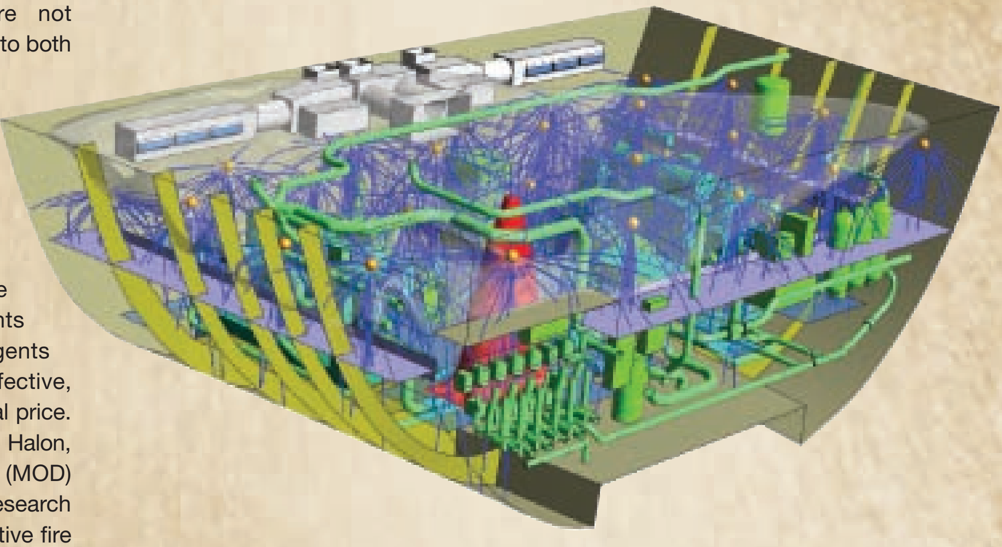
# Fighting Fire with Simulation

## The U.K. Ministry of Defence uses engineering simulation to find alternatives to ozone-depleting substances for fire suppression.

*By Michael Edwards and Michael Smerdon, U.K. Ministry of Defence, Bristol, U.K.*
*Yehuda Sinai and Chris Staples, ANSYS, Inc.*

Fires onboard ships are not uncommon and pose a danger to both crew and equipment. It is vital to develop effective methods to extinguish these fires. At the same time, international agreements such as the Montreal Protocol on Substances that Deplete the Ozone Layer have been signed. These agreements limit the use of firefighting agents such as Halon that, though effective, come with a high environmental price. In order to find an alternative to Halon, the U.K. Ministry of Defence (MOD) completed a comprehensive research program that looked at alternative fire suppression technologies for use on Royal Navy vessels. The work led to the development of a low-pressure water mist system, or fine water spray (FWS). This new FWS system combines salt water from a ship's high-pressure salt water (HPSW) system, which typically operates at a pressure of 7 bar, together with a 1-percent-concentration aqueous film-forming foam (AFFF).



Temperature isosurfaces and droplet trajectories before fire extinction is completed in a ship's machinery space

As part of this program, MOD validated and used simulation as a tool to assess the performance of the FWS system, with and without additive, when fitted onboard a ship. This analysis decreased the need for expensive fire testing for future assessments and design of fire control measures. The United Kingdom ANSYS office developed a fluid dynamics model using ANSYS CFX software, validated it blindly against MOD's full-scale experiments, and demonstrated its application to a real vessel.

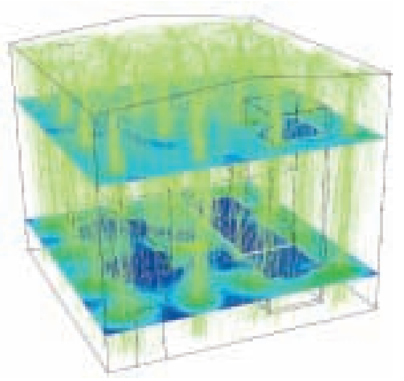
Because of the complexity of the application, the simulation involved a large number of software models that included existing capabilities, existing models that required some special functionality extended through FORTRAN™, and some models that were implemented entirely through FORTRAN. The simulation models were validated against data from a large-scale experimental rig.

Measurements of the FWS droplet initial conditions, in air and without fire, were commissioned at South Bank University (SBU), London, using high-speed photography. This provided information at a specified, small radial distance from the nozzle, for velocity (predominantly radial) and mass flow for each of a group of droplet-sized bands, as a function of azimuth and elevation. SBU performed measurements for two working fluids: water and water with 1 percent by volume AFFF. The university also measured to ascertain whether the additive affected the terminal speed of a droplet with a given mass. The SBU measurements were employed in the initial conditions for the particle transport model.

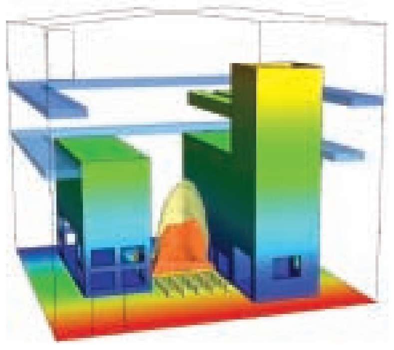
To determine how the fire becomes extinguished, the combustion model calculates the fuel evaporation rate from the heat delivered to the fuel by the fire. The model then predicts where and how rapidly fuel vapor is burned and heat is released exothermically. As the fire cools after spray initiation and radiation is attenuated by the spray, soot, and gaseous products (as well as the foam film when that is present), the heat returned to the pool of liquid fuel is diminished and so is the evaporation rate. If the spray system is appropriately designed, then extinction is achieved when combustion process ceases. Fuel vapor usually vanishes a short while after the fuel evaporation rate falls to zero.

The MOD and ANSYS research teams validated the fluids model by comparing it to data from a MOD experimental rig. The rig was large scale with a volume of 1,080 cubic meters. Inside the experimental rig there were mockups of the large equipment — diesel generator and gas turbine enclosures typically found within a Royal Navy (RN) machinery space. The FWS comprised 16 GW LoFLow™ K15 nozzles fixed on a 3-meter grid near the ceiling. Buckets at the floor were used to measure cumulative water delivery. Additional instrumentation was added to the space to enable validation of the model. Liquid fuel (F-76, which is a common fuel for shipboard diesels, gas turbines) was provided in one of two rectangular trays, having areas of 3 and 1.5 square meters, respectively. The teams validated the simulation against two separate tests: water spray for the larger tray and water spray with additive for the smaller tray.

The results of the validation were generally encouraging, and the predicted extinction times and method of extinguishment were reasonably predicted. There were some noticeable discrepancies, and there was evidence that building leakage (the effects of which had been studied in previous research by ANSYS) was an important factor in this regard. Other influences on the results of the model were identified: The fuel model used heptane rather than F-76; the coefficient of restitution was set at zero for water droplets so that when they hit structures they were removed from the model; and positioning of the mockup structures, fuel trays and nozzle positions represented a worst case.



Droplet trajectories and maps of water vapor mole fraction after spray inception



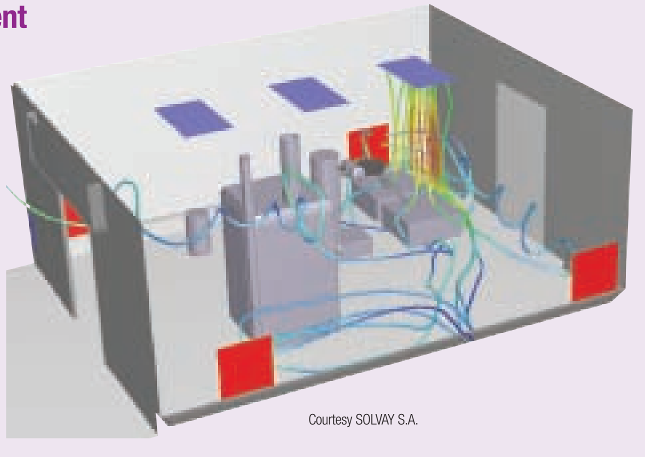
Simulation model of the rig geometry and temperature isosurfaces before spray inception

## Engineering Simulation for the Built Environment

The technology from ANSYS that can be applied to fire propagation, fire suppression and smoke management for ships, airplanes, trains, cars and trucks is also used for ventilation and thermal modeling in the built environment industry. These comprehensive multiphysics capabilities, which address safety and comfort concerns, are frequently used upfront during the design and construction of buildings.

In order to provide information for design improvement, design optimization and energy efficiency in the built environment, predicting conditions such as air velocity, temperature, relative humidity, thermal radiation and contaminants is extremely important. The simulation must also take into account ventilation, heat loss and solar radiation effects on the structure walls, roof, floors, windows and doors, as well as the presence and activity of people and equipment in these areas. Simple air flow modeling assists engineers and architects in quantifying and simulating the impact of structural and equipment design modifications on the thermal comfort of a space's occupants.

Engineering solutions from ANSYS provide a cost-effective and accurate means of designing efficient smoke management and detection systems. The unparalleled breadth of solutions across multiple disciplines provides the ability to quantify the behavior of materials subjected to fires or extreme heat and possible structural deterioration during catastrophic events. These can be analyzed in detail using explicit dynamics and structural modeling. Solutions from ANSYS allow for the analysis of events ranging from explosions that encompass blast waves (in the context of homeland security) to deflagrations in combustible mixtures.



Courtesy SOLVAY S.A.

— Thierry Marchal
Industry Marketing Director
Materials and Consumer Care, ANSYS, Inc.

For more information, visit www.ansys.com/industries/hvac.

| Descriptions of Models Used in the Simulation | | |
|---|---|---|
| **Model** | **Implementation** | **Purpose** |
| RANS turbulence modeling (SST) | Existing model | Determines turbulent transport |
| Laminar flamelet combustion modeling (Peters) | Extended current model | To include combustion modeling of heptane fuel and evaporated water vapor, with reduced set of species |
| Soot modeling (Fairweather et al.) | Implemented new model | Assesses impact of soot on infrared radiation and visibility |
| Transient Lagrangian particle transport model | Existing model | Assesses the impact of water spray on fire and fuel, with two-way coupling of mass, momentum, convective heat and radiant heat |
| Multiple droplet size groups | Existing model | Determines penetration since larger drops are better at penetrating key regions directly, small droplets evaporate quickly and can reach key regions by entrainment |
| Coupled fuel evaporation | Implemented new model | Calculates fuel burning rate |
| Subgrid droplet–congestion interactions | Implemented new model | Estimates direct removal rate of droplets by subgrid congestion |
| Soot scavenging by water droplets | Implemented new model | Determines how scavenging affects infrared radiation and visibility; also predicts delivery of scavenged substances to boundaries |
| Additive effects on water spray and fuel evaporation rate | Implemented new model | Predicts attenuation of radiant heat arriving at pool surface |

After completion of the validation, the model was successfully applied to a real machinery space aboard an RN ship. MOD is proposing the use of FWS in its future vessels for fire suppression that was validated by the experiment [2] and this work. ■

## References


[1] Sinai, Y., Staples, C., Edwards, M., Smerdon, M., "CFD Modelling of Fire Suppression by Water Mist with CFX Software," *Proc. Interflam* 2007, Vol. 1, 2007, pp. 323–333.

[2] Hooper, A., Edwards, M., Glockling, J., "Development of Low Pressure Fine Water Spray for the Royal Navy: Results of Full Scale Tests," Proc. Halon Options Technical Working Conference, 2004.


## Acknowledgments


This work was a team effort. The authors wish to thank Dr. J. Glockling of the Fire Protection Association, Dr. G. Davies and Prof. P. Nolan of South Bank University, as well as P. Guilbert, P. Stopford, H. Forkel and P. Everitt of ANSYS, Inc. for their contributions.