Third International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
10-12 December 2003

NUMERICAL SIMULATION OF WATER MIST SUPPRESSION OF TUNNEL-FIRE SCENARIOS

HART, R. A. [1], REN, T. X. [2], AND EDWARDS, J. S. [1]

[1] SCHEME, University of Nottingham, University Park, Nottingham. NG7 2RD. UK

[2] Formerly University of Nottingham, now with CSIRO, Queensland, Australia

ABSTRACT

Water-mist fire suppression systems offer a number of potential advantages over conventional sprinkler systems. A water-mist system uses very fine droplets of around 50μm diameter, which results in entirely different behaviour. In particular the water-mist remains suspended in air for long periods, and is influenced by the airflows caused by the fire and the general ventilation regime. In order to better understand the processes involved, a numerical model has been developed that models the effect of a water-mist fire suppression system on a typical tunnel fire scenario. A commercial available CFD package (*fluent*) is used. In this model, the water-mist is represented by a *discrete phase model* and the fire is modelled with a *non-premixed combustion model*. For simplicity a propane fire was used. Initial results show behaviour consistent with real fires and experiments. However limitations of the combustion model have been identified, which limit its suitability for this scenario.

KEYWORDS: CFD, Fire, Water-mist, Fire-suppression, Tunnel, Discrete Phase Model.

NOMENCLATURE

d	diameter (m)
C_D	drag coefficient
F	force (N)
g_x	gravity vector (ms^{-2})
Re	Reynolds Number
t	time (s)
u	velocity (m/s)
ρ	density (kg/m^3)
μ	viscosity

Subscripts

d	drag
p	particle

INTRODUCTION

Water is one of the most widely used fire-suppression agents as it is both effective in terms of it's thermal characteristics, and has a low environmental impact. In particular the phase change from a liquid to a gas can remove large quantities of heat, and in the right circumstances, reduce oxygen concentrations due to the build up of the water-vapour (Grant et al, 2000)

There has been a renewed interest in water based fire-suppression systems in recent years, due in part to new regulations requiring the retrofit of fire-suppression systems on commercial maritime vessels (SOLAS, 1997), and the phasing out of ozone-depleting Halons (UNEP, 1987). In particular this has lead to the development of *water-mist* systems, which have a *low-water demand*, and preserve some of the advantages of a 'total flooding' agent.

The water-mists utilised by these systems consist of very fine droplets, with diameters orders of magnitude smaller than the droplets created by conventional sprinkler systems – typically mean droplet diameters of 50-500μm. The small droplet size causes the system to behave in a very different way than a conventional system. Significantly the increased surface area to volume ratio of the system means:

- *Efficient heat-exchange*. Maximum effectiveness is achieved when all the applied water is able to evaporate. The high surface area of the mist means that each droplet is able to rapidly absorb heat from the fire environment.
- *Low terminal velocity*. For small droplets, aerodynamic drag forces are very significant. This results in a mist that is able to disperse throughout an enclosure like a gas, and which maybe carried around obstructions by the air. This aids the effectiveness of the system as a large proportion of the water-mist will eventually reach some part of the fire due to the convection currents caused by the fire.

Many experiments have shown water-mist systems to be very effective particularly against enclosures fires and in cases where *fuel splashing is undesirable*. However, the enclosure plays a significant role as it allows the mist to build-up, and permits the extinguishment of the fire by oxygen depletion. Experiments on open-space fires have shown that their successful suppression is more difficult, requiring a more targeted mist-delivery system. (Heskestad, 2003, Bill et al, 1997)

There is considerable interest in the potential of water-mist systems for underground mining applications due to the potentially devastating nature of fires in mine workings.

A mine tunnel is neither an enclosed space, nor an open-space, and could be described as *semi-open.* It would therefore be a completely new application of water-mist, and given the current limited understanding of the behaviour of water-mist, the effectiveness of the system on a tunnel fire, and the optimum operating parameters for the system needs to be investigated.

This study uses CFD (computational fluid dynamics) techniques to create a numerical model of a tunnel fire and water-mist system.

Figure 1. Shematic of Model Scenario

MODEL DESCRIPTION

Modelled Scenario

The model consists of a 2m square cross-section tunnel. A 24m length of the tunnel is modelled. A 5cm square fuel 'inlet' is located on the floor of the tunnel 8m from the intake. Water-mist sprays can be located at arbitrary locations (and orientations, etc) throughout the model. In this initial study these were located in a single row along the tunnel roof. See fig 1.

This scenario is intentionally kept as simplistic as possible however it should be able to produce results applicable to real-world fire scenarios.

Discrete Phase Model

A water-mist consists of a *very large* number of droplets, and it would be effectively impossible to simulate them all individually. Instead, a *discrete phase model* is used to model the water-mist, using a much smaller number of *tracking particles*. Each particle represents many similar water droplets.

The particles interact with the *continuous phase*, by a number of laws, which govern the transfer of heat, mass, and momentum. This transfer affects both the particles and continuous phase, and in the case of heat and momentum, it is bi-directional. This means that an extra level of iteration is required in the solution process (see Figure 2), which can lead to significantly longer run times and a reduction in stability then would be necessary for a single-phase solution.

Trajectories

The trajectory of a particle is found by considering the force balance acting on it:

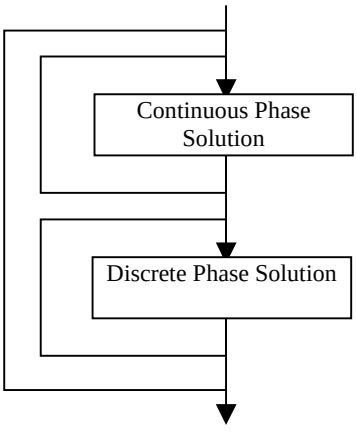


Figure 2: Solution process for coupled continuous & discrete phase

$$\frac{du_p}{dt} = F_D\left(u - u_p\right) + \frac{g_x\left(\rho_p - \rho\right)}{\rho_p} + F_x \quad (1)$$

Where $F_D\left(u - u_p\right)$ is the drag force per unit mass, g_x is the gravitational vector, and F_x represents other body forces (none applicable in this case)

$$F_D = \frac{18\mu}{\rho_p d_p^2}\frac{C_D \,\mathrm{Re}}{24} \quad (2)$$

Where d_p is the droplet diameter, C_D the drag coefficient, Re is the flow Reynolds number, and μ is the fluid viscosity.

The drag coefficient can be found in a number of ways. For the purposes of this study it is assumed that the droplets are spherical, and an appropriate derivation is used. (Morsi and Alexander, 1972)

The stream velocities (u) calculated for the continuous phase are mean values, and do not represent the fluctuations of velocity caused by turbulence. This can cause unrealistic results because it is these small-scale variations that are responsible for the dispersion of particles. For this reason a stochastic method (random walk) is used to determine the instantaneous gas velocity.

The force on each particle equates to a momentum source (or sink) in the continuous phase.

Heat & Mass Transfer

A discrete-phase particle is in one of three states, which controls how the heat & mass transfer is calculated:

- *Inert* – mass transfer is assumed to be zero, heat transfer based on conduction & convection.
- *Vaporising* – evaporation takes places, and is an additional source of heat transfer
- *Boiling* – temperature of droplet cannot increase and a corresponding increase in rate of evaporation occurs.

The state of a particle is determined by its temperature. The distinction between the inert state and the vaporising state is fairly arbitrary as the rate of evaporation will be correctly calculated as zero below the dew-point temperature in either.

In a similar way to the trajectory calculations the heat & mass transfer is bi-directional. E.g. Evaporation from the particle is a source of water vapour in the continuous phase.

Combustion

For reasons of simplicity, this study uses propane as the fuel for the fire. A solid or liquid fuel requires a separate

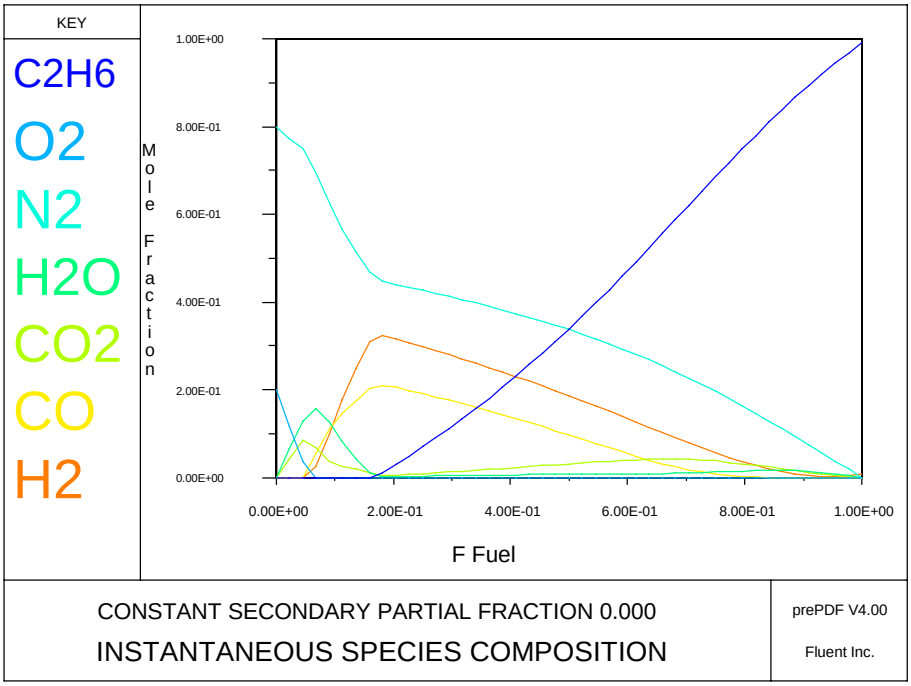


Figure 3: Pre-calculated Thermo-chemistry

volatilisation process, which adds unnecessary complexity to the model at this stage.

The complete combustion of propane has an standard enthalpy of -2220kJmol^{-1}, and produces carbon dioxide and water-vapour: $C_3H_8 + 5O_2 \rightarrow 3CO_2 + 4H_2O$

In a fire scenario, incomplete combustion may also take place, which produces other substances such as carbon monoxide.

The combustion is modelled using a *non-premixed combustion* model. This simplifies the thermo-chemistry to consider only three 'streams' of reactants (see Table 1). This considerably reduces the number of transport equations that must be solved and reduces the chemical kinetics to depend on only two mixture fractions and the instantaneous enthalpy. This allows the kinetics to be pre-computed, and merely looked-up during the CFD calculations. (See fig 3)

Species Considered	Carbon Dioxide, Carbon Monoxide, Nitrogen, Oxygen, Propane, Water vapour.
Fuel Stream	Propane
Oxidiser Stream	Nitrogen (80%), Oxygen (20%)
Secondary Stream	Water vapour

Table 1: Streams used by non-premixed combustion model

Details of Runs Carried Out

The initial run of the model, used a (relatively low) 0.1m/s air velocity, the fuel inlet was 1g/s of propane, and the water flow rate was 0.16kg/s/injection. There were 6 injection points, which were located in the centre of the tunnel roof, spaced at 1m intervals, between 3m and 8m from the inlet. The injection properties are shown in Table 2.

As the scenario is symmetrical, only half of the tunnel is modelled. A structured hexahedral mesh is used, which is refined around the boundaries, and in the volume around the fire and the water-mist nozzles. In total the mesh contains approximately 150,000 cells.

Injection Type	Conical
Number of Injections	6
Streams per Injection	6
Cone Angle	35°
Droplet Diameter	50μm
Initial Velocity	0.1 m/s
Initial Temperature	300 K
Activation Time	3 s

Table 2: Injection Properties in the initial model

A constant time-step of 0.02s was used. For each time-step one DPM particle per 'stream' is injected (i.e. 900 particles for each second modelled).

The second run was identical with the exception of the *water flow rate*, which was reduced to 0.016kg/s/injection. (i.e. by a factor of 10)

An each case an initial solution is found with the fuel and water flow-rate set to zero. Following this, the fuel inlet is activated, and the model is run (with no water-mist), in order to allow the combustion, and related airflow regime to develop. The injections were activated after three seconds, and the model was run for several more seconds in order to observe the behaviour of the mist, and it's effect on the fire.

Assuming complete combustion, the heat input from the propane is 50.45kW, and assuming all the water-mist is heated to 100˚C and evaporated the potential heat-absorption is 2460kW for the high-flow and 246kW for the low-flow scenario. In other words, approximately 2% and 20% respectively, of the mist must be evaporated to be effective.

Plots from the two runs are shown in the appendix. These show droplet and mid-plane air temperatures. Net airflow is left to right. The scale used is 300-500K for air, 300-400K for droplets.

Discussion of Results

In the high water-flow run, the tunnel rapidly fills with mist, the growth of the fire plume is immediately halted, and the heat is quickly removed from the tunnel as the mist evaporates. It seems likely that the fire would be easily extinguished by this configuration. The high-flow rate means that the momentum transferred to the air from the injected mist is significant and allows the mist to easily penetrate into the tunnel despite the low terminal velocity of individual particles.

In the low water-flow run, it takes much longer for the tunnel to fill with mist, and the nozzle located immediately above the fire has little effect on the fire plume. The introduction of the mist only downstream of the fire has a noticeable effect on the airflow and causes the plume to be drawn downstream. Once the body of the mist has filled the tunnel sufficiently it enters the plume and significantly cools it. However after an initial high, the rate of evaporation falls, and the plume is able to re-establish. It is unclear if a steady-state would be reached, or how long that would take.

Problems with combustion model

The non-premixed combustion model, assumes *fast-chemistry*, and consequently the model will *never predict extinction*, and will *over-estimate* the rate of reaction close to extinction, because at *low* temperatures, the combustion reaction is limited by the Arrhenius equation, and not by the fuel-oxidiser mixing process.

In addition the limit of three fixed streams effectively limits this model to gaseous combustion, as the secondary stream is used to represent the water vapour.

The alternative to the non-premixed combustion model is to use a *generalised finite-rate* chemistry model. Unlike the *non-premixed* model, the generalised model is not limited to any number of species (although each additional species adds to the solution time and storage requirements of the model). The rate of reaction is calculated using a hybrid method that considers both the Arrhenius rate and the *Eddy Dissipation Concept* (EDC), and uses the lower predicted rate. This allows both slow and fast-chemistry behaviour, and should allow extinction to occur.

CONCLUSIONS

Suitability of Numerical Model

Despite the problems with the combustion sub-model used, this study shows the potential of Numerical Modelling of water-mist systems. Useful results were obtained in a reasonable amount of time, showing that CFD techniques could conceivably be used as a design tool. In particular, the discrete phase model allowed the complex interaction between the water-mist and the tunnels air-flow regime to be simulated and easily visualised.

Effects of Water Mist in a Tunnel Fire Scenario

The results show a few different ways in which the water-mist affects the fire.

Effects on airflow

Momentum Transfer - The water droplets are injected into the tunnel with an initial velocity. As the droplets are small, the aerodynamic drag force is quite strong, and the droplets quickly slow down, transferring their momentum to the air. In the high water flow study this effect had a noticeable effect on the air currents around the fire. This in turn allows the mist to penetrate more rapidly into the tunnel.

It should be noted that in this study, the initial droplet velocity used was fairly arbitrary, and there is a significant variation in water-pressures (and hence initial velocities) used by the various available types of water-mist system.

Natural Convection - The heating and evaporation of the droplets cools the air. As the mist is injected at high level, this can increase (or counteract) the convection currents caused by the fire. This alters the airflow to the fire, which could significantly alter the rate of combustion in itself.

Effects on combustion

Evaporation occurring close to the fire removes heat and produces water vapour both of which will inhibit the rate of combustion.

The non-premixed combustion model is not suitable for predicting extinguishment, and so no precise quantitative conclusions concerning extinguishment can be drawn.

ACKNOWLEDGMENTS

The authors would like to acknowledge the financial support of the ECSC for this research.

REFERENCES

BILL, RG., HANSEN, RL., and RICHARDS K., (1997), “Fine-spray (water mist) protection of shipboard engine rooms.” *Fire Safety Journal*, 29(4):317-336

FLUENT (2001), *Fluent 6 User Guide*, Fluent Inc.

GRANT, G., BRENTON, J., and DRYSDALE, D., (2000), “Fire Suppression by Water Sprays”. *Progress in Energy and Combustion Science*, 26:79-130.

HESKESTAD, G. (2003) “Extinction of gas and liquid pool fires with water sprays”, *Fire Safety Journal* 38:301-317.

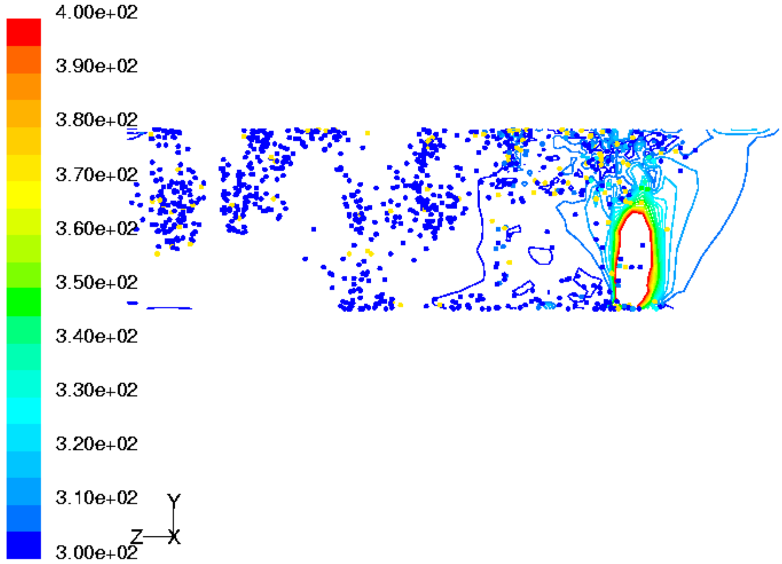
MORSI, S.A. and ALEXANDER, A.J., (1972), “An Investigation of Particle Trajectories in Two-Phase Flow Systems.” *J. Fluid Mech.*, 55(2):193-208,

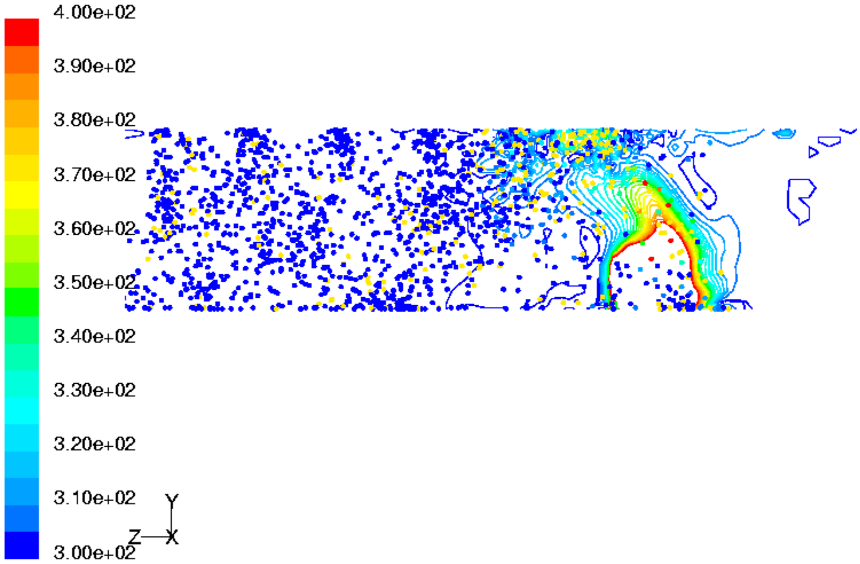
SOLAS (Safety Of Life At Sea) regulations (1997), “Construction: fire protection, fire detection and fire extinction,” chap. II-2. 1997 Consolidated Edition, International Maritime Organisation, London, July 1997

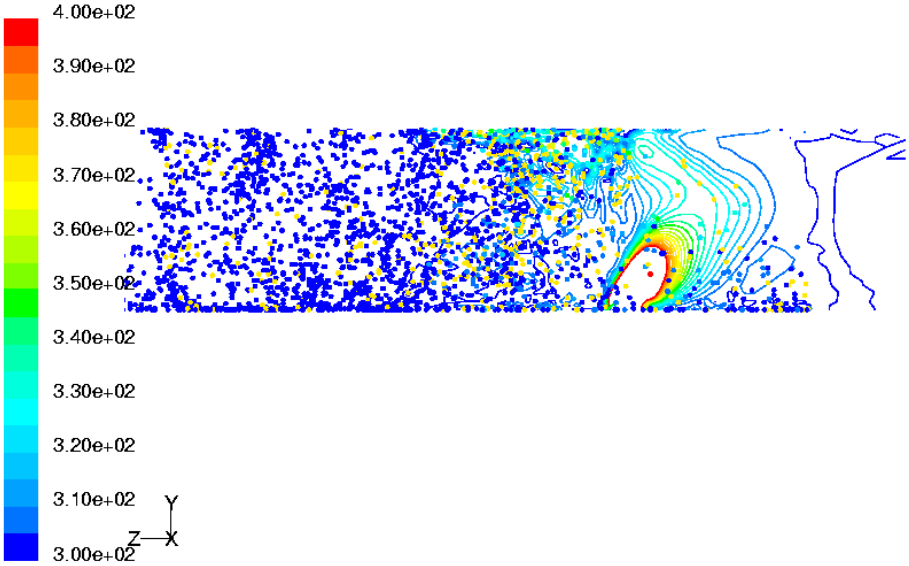
UNEP (1987) “Montreal protocol on substances that deplete the ozone layer.” Final Act, Montreal, Canada, UNEP, September 1987

APPENDIX A

High flow (0.16kg/s) (at 1,2 & 3 seconds after mist activation)







Low flow rate (0.016kg/s) (at 2, 4 & 5 seconds after mist activation)