

**Suppression, Detection and Signaling Research and Applications**  
**A Technical Working Conference (SUPDET 2012)**  
**March 6-9, 2012**  
**Sheraton Crescent Hotel, Phoenix, AZ**

**Title: Simulation of Sprinkler Activation Time with CFD tools**

**Authors: Luca Iannantuoni(1), Daniele Ettorre(1), Andrea Ferrari(2)**  
(1)Dipartimento di Energia, Politecnico di Milano  
(2)Huges Associate Europe

**Abstract**

Automatic sprinkler systems are considered the most important ones in fire protection strategies to reduce deaths, injuries and property damage. To assure that they have been properly designed in order to assure adequate reliability in the given scenario, referring to a specified level of hazard, real scale tests and models calculation have been widely used.

In facts, unsatisfactory sprinkler system performances may occur because the application is slightly different to a certain standard protocol or because, later than the design stage, the structures change in characteristics, use and function. The sprinkler systems' effectiveness during a fire and so its analysis is strongly related to the activation time of the nozzle, commonly accounted with the Response Time Index (RTI) parameter, determined with experimental tests.

The response of detection devices in general, is due to the hot flow of the combustion products that moves due to buoyant forces and mechanical ventilation, and this flow is strongly affected by geometric details, such as the ceiling slope or obstacles, the location and the heat release rate of the fire, the different ambient conditions. Real scale fire tests are often be conducted to foresee if the estimated response times match well with the actual ones. Among many models that have been developed in the last decades to predict the behavior of thermal detectors, smoke detectors and sprinklers, the CFD models are nowadays the more promising ones. Although they still hold the most common mathematical models to calculate the energy balance through the sensible element, without tacking into account the complex geometry of the nozzle, open the way for a deeper investigation among the convective flows that may occur in the particular fire scenario, allowing the scrutinizing of the relevant factors that may affect the device thermal response and so the sprinkler to operate.

The aim of the present work is to validate a model based on OpenFOAM© CFD Toolbox that is able to correctly predict the RTI of a given sprinkler nozzle, comparing results with the experiments conducted in the plunge oven heat test, with which the RTI itself it's calculated. Once the model is validated and the sensitivity analysis is carried out among mesh sizes and numerical parameters, the model is applied to a case study in a warehouse to estimate the actual time response of the designed system to verify if it meets the design criteria.

Results are shown also to give evidence of the accuracy of such estimations in order to understand how experiments could be better addressed to characterize such devices and how the models could be an interesting tool for fire investigations and design purposes.