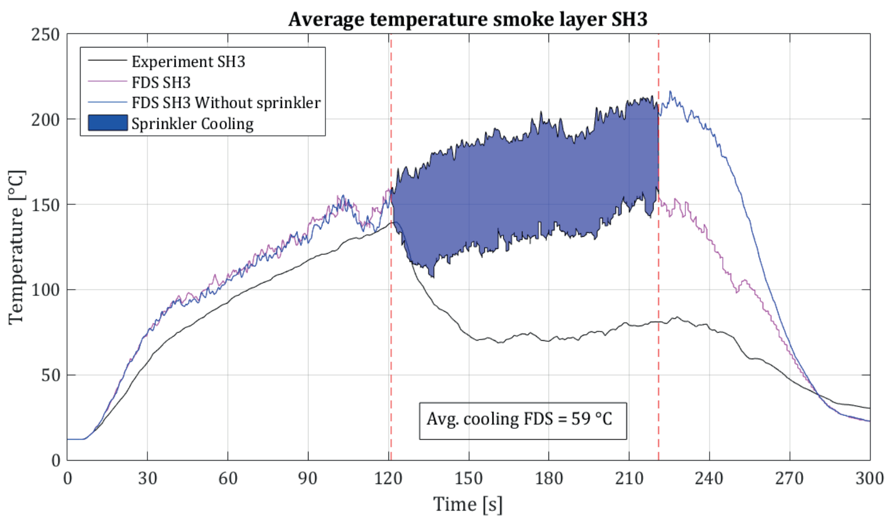

Figure 6. FDS predictions of average smoke layer temperature SH3

Figure 7. Smoke logging, water flowrate 56, 71 and 93 l/min

Also, a simple sprinkler spray pattern is used to see if the complex sprinkler spray model results in better predictions. In the simple spray model the spray envelope is not divided in small surfaces and a Gaussian distribution is applied between an elevation angle of 0° and the maximum elevation angle.

The difference of 2°C in smoke layer cooling with the sophisticated sprinkler model is negligible. However, the simple sprinkler spray drags more smoke down meaning the average temperature of the smoke layer is more difficult to compare to the measurements. The results of the different models are given in Table 1.
The results for the models with coarse grids are contradictory. More energy is transferred to the water particles but the temperature decrease is smaller than for models with a finer grid and lower. This implies that with a coarse grid other forms of energy transport are underpredicted. Additional research is required to gain more insight into these effects.

It can be seen in Figure 7 that the smoke logging effect increases when the water flowrate is increased. For a flow rate pressure of 56 l/min bar almost no smoke logging was observed where for 93 l/min a diffuse smoke layer reduces the visibility significantly.

CONCLUSION

This study has shown the limitations of FDS when used for predicting the effects of smoke layer cooling caused by an activated sprinkler nozzle. Multiple experiments with and without sprinkler activation are performed and thereafter simulated with FDS. The injection of water droplets into the sprinkler spray reduces the smoke layer temperature. The experiments show that increasing the water flow rate of the sprinkler nozzle result in a larger temperature decrease. The thickness of the smoke layer increases within the sprinkler spray envelope. A water flow rate, with relative large droplets, causes a very small amount of smoke logging, where a flow with smaller droplets, result in an unstable smoke layer and significantly reduced visibility. It can be concluded that smaller droplets amplify the downward smoke displacement.

All simulations that have been done with FDS underpredicted cooling by the sprinkler spray. The spray pattern that was modelled corresponded with a measured water distribution at the floor surface. Simulations with a low-detailed, simple spray pattern did not result in significant differences for the smoke layer cooling. Regardless of the level of detail from the sprinkler spray, the models embedded in the FDS code to solve the numerical equations are not capable of predicting the smoke layer cooling by water droplets.

The non-dimensional expression between the characteristic fire diameter and cell-size is often used to express a coarse, medium or fine grid. However, this ratio is dependent on fire size and results for large fires in a relatively large cell size, even for 'fine' meshes. The numerical simulations showed that in the sprinkler region coarsening of the mesh results in less accurate results. Therefore, the non-dimensional expression is not always applicable in the sprinkler region, meaning that the modeller needs to make a well-considered choice in this region.

Cooling of the smoke layer by a sprinkler spray is underpredicted in the FDS simulations, which results in a conservative outcome when studying the smoke layer temperature. In practice, the combination of conservative outcomes, high computational times, limited information about water droplet distributions and the required level of understanding, makes modelling sprinkler cooling with FDS less feasible.

[1] M. L. Bullen, "The effect of a sprinkler on the stability of a smoke layer beneath a ceiling," Fire Technol., vol. 13, no. 1, pp. 21–34, 1977.

[2] K. Y. Li, M. J. Spearpoint, J. Ji, R. Huo, Y. Z. Li, and L. H. Hu, "A Mathematical Model of the Drag Component of a Sprinkler Spray Adjacent to Horizontal Smoke Vents," J. Fire Prot. Eng., vol. 20, pp. 27–54, 2010.

[3] S. C. Li, Y. Chen, and K. Y. Li, "A mathematical model on adjacent smoke filling involved sprinkler cooling to a smoke layer," Saf. Sci., vol. 49, pp. 670–678, 2011.

[4] D. T. Sheppard, "Spray Characteristics of Fire Sprinklers," Evanston, 2002.

[5] D. van Venrooij, "Validation of Fire Dynamics Simulation with sprinkler sprays," 2016.

[6] K. McGrattan, S. Hostikka, R. Mcdermott, J. Floyd, C. Weinschenk, and K. Overholt, Fire Dynamics Simulator User ' s Guide. 2017.

[7] S. C. Li et al., "Studies of Cooling Effects of Sprinkler Spray on Smoke Layer," Fire Saf. Sci., pp. 861–872, 2008.

[8] K. Y. Li, R. Huo, J. Ji, and B. B. Ren, "Experimental investigation on drag effect of sprinkler spray to adjacent horizontal natural smoke venting," J. Hazard. Mater., vol. 174, pp. 512–521, 2010.

[9] Nederlands-Normalisatie-instituut, NEN-EN 13823+A1:2014 Bepaling van het brandgedrag van bouwproducten. The Netherlands, 2014

[10] T. Beji, S. E. Zadeh, G. Maragkos, and B. Merci, "Influence of the particle injection rate , droplet size distribution and volume flux angular distribution on the results and computational time of water spray CFD simulations," Fire Saf. J., vol. 91, pp. 586–595, 2017.