
Jack R. Mawhinney and Javier Trelles

ABSTRACT

In this study a computational fluid dynamic (CFD) model was applied to a series of full-scale fire tests of water mist systems conducted in 2006 in the San Pedro de Anes test facility in Spain. Data were collected on the heat release rates (HRR) of very large test fires involving wood pallets and plastic pallets under controlled conditions. Thermocouples placed on the tunnel ceiling and at different distances from the fire recorded temperatures throughout the fire tests. The tests provided valuable insight into the dynamics of water mist interacting with very large fires in a tunnel. The fires were more dynamically complex than assumed in selecting indicators of performance. To assist in understanding the dynamics of such fires and to aid in identifying reproducible “global” performance criteria as an alternative to single-point measurements, the Marioff San Pedro tunnel fires were modeled using a CFD model. The purpose was to evaluate the performance of water mist systems over a broader range of performance indicators than could be measured during full-scale tests. The NIST Fire Dynamics Simulator version 4 (FDS4), a CFD model with a water mist spray nozzle algorithm, widely used in fire science and engineering, was used to simulate the Marioff San Pedro tunnel fires. There is presently extensive international activity by many agencies in applying CFD to tunnel safety problems. However, there are numerous challenges involved in applying a CFD model, with discipline and intent for accuracy, to very large fires interacting with water mist. This paper discusses how specific modeling challenges were approached for this study. The paper illustrates that CFD modeling is a powerful tool for understanding the global benefits of water mist systems.

KEYWORDS: computational fluid dynamics, fire control, fire testing, FDS, heavy goods vehicle, performance criteria, tunnels, water mist.

1 INTRODUCTION

Before a manufacturer’s water mist system design will be accepted by transportation authorities, highway safety officials within each country require fire testing to evaluate the performance and establish design criteria. Since 2000, various full-scale fire test programs involving water mist fire protection systems have been carried out by different groups in numerous countries in Europe. Ideas about how to design fire tests to evaluate system performance and about what constitutes acceptable performance of a water mist system vary between countries and agencies. A common feature of all of the fire test programs undertaken in recent years, however, has been that the test fires are much larger in scale than has been the norm for performance testing of fire protection systems in traditional industrial applications. This is largely the result of tests conducted in 2003 in the Runehamar tunnel in Norway [1], which revealed that the heat release rates from uncontrolled fires in common heavy goods vehicle (HGV) fuel loads could range from 75 to 200 MW.

Very large test fires introduce a number of difficulties for approval-test programs. The large fire sources used, consisting of large numbers of stacked wood or plastic pallets, are not well understood or characterized, particularly in the special conditions created by confinement in a tunnel. Furthermore, in establishing criteria for the minimum level of performance of a fire-control system,
often the choice of a particular condition as a pass or fail criterion is based on a theoretical construct that does not take into account the chaotic environment of a very large fire. Most fire test protocols for water mist systems establish pass/fail criteria based on single-point measurements, such as a thermocouple temperature reading or a heat flux value at a certain height and distance from the fire. It is difficult to compare single-point measurements between tests because of the highly variable local conditions generated by large fires. Assuming that the test materials have consistent burning properties, local ventilation conditions may be modified by obstructions, with a disproportionate effect on heat release rate and fire growth rate. Transient flame extensions create erratic readings such that it is not possible to ensure a specific condition such as temperature or heat flux, to within one or two meters. Fuel packages change shape and height over the course of a fire, causing apparently illogical spatial and temporal variations in readings.

In the fire test scenarios developed by the International Maritime Organization for evaluating water mist systems for marine machinery spaces on ships, a 10 MW fire was considered to be a “large” fire [2]. The IMO water mist test protocol for machinery spaces intended that the fires be extinguished within 15 minutes. In contrast, a water mist fire protection system for tunnels is not expected to extinguish HGV fires. The objective of the water mist system is to control the fire, prevent temperatures in the tunnel from exceeding extremes, and to prevent fire propagation to other vehicles in the tunnel. A controlled fire in a tunnel may have peak heat release rates of 20 to 50 MW while being prevented from growing to its full potential, which may be as high as 100 to 150 MW. Although the HRR during the control period may be only a fraction of what the fuel package would achieve without the water mist system, a sustained 20 MW fire is nonetheless a large fire, and it generates a large volume of flame. The controlled fires are highly sensitive to apparently small changes in ventilation or spray penetration; therefore the HRR itself will not be a steady-state value. It is important to recognize the dynamic nature of these fires in selecting appropriate criteria for measuring performance of the fire protection systems. The progression of events associated with controlled, but still large, test fires in tunnels does not follow a linear path that can be consistently confirmed by a few single point measurements.

Marioff Corporation Oy of Finland conducted a program of full-scale fire test involving HGV fires in the San Pedro de Anes Tunnel Safety Test (TST) facility (San Pedro tunnel) in Asturias, Spain in 2006. Hughes Associates, Inc. (HAI) acted as a third party witness and reporting agent for the tests. The tests evaluated the performance of a high pressure water mist system against fires in stacks of standardized wood pallets and of wood pallets interspersed with polyethylene (HDPE) plastic pallets. The tunnel was equipped with instrumentation to allow calculation of the heat release rate (HRR) of the fires by oxygen calorimetry, and thermocouples were mounted on the ceiling and at various locations in the tunnel. Details of 11 tests conducted in the San Pedro tunnel, including heat release rate data and temperature plots are provided in the test report [3].

Examination of the HRR data and the time-temperature plots in the San Pedro tunnel tests supported the following observations:

- The water mist prevented fires from reaching the full potential peak HRR for the fuel array; fuel arrays of potentially 75 MW were sustained between 20 and 40 MW; fuel arrays of potentially 100 MW were limited to approximately 60 MW.
- Continued burning of fuel packages at 20 to 25 MW generated significant volumes of flame
- Because of shadowing by unburned pallet stacks temperatures closer to the fire could be cooler than temperatures farther from the fire, confounding simplistic performance criteria.
- Temperatures directly over the fuel package where flames impinged on the ceiling were 800°C or higher over short sections of ceiling.
- Expectations that the water mist system should limit temperatures above the fire to 500°C, or that temperatures farther from the fire “ought to be” lower than closer to the fire, could not be met.

At the same time, it was evident that dramatic reductions in the severity of the impact of the fires on the overall tunnel environment were realized in all tests. Whether the temperature measured at a point
was 450°C instead of 350°C, at 8-m instead of 5-m from the fire, or at 15-min instead of 10-minutes after activation, imposed a demand for precise control that was inconsistent with the chaotic variability of the large fires. A few single point readings of temperature at pre-selected points were not adequate to reflect the broader thermal management benefits of the water mist system. The quest was initiated for “global performance criteria” that would allow one to evaluate the benefits of a fire control system, in spite of the large variability and uncertain reproducibility of single point measurements. Global performance criteria, for example, should be based on measures that are not subject to the large variability associated with single-point measurements.

A review of tunnel safety literature from recent conferences and symposia reveals that CFD modeling is being widely used as an investigative tool for tunnel safety issues [4, 5]. Computational fluid dynamics (CFD) modeling provides a means to visualize and analyze the phenomena involved in the tunnel fires beyond what can be observed or measured directly. To this end, HAI simulated the Marioff San Pedro tunnel fire tests using the NIST Fire Dynamics Simulator (FDS4), which is a CFD model with water mist spray nozzle algorithms. If the model can be shown to predict conditions similar to those measured during the full scale tests, within the uncertainties inherent in the test data, it will be confirmed to be a useful tool for evaluating the performance of water mist systems over a broader range of conditions than could affordably be tested.

2 DESCRIPTION OF THE FDS4 MODEL

The San Pedro de Anes Tunnel Safety Testing Centre (San Pedro test tunnel) is 600-m long, with an S-bend curvature and a -1% gradient. The tunnel is built at grade in concrete, with dimensions equivalent to a two-lane road tunnel. A false concrete ceiling was installed to create a rectangular cross-section with dimensions 9.50 wide by 5.17-m high. The dimensions of the test tunnel are as shown in Table 1 and Figure 1.

<table>
<thead>
<tr>
<th>Table 1. Dimensions and features of the San Pedro test tunnel.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length:  600 m</td>
</tr>
<tr>
<td>Width:  9.50 m</td>
</tr>
<tr>
<td>Height:  5.17 m with false ceiling</td>
</tr>
<tr>
<td>Cross section (with false ceiling):  49.4 m²</td>
</tr>
<tr>
<td>Minimum radius (S-bend):  400 m</td>
</tr>
<tr>
<td>Longitudinal gradient:  (-1 %)</td>
</tr>
<tr>
<td>Transversal gradient:  2 %</td>
</tr>
<tr>
<td>Emergency gallery:  4 m by 2.50 m</td>
</tr>
<tr>
<td>Emergency exits:  3 (one each 150 m)</td>
</tr>
<tr>
<td>Ventilation source:  6 ceiling jet fans</td>
</tr>
</tbody>
</table>

Figure 1. Cross-sectional view of the San Pedro test tunnel.

Computational Domain

Fire Dynamics Simulator, version 4.0.7 (FDS4), was used. FDS4 is a three-dimensional large eddy simulation (LES3D) CFD program developed at the National Institute of Standards and Technology (NIST) Building and Fire Research Laboratory (BFRL) [6, 7]. FDS is developed specifically for studies related to fire science and engineering. At the time this work began Version 5 was at an early release state. Version 4.0.7 was employed for the current study.
To achieve an adequately small cell size, only a 140-m long section of tunnel between Station 0+320-m and 0+460-m was modeled. This section incorporated the second half of the S-bend, the water mist system piping and the instrumented portion of the tunnel. Figure 2 illustrates the domain and Table 2 summarizes the characteristics of the computational domain. Although the tunnel was 9.5-m wide, the FDS domain had to be 23-m wide to accommodate the curvature. The FDS4 domain was divided into cells of dimension 0.250-m × 0.230-m × 0.215-m. This was adequate for tunnel flows, but not sufficient to resolve flow in the pallets. The domain included a centerline, stations marked at 10 m intervals, ceiling thermocouples, four thermocouple tree stations, and a fuel load. The left end of the tunnel contains the forced-flow boundary condition governing the inlet air velocity. The right end is open. Unused space is blocked out. The line down the middle of the tunnel represents the tunnel centerline. Vertical bars demarcate the projections of the 10 m stations along the tunnel centerline and are numbered accordingly. This domain contains three parallel lines of water mist nozzles, 3.3-m on either side of a center line, with nozzles 0.1 m below the ceiling at 4-m spacing between stations 356 m and 424 m. Thermocouples and water mist nozzles are placed to within plus or minus half the resolution given in Table 2.

Thermocouples were spaced at 5-m intervals along the centerline 0.1-m below the ceiling surface. There were four vertical thermocouple tree stations placed along the tunnel, two upstream (T1 and F1) and two downstream (F2 and T2) of the fire location. Plots of the temperatures recorded at these locations throughout each test were presented in the test report [3]. These data could be used to compare with the results of the FDS4 simulation, as a means of validation.

**Figure 2.** The computational domain for the San Pedro test tunnel fire simulations.

**Table 2:** Summary of FDS4 input parameters used in the simulation series.

<table>
<thead>
<tr>
<th>Category</th>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD Domain</td>
<td>Facility</td>
<td>San Pedro de Anes Research Tunnel</td>
</tr>
<tr>
<td></td>
<td>Simulation dimensions</td>
<td>140 m×23 m×5.17 m</td>
</tr>
<tr>
<td>Numerical</td>
<td>Grid dimensions</td>
<td>560×100×24 cells</td>
</tr>
<tr>
<td></td>
<td>Cell size</td>
<td>25.0 cm×23.0 cm×21.5 cm</td>
</tr>
<tr>
<td></td>
<td>Total # of cells</td>
<td>1,344,000</td>
</tr>
<tr>
<td></td>
<td>Wall boundary conditions</td>
<td>Concrete</td>
</tr>
<tr>
<td></td>
<td>Floor boundary conditions</td>
<td>Concrete</td>
</tr>
<tr>
<td></td>
<td>Ceiling boundary conditions</td>
<td>Concrete/Promat Promatect-H</td>
</tr>
<tr>
<td></td>
<td>Gravity vector (-1%)</td>
<td>(0.0981, 0.0, -9.8095) m/s²</td>
</tr>
<tr>
<td>Spray Nozzle</td>
<td>Type</td>
<td>Marioff 4S1MD6MD(1000,10RE) water mist</td>
</tr>
<tr>
<td></td>
<td>Configuration</td>
<td>4 m×3.3 m grid</td>
</tr>
<tr>
<td></td>
<td>Activation criteria</td>
<td>Times as determined from test data</td>
</tr>
</tbody>
</table>

**Water Mist Drop-Size Characterization**

The original spray model in FDS4 was based on the spray distribution characteristic of standard sprinklers. High pressure water mist nozzles have a very different drop size distribution and range of velocities and spray angles. The standard spray model in FDS4 uses a composite Rosin-Rammler log-normal distribution, as shown in Eq. (1).
The parameters of interest are the log-normal standard deviation $\sigma$, and the Rosin-Rammler exponent, $\gamma$. In a study performed by HAI in 2004 [8], the FDS spray model was found to provide a poor fit to the fine fraction leg of the distribution because FDS4 calculates $\sigma$ from $\gamma$ and imposes a slope continuity requirement at the intersection between the two branches of the composite distribution curve. Hence FDS4 was modified so that both $\gamma$ and $\sigma$ could be input and then processed as input in all the pertinent calculations within FDS4. The drop size distribution for the 4S 1MD 6MD 1000 open spray nozzle was determined experimentally by Marioff. The second nozzle used in the tests (thermally activated 2N 1MD 6MD 10RE) had the same orifice sizes, therefore the same drop size distribution was used for both types of nozzles. The three measured values for $Dv_{10}$, $Dv_{50}$ and $Dv_{90}$ were used. These represent the drop diameters for the 10, 50 and 90 percent cumulative volume fractions of the spray, where $Dv_{50}$ is the median diameter $d_m$. The values from the measured cumulative volume fraction distribution curve are the best choice since it is the unadulterated distribution. The measured drop size distribution gave $Dv_{[50]} = d_m = 89$ micron, $Dv_{[10]} = 35$ micron, and $Dv_{[90]} = 171$ micron. From these data it was determined that the $\gamma = 1.84$ and $\sigma = 0.728$. Figure 3 shows the resulting Rosin-Rammler/log-normal distribution for the water mist spray characterization.

![Figure 3. Cumulative volume and number fractions of a Marioff HI-FOG water mist nozzle.](image)

**Spray Nozzle Characterization**

The spherical model within FDS4 was used for characterizing the spray distribution pattern of a multi-port nozzle. Figure 4 shows the sphere with radius of 0.2 m divided into 1056 solid angles. Droplets can be introduced through any of the user-defined solid angles that make up the sphere. The nozzle had one port on the center axis and 6 circumferential ports, which were evenly spaced around the circumference 45° from the south pole. The single center jet was modeled using the 48 solid angles around the south pole of the sphere. Thus, 54 of the solid angles were assigned a non-zero flow value. Required inputs are the initial droplet velocity $u$, and the mass flux, $\dot{m}$, through the face of each solid angle. The total mass discharge rate from all orifices, as a function of pressure, was provided by the “K-factor” for the nozzle. FDS4 would introduce droplets from each face in Figure 4 according to the relative value of $\dot{m}$ for that solid angle.
Each batch of drops reflected the drop size distribution determined for the overall spray defined earlier. The initial velocity at each face was determined from orifice flow calculations. It was noted that the temperature of the water in the piping system varied in different tests, depending on the allowed pre-burn time and the number of nozzles opened. An estimate of different initial water temperature was made. Table 3 presents the nozzle characterization parameters used for the tests simulated in this study. It should be noted that in the previously referenced HAI spray characterization study [8], it was found that the quality of the tracking of the particle distribution from the nozzle (under non-fire conditions) improved significantly with decreased grid size, and was optimal at a grid size approximately 1/10th the size used in this simulation. Due to computational limitations such a fine resolution grid was impossible to apply to the 140-m tunnel length.

**Table 3. Nozzle characterization parameters used in the simulations.**

<table>
<thead>
<tr>
<th>Test Identifier</th>
<th>Run Pressure (bar)</th>
<th>$T_{H_2O}$ ($^\circ$C)</th>
<th>4S 1MD 6MD 1000</th>
<th>2N 1MD 6MD 10RE</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td>$u$ (m/s)</td>
<td>$m_{c}^n$ (a)</td>
</tr>
<tr>
<td>1</td>
<td>100</td>
<td>50</td>
<td>123.5</td>
<td>51.2</td>
</tr>
<tr>
<td>2</td>
<td>80</td>
<td>50</td>
<td>110.5</td>
<td>45.8</td>
</tr>
<tr>
<td>10</td>
<td>90</td>
<td>80</td>
<td></td>
<td>110.8</td>
</tr>
<tr>
<td>13</td>
<td>80</td>
<td>50</td>
<td>110.5</td>
<td>45.8</td>
</tr>
</tbody>
</table>

*a.) The subscript "c" stands for center orifice; subscript "r" for ring orifices.

Activation times for the water mist spray nozzles and the operating pressure in each test were obtained from the test data. The pressure in the nozzle characterization files was modified to match the nominal zone pressures for each test.

**Modeling the Fuel Package and Heat Release Rate**

The HGV fire fuel packages consisted of large stacks of wood “euro-pallets”, with dimensions 7.7 m x 2.4 m x 2.1 m (L x W x H), placed on a 1-m high platform in the center of the tunnel. From oxygen calorimetry a HRR curve for each fire under controlled conditions was obtained. Inputting an actual HRR versus time plot for a crib-like fuel array was challenging. The heat release rate from any given section of a pallet stack can rise or fall according to the wind blowing through the array, contributions from other members as the combustion reaction spreads, and radiative feedback from walls and
ceilings. The method employed for this study attempted to capture important phenomena such as the gross heat release rate, flame spread across the fuel load, air flow though the pallet arrays, and the progressive collapse of consumed sections.

Figure 5. Left: photograph of the end-view of pallet stacks showing 5 “columns”. Right: the support structure used in the model for the top-cell method, to allow air to flow through the fuel array.

Because the 0.25-m resolution in the cell height was greater than the vertical dimensions of the individual pallets, the full detail of the stacked pallets could not be represented. Instead, methods were explored by which some of the flow-through-porous-media effects could be obtained given the limitations of FDS and the resolution used for the simulations. The stacks of pallets were modeled as 5 vertical “columns” representing the stacked wood blocks in the pallets, with air space between and a solid plane for the upper surface of the array. The combustion phenomenon was assigned to strips of cells on the upper plane of the array, as is described below. The structural representation of the fuel load was solely to support the top cells and to model the obstruction-effect of the stacks without creating a solid block obstruction.

Figure 6 is a photograph of a pallet fire at approximately 3 minutes after ignition, with no wind-breaks or tarpaulins and 2 m/s ventilation velocity and prior to activation of water mist. Several approaches were investigated to recreate the flame volume observed around the burning pallets. The approaches included a “block method”, a “full-height top-cell method”, and a “half-height top-cell method”. The “half-height top-cell method” provided the best representation of flame volume above and around the fuel array.

The “Half-Height Top Cell Method” of Modeling Stacked Pallet Fire Growth

For the top cell method, flames were only allowed to come from the top of the support structure representing the pallet load. This was chosen because FDS’s flame height algorithm was calibrated using flat fire bed examples. After trials with a full-height support structure, instead of placing the plane of cells at the elevation of the top of the pallets in the tunnel, the surface of cells was placed at approximately half the height of the pallet stacks. The result was a collection of upward facing “fire bed” cells at approximately 2.9-m below the ceiling. This allowed for a greater volume to be filled with flame than would be the case if the flames originated from the true top surface of the pallets, only 1.9-m below the ceiling. The goal was to get the majority of the flames occurring outside the pallet load as the test photographs indicated.
The flame spread methodology was as follows. The top surface of the support structure was divided into cells as shown in Figure 7. All cells were assigned a heat release rate per unit area $q'$. In the top cell method, cells burn from front to back along the top of the commodity only. The combustion progresses as a traveling band of open cells. Uninvolved cells are denoted with an $\times$. Cells that burn-out are also shown with an $\times$. For these cells, the supporting structures are removed in order to simulate the collapse of the commodity.

This quantity, $q''$, varied from simulation to simulation, and was determined by taking the maximum HRR achieved in a test and multiplying it by a scaling factor proportional to the cell-life-to-run-time ratio, dividing it by the number of available cells, and dividing it by the area of one top cell. Starting from the middle of the width at the upwind end, and working laterally in both directions, individual cells would start to burn according to the input HRR curve. In the absence of longitudinal flame spread data, each cell was given a finite life of about a quarter of the total simulation time. This created a de facto spreading front across the surface of the fuel load. The number of available top cells varied from simulation to simulation as well, being determined by the fuel array dimensions and by the percentage of the pallets not consumed. Once a strip of top cells ceased to burn, the supporting obstacles were removed as well, simulating the collapse of consumed pallet stacks.

FDS's oxygen depletion algorithm was turned off. This function would have reduced the HRR if there was insufficient oxygen in a cell. Since the goal of the simulations was to match a confirmed input heat release rate, further automatic reduction by FDS4 was unwanted.
Although it was not possible to conduct a free burn (uncontrolled) fire test in the San Pedro test tunnel, a simulation of a fire without water mist was performed using the FDS4 model. The uncontrolled fire utilized a HRR curve for Test 10, which was a severe fire that reached a peak of 57.5 MW with a minimal application of water mist. The Test 10 HRR curve was used as input for a fire in the location and with the ventilation conditions of Test 1, but without applying any water mist. The curve was a conservative (under) estimate of the severity of an uncontrolled fire in the wood pallets. The results of the simulation revealed the extent of extremely high temperatures and heat flux in both the upwind and downwind directions. The results provided an unambiguous benchmark to which the global performance benefits of a water mist system could be compared.

3 RESULTS

Several plots from Test 1 will be compared to simulation results for Test 1 to demonstrate that the level of agreement was sufficient to justify use of the model for a scenario not tested, namely the uncontrolled fire case. Two field plots are presented to show the contrast between the thermal conditions for a controlled fire case (Test 1) and a simulated uncontrolled fire.

As each simulation run proceeded, a check was performed to determine how closely the top cell algorithm value for HRR at time $t$ was to the smoothed HRR test data. Figure 8 shows the comparison for Test 1.

![Figure 8. Comparison of HRR from test data, with the FDS4 simulated HRR curve for Test 1.](image)

In the simulation, the HRR averaged around the smoothed test curve, and the magnitude of the “noise” either above or below the average value was contained within the estimated uncertainty bars. The agreement shown between the simulated and the measured HRR input is excellent.

Figures 9 and 10 show the ceiling thermocouple temperatures for Test 1, for the test and the simulation, respectively. The figures show that the simulation reproduced the major cooling effects associated with the water mist acting on the fire. Just prior to activation of the water mist system at 7:30-mins, the temperature at TC C07, 26 m from the fire area, was measured at 350 °C; the simulation indicated a temperature of 300 °C. At the same time, TC C11 indicated temperatures just over 500 °C; the simulation showed approximately 550 °C. The differences between test and simulation temperature before activation of the mist system were within 50 °C. The thermocouples above the fuel package recorded temperatures from 800 °C to 1000 °C indicating contact with flame. The simulation predicted temperatures in the same range. Following activation of the water mist, the simulation captured the large reduction in temperatures downwind of the fire. For TCs C10 to C07 the simulation temperatures were generally between 150 °C and 200 °C, whereas the measured temperatures were between 50 °C and 100 °C. Thus, the simulation temperatures were approximately 100 °C higher than measured. While thermocouples in
wet conditions may be cooled by the presence of water condensate, it is also the case that the model results were sensitive to the properties of walls and ceiling. The floor pooling function in FDS was turned off to minimize computational demand. It is possible that downwind temperatures would have been lower (in the simulation) if the effect of water accumulated on the floor had been included.

Figure 9. Ceiling temperatures at 5-m spacing, measured in Test 1. TC C13 (highest temperatures) is exposed to flames directly above the fuel array.

Figure 10. Ceiling temperatures at 5-m spacing, from simulation of Test 1. TC C12 (highest temperatures) is 1-m downwind from the end of the fuel array.

Figures 11 and 12 compare selected longitudinal temperature profiles in the tunnel for Test 1, from the test data and the simulation, respectively. The tunnel ventilation air moves from left to right. These plots show the center-line temperature at 5-m intervals between station 345 m and 450 m at the indicated times. In Figure 11, the highest temperatures measured in the test occurred just prior to activation of the water mist system at 450-seconds after start of the simulation. The temperature at station 360 m, 25-m upwind was measured at 200°C – indicating back layering. The simulated temperature at Station 360 m in Figure 12 shows 240°C, within the error bar of 200°C. Following activation of the water mist, the back layering disappears to approximately station 380 m in both figures. Thus, the line of ceiling thermocouples extending over 135-m of tunnel, revealed several “global” benefits of the water mist system – elimination of the back-layering in the upwind direction, and a reduction in down wind temperatures measured in 100’s of degrees. To reveal these global effects of the water mist system required a line of over 20 thermocouples. The effect on the back-layering would not have been so clearly evident from a single-point reading in an area of erratic, variable readings.

Prior to activation of the water mist at 450-s, the downwind simulation results in Figure 12 agree very well with the test temperatures shown in Figure 11. After activation of water mist, the agreement is not quite as close, as simulation temperatures were roughly 100°C higher than measured.
Nevertheless, the simulation conservatively predicted the global performance, i.e., the scale of the thermal management provided by the water mist system.

Figure 11. Ceiling temperature profile along the length of the tunnel, at instant just prior to water mist activation, and at 5 minute intervals thereafter, measured in Test 1.

Figure 12. Ceiling temperature profile along the length of the tunnel, at 420-s, prior to water mist activation, and at 5 minute intervals thereafter, from simulation of Test 1.

The FDS model can be used to evaluate global benefits of a fire protection system beyond what can be measured. In this study, contrasting conditions with and without water mist provided a striking example of how the model assists understanding of the scale of the benefits. Figures 13 and 14 show the results of a simulated heat flux calculation for an uncontrolled and a controlled fire, respectively, at 16 minutes after ignition. The darkened zone (red in color) in the uncontrolled fire (Fig. 13) represents the zone wherein the heat flux to the floor exceeds 5 kW/m², high enough to be untenable for unprotected humans, and to present a serious challenge even for fire fighters in protective clothing. Heat fluxes at elevations above the floor would be higher still. Without a water mist system, the dark area extends from station 390 m to Station 435 m. With the water mist system (Fig. 14) it is confined to a small region at the end of the burning vehicle. The point is made that, when contrasted against conditions resulting from a fire in a tunnel with no fire protection system, the global performance benefit of the water mist system is unquestionable. It is advisable to provide sufficient instrumentation for performance tests to identify global performance benefits of this scale, without undue reliance on single-point measurements which could be in a zone of high variability.

4 CONCLUSION

This paper has described how full-scale fire tests involving very large HGV fires interacting with water mist in a tunnel are much larger, more chaotic hence less predictable than smaller fires commonly used in fire testing. Criteria used to evaluate performance of water mist systems should not be based on single-point measurements that are highly variable, not reproducible with any degree of consistency and fail to reveal the actual scale of the benefit provided by the water mist system. By
combining CFD analysis with full-scale fire test results to validate the general accuracy of the model, it is possible to illustrate the benefits of the fire protection system beyond what can be measured or even practically tested. CFD modelling is a powerful tool for understanding the global performance of water mist systems in tunnel fires. Use of the model makes it easier to develop instrumentation strategies that are consistent with the scale of the uncertainties inherent in large, chaotic test fires.

**Figure 13.** Heat flux to the floor at 16 min 5 s, for an uncontrolled fire at 16 min 5 s. The elongated dark area between station 390 m and 435 m indicates heat flux at the floor > 5 kW/m².

**Figure 14.** Heat flux to the floor, Test 1 with the water mist system, at 16 min 5 s. The small dark “island” between stations 394 m to 400 m indicates heat flux at the floor > 5 kW/m².

**REFERENCES**