Fifth International Conference on CFD in the Process Industries
CSIRO, Melbourne, Australia
13-15 December 2006

THE MODELLING OF FIRE SPREAD AND SUPPRESSION WITHIN UNDERGROUND MINE TUNNELS

Ian LOWNDES [1], Hervé MORVAN [2], Stephen SILVESTER [1] , Stephen PICKERING[3], Robert HART [1] and Yixiang CAI[1]

[1] Nottingham Mining and Minerals Centre, School of Chemical, Environmental and Mining Engineering
[2] School of Civil Engineering, [3] School of Mechanical, Materials and Manufacturing Engineering
University of Nottingham, UNITED KINGDOM

ABSTRACT

Underground fires in deep coal mines are an ever present hazard to personnel and production. To initiate the safe evacuation of personnel and the salvage of the mine infrastructure requires the successful and timely detection of the seat and nature of the fire and the initiation of suitable fire fighting systems.

In recent years, conveyor belt installations have been recorded as the predominant source of fires within deep UK coal mines. This paper reports the results of an experimental and computational study conducted to characterise the initiation and spread of fire along a conveyor belt within a ventilated full-scale experimental fire test gallery. A novel modelling method is proposed to represent the observed flame spread along the conveyor belt surfaces.

This paper reports the results of an experimental and computational study conducted to characterise the initiation and spread of fire along the upper and lower surfaces of a conveyor belt mounted within a ventilated full-scale experimental fire test gallery. Experimental data was obtained from the instrumented fire test gallery that recorded the temperature gradients, and airflow profiles produced within the gallery due to the spread of the flame front under various ventilation flow rates. The computational models were constructed using the three-dimensional commercial CFD software code, FLUENT™. A novel modelling method is proposed to represent the observed flame spread along the conveyor belt surfaces. It is concluded that this model was able to qualitatively replicate the flame spread observed on the belt surfaces within the test gallery.

In many conveyor roadways deluge water sprinkler systems are commonly installed. The early detection of a fire by strategically positioned banks of monitoring sensors trigger an initiation of this sprinkler system. Recent research studies have suggested the potential suitability of water mist systems to tackle fires within mine galleries. The paper presents an analysis and discussion of the results from a computational modelling study that has recently investigated the potential application of water mists to extinguish fires in mine roadways.

NOMENCLATURE

A pre-exponential factor, 1/sec
$C_{i,s}$, vapor concentration at the droplet surface, kgmol/m^3
$C_{i,\infty}$ vapor concentration in continuous phase, kgmol/m^3
C_p specific heat, J/kg K
E_A activation energy, kJ/kg-mole
h enthalpy, J/kg
h_f convection coefficient, kJ/m^2-sec K
k thermal conductivity ,W/m K
k_c mass transfer coefficient, m/s
m mass, kg
Ni molar flux of vapor, kgmol/m^{2}s
Nu Nusselt number
n number of spatial nodes
Q heat of decomposition, J/kg
q heat flux, kJ/m^2-sec
R gas constant, 8.314kJ/kg-mole k
t time, sec
T temperature, K
x spatial co-ordinate, m
ρ density, kg/m^3

Subscripts

c char
f fluid, fuel
g gas
p particle
s surface
rad radiation
w wall

INTRODUCTION

Underground fires represent a constant threat to the safety of underground personnel. Miners in the immediate vicinity may face intense heat, blinding smoke, toxic fumes, fall of ground and other direct effects of a fire. However, the vast majority of victims never actually see the fire, and are overcome either by deadly fumes in the ventilating current or by asphyxiation. The mine ventilation system, which maintains a sustainable atmosphere to the working places, can transport products of combustion with equal efficiency. Miners who are remotely situated from a fire may be forced to evacuate considerable distances, through dense smoke and fumes, or even become cut-off from escape. Additionally, miners may become confused by unfamiliar ventilation characteristics caused by fires.

Conveyor belts are widely used for transporting minerals in mines as well as in many other industrial situations. Because they contain large amounts of polymeric materials their use in certain environments, especially coal

mines, but also in steel works, power stations and other enclosed areas must be controlled to minimize the fire risk. Over last nine years there have been sixty fires in underground mines in the UK alone and that of these, half have been associated with conveyors.

This paper summarises the results obtained from a recent research project that has developed a preliminary computational thermal and fluid dynamic model to represent the spread of flame along a typical conveyor belt used in UK deep coal mines. The results of these models have been calibrated against a series of experimental tests performed within a surface fire test gallery. An improved knowledge of the burn characteristics and rate of release of the products of combustion from conveyor belt fires will assist in the development of better fire detection and fire fighting techniques.

Three dimensional commercial CFD codes have been used to simulate and predict the effects of fires. However, the physical simulations produced by these CFD models, need to be validated against the experimental data, i.e. the accuracy of the predicted model parameters needs to be assessed against the corresponding measured experimental data. In addition, a range of initial and boundary values required by the computational models need to be determined from physical observation and experiment.

INITIAL APPLICATION OF CFD TO CONVEYOR BELT FIRE SPREAD PREDICTION IN MINES

An initial computational modelling programme was initiated to evaluate the use of a commercial CFD code to successfully replicate the spread of flame along a test section of conveyor belt placed on a trestle and heated and finally ignited by a manifold of propane burners positioned above and below the leading edge of the belt on the trestle. The detail and the results of this initial modelling programme are detailed in the literature (Lowndes et al, 2005 and Lowndes et al, 2006). The simulation results qualitatively replicated the steady spread of flame observed along the surface of the belt material during the experimental studies. However, these initial models were not able to satisfactorily replicate the onset of char burn following devolatilisation that may lead to a 'burn through' of the belt conveyor material. Consequently, a recent extension to this initial research work has been initiated to construct a discretised solid burn model that is able to replicate both the aerodynamics of a belt within a ventilated tunnel, and the physical characteristics of a heated belt, devolatilisation, ignition, flame spread, char burn and burn through.

FLAME SPREAD ACROSS HORIZONTAL SURFACES

In the combustion of solid materials, as a solid is heated by an external burner or by frictional heat, the temperature of the solid materials will rise. Once the temperature of the solid reaches a critical value, volatile gases will be released from solid. These gases will take the heat from the solid materials and affect the heat transfer in the solid materials. This process is called devolatilisation.

Following a critical review of the literature it was decided to adapt the devolatilisation model developed by Henderson et al (1985, 1987) to numerically simulate the devolatilisation of polymer composites exposed to high temperatures. This model assumes:

- no accumulation of decomposition gases in the solid material
- no thermo chemical expansion;
- a thermal equilibrium exists between the decomposition gases and the solid material.
- no intermediate compounds are produced when the resin materials sublimes;
- thermal and transport properties of the laminate are constant during the decomposition process of the material;
- a zero value for the final density of the resin material is applied.

Based on the assumptions, the following basic governing equation of their model was deduced:

$$\rho C_p \frac{\partial T}{\partial t} = k\frac{\partial^2 T}{\partial x^2} + m_g \frac{\partial}{\partial x} h_g - \rho A \exp\left(-\frac{E_A}{RT}\right) \times \left[Q - h_c + h_g\right]$$

The equation is an extended equation of the basic conduction equation. The term in the left side of the equation stands for the time-dependant heat changes in each cell of the material, while the first term, second term, and the third term on the right hand side represent the heat change due to conduction, movements of thermal mass of gases and decomposition, respectively.

This model requires the behaviour of the thermal properties, e.g. conductivity, specific heat, with respect to temperature to be determined by performing a series of experimental characterisation studies. The relationship of the thermal properties on the temperature was then specified by the use of empirical curve fitting methods.

In addition, an nth order kinetic rate equation was used to calculate the mass changing rate inside the discretised grids representing the solid polymeric belt.

$$\frac{\partial m}{\partial t} = -A \cdot m_0 [(m - m_f)/m_0] \exp(\frac{E_A}{RT})$$

To simulate the heating of the belt by external propane burners convection and radiation boundary conditions were applied to the discretised numerical model of the belt. This model can be further improved by taking into account the effects of material expansion and thermal chemical reaction rate.

Combination of one dimensional devolatilisation model and FLUENT

The devolatilisation model was later combined with a combustion model in FLUENT. In this comprehensive model, the combustion and flow movements in the tunnel were simulated using existing FLUENT combustion model. The devolatilisation process in the solid was encoded and solved as a User Defined Function (UDF) in FLUENT. An idealised two dimensional case was firstly considered using a low resolution grid of 3000 cells specifically for evaluating the performance of the UDF. The geometry is created as following: The two metre

length of conveyor belt and was centrally located on a trestle within a twenty-metre long wind tunnel. Two propane burner manifolds were located above and below the belt, 0.2 metre from the leading front edge of the belt. The wind tunnel was ventilated by a uniform flow field of 1.5 m/s.

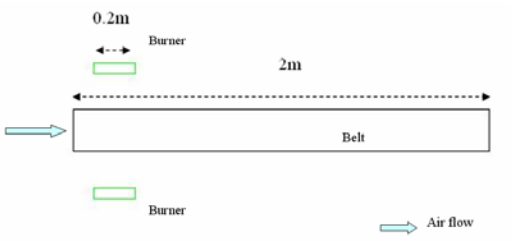

Figure 1: Systematic view of the geometry in two-dimensional model

The behaviour of the airflow, the propane combustion and resultant heat transfer within the tunnel and around the belt and burners is determined by the FLUENT computational fluid dynamic code. The discretisation of the air flow domain was achieved using a mixed grid scheme. The solid belt surface was divided into 40 parallel cells running vertically through the section of the belt. For this initial model it was assumed that was negligible lateral heat transfer between adjacent belt cells along the belt, as compared to the heat transfer experienced through the thickness of the belt. Each belt cell was further sub-divided into a number of smaller cells across the thickness of the belt thickness. The heat transfer and mass transfer across each cells due to the ignition of the propane burners is determined by a solution of the one dimensional devolatilisation model. The following figure shows the heat and mass transfer across the belt:

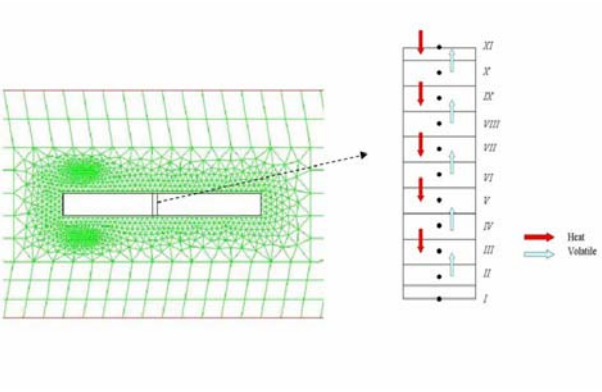

Figure 2: The airflow and belt domain discretisation schemes

To solve these models requires a coupling of the surface temperature of the belt. This temperature is defined by both the net heat transfer across the airflow domain and by the heat transfer experienced within the solid belt. To solve this model, a function called Define_Profile was created in FLUENT to determine the change in temperature of the solid surface of each cell.

The interface and exchange of data between the airflow and the solid belt domain models is defined by User Defined Functions (UDFs). The figure below illustrates the combination and data exchange between these two domain models:

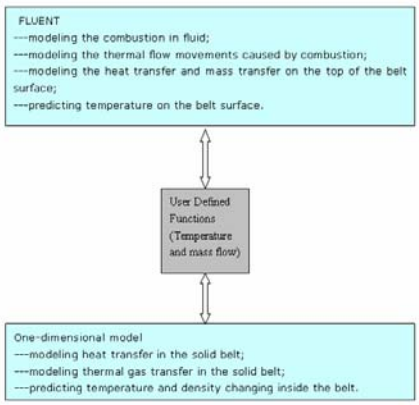

Figure 3: The coupling of data exchange between the two models using a UDF

The heat flux to the solid belt wall from an adjacent fluid cell is computed as:

$$q = h_f (T_w - T_f) + q_{rad}$$

The heat flux data was later used in the UDF as input data for the boundary cell energy balance calculation.

$$T_s^{'} = T_s + dt \times (q - k\frac{T_1 - T_s}{dx}) / \rho C_p$$

where T'_s is the surface temperature of each belt cell at the next time step. Following the calculations performed by the UDF, the value of the surface temperature was returned to FLUENT code to represent the surface temperature of the belt surface.

The mass flow rate of the devolatilised gas is another parameter exchanged between the fluid flow domain and the solid belt. The gas created from the heating of the solid belt will migrate across the solid-fluid interface and take part in the surface combustion reactions once the ignition temperature is achieved. In the one dimensional model of the solid belt, it was assumed that there was no mass accumulation of gas when gases travel from within the belt to the surface. Thus, the mass flow of released gas is calculated using the following equation:

$$(\dot{m}_g)_n^{t2} = -\sum_{i=1}^{n}\left(\frac{\rho_i^{t2} - \rho_i^{t1}}{\Delta t}\right)\Delta x_i$$

To define the mass transfer from the solid to the fluid, a small interfacial layer was created on the surfaces of the belt.

The macro function Define_Source in FLUENT was used to define the amount of the gas emitted into each interfacial surface cell. The mass exchange rate into each

cell was determined using a first order temperature-dependant Arrhenius equation.

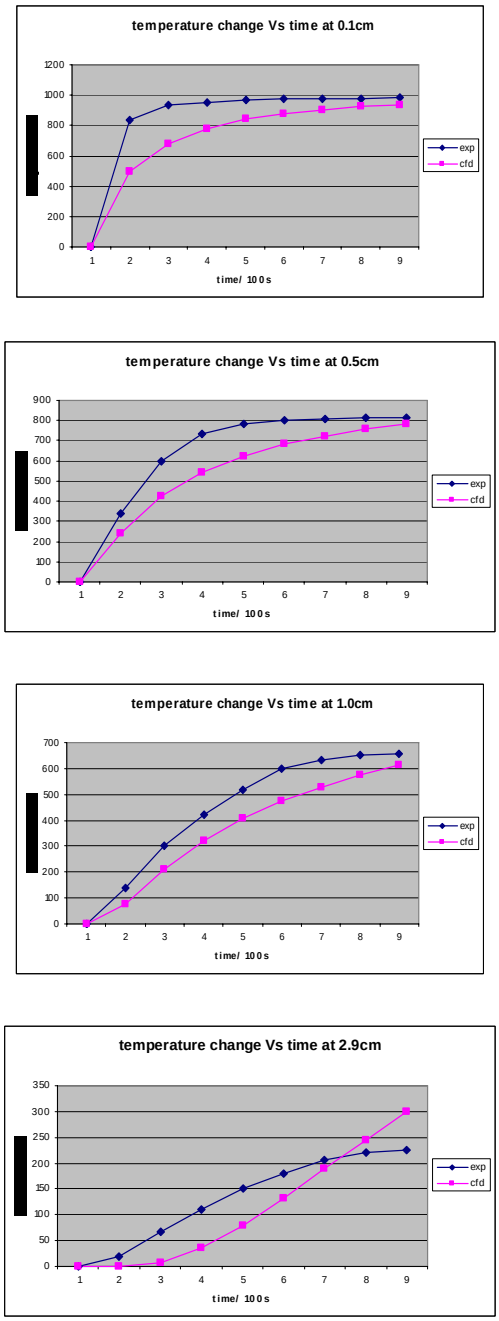

Figure 4: Comparison of the experimental and predicted temperature profiles across the thickness of the belt

The results obtained from the above coupled model were compared to the experimental results obtained by a study by Henderson et al. (1985). A comparison of the measured and predicted temperature profiles across the composite was conducted. An analysis of the data presented on Figure 4 shows that the temperature at the depth of 0.1cm, 0.5cm, and 1.0 cm reached a close level to that from experiments after 900 seconds. However, there appeared a higher temperature in the depth of 2.9cm, where is near the lower surface. The results can be further improved by carrying out further investigations to determine the relationship between thermal properties changes and temperature.

Work is now under progress to develop a full 3D grid independent extension to the above model that will both simulate flame spread along the surfaces of the belt and include a char burn model to allow for the simulation of belt *burn through*.

WATER MIST

The interest in water mist as a fire fighting technology has initially been driven by its potential as a replacement for environmentally harmful halon-based systems which have now been banned, with strong backing from the militaries for nautical applications in particular. Recent research has suggested the potential suitability of water mist systems as an alternative to the more conventional deluge water sprinkler typically found in mine galleries. Such an application has therefore been investigated numerically by the writers. Their method and results are presented below.

Use of mist in tunnels – Experimental data

The only known experimental testing of water mist in tunnels was carried out in Germany by DMT as part of the European Coal and Steel Community project ECSC-PR/094 (DMT, 2004). These experiments employed a pair of conveyor belts ignited by a 150 kg wooden crib located below the belts at the inlet end of the tunnel. Water mist was supplied by two rows of four nozzles located 2.5 m above the tunnel floor (1.2 m above the upper belt) and approximately 1.5 m apart. Typically each nozzle had a flow rate of 0.17 kg/s. Tests were performed with ventilation rates equivalent to inlet velocities of 0.8 m/s, 1.2 m/s and 2.4 m/s.

CFD MODEL OF FIRE SUPPRESSION BY WATER MIST

CFD models of the various DMT set ups (DMT, 2004) were built in FLUENT using the following options:

- Grid independent hex mesh (422,000 cells)
- Standard k-ε model with added buoyancy terms
- Isothermal walls
- Velocity inlet and pressure outlet
- Eddy Break-Up (EBU) combustion model
- Discrete Ordinance (DO) radiation model
- Second order discretisation in space and time

The fuel for the numerical simulation of the DMT Gallery 4 case was taken to be $C_{19}H_{30}$ for its heavy fuel properties (DMT, 2004). The rate of fuel release was adjusted at 10^{-3} kgmol/s to match the 11 MW heat release reported.

Validation

Both the combustion and mist models were tested and validated for a simple enclosure case based on Kym and Riou (2003) prior to their applications to the present tunnel case.

The combustion and heat transfer set-up was also validated independently for tunnel cases, first against Wu

and Barkar's critical velocity work (2000) and then for horizontal, upward and downward flows, at two ventilation rates, using the thermocouple data supplied by DMT (DMT, 2004).

A sample of results for the DMT case is shown below, Figure 5, for illustration purposes. Sensitivity of the results to the physical set-up (below the existence of a lip on the pan for instance) was thoroughly tested. Test included simulations with soot models, modified radiation models and boundary conditions. More complete results are available in Hart (2006).

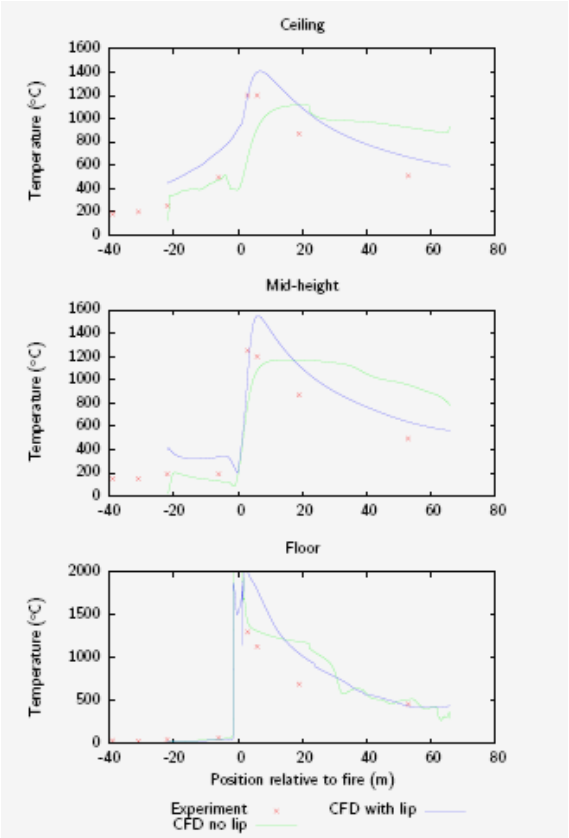

Figure 5: Comparison between CFD predictions and data with a ventilation velocity of 0.8 m/s

Mesh independence was also ascertained for each simulation. For the DMT case several meshes with grid cell number ranging from 129,000 to 280,000 were tested, with 213,000 found to be the optimal resolution (Gallery 4).

Mist model

The mist model implemented relies on FLUENT discrete phase model (DPM), a coupled (here) Lagrangian particle tracking algorithm. However each mist particle is not modeled individually; the tracked particles are used to represent several thousands of similar droplets instead.

The evaporation of the droplet into the continuous phase is governed by gradient diffusion from a vapor concentration at the droplet surface, $C_{i,s}$, to the same concentration in the continuous phase, $C_{i,\infty}$:

$$N_i = k_c\left(C_{i,s} - C_{i,\infty}\right)$$

The concentration of vapor in the bulk gas is known from the continuous phase calculation and the concentration at the droplet surface is calculated by assuming the partial pressure of the vapor is equal to the saturated vapor pressure at the particle temperature:

$$C_{i,s} = \frac{p_{sat}\left(T_p\right)}{RT_p}$$

The mass transfer coefficient, k_c, is calculated from the following Nusselt correlation:

$$Nu_{AB} = \frac{k_c d_p}{D_{i,m}} = 2.0 + 0.6 \times \mathrm{Re}_d^{1/2}\, Sc^{1/3}$$

Mist model parameters

The results presented below are for the DMT Gallery 4 of cross section 9.3 m^2. The specifications for the water mist are taken to be the same as Kim and Ryou (2003):

Sauter Mean Diameter (SMD)	121μm
K-factor	1.66
Spray pattern	Hollow cone
Spray angle	70-90°
Spray velocity	14.1 m/s
Trajectory	DRW

Given the practical constraints governing these properties, these are left unchanged, leaving only the number and location of nozzles as the design parameters.

Since the fire heat release rate is known, 11MW, it is possible to calculate the water mist supply rate needed by assuming an energy balance between the heat of evaporation and the rate of combustion for this particular case. It is computed to be 2.04 kg/s (maximum).

Results

In order to provide protection an arrangement of nozzles must be determined that allows water droplets to reach the fire. Initial tests were therefore carried out without combustion and heat transfer model for the various ventilation rates of interest in order to ascertain the nozzle arrangement, Figure 6.

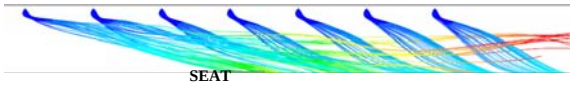

Figure 6: Spray pattern for a single line of nozzles at 3-m spacing. Air inflow 2.4 m/s, water flow rate 1 kg/s, initial droplet velocity 14.1 m/s

It was found that the distribution of mist depends on many factors, particularly droplet size, total mist momentum and air flow velocity. A single line of nozzles at 3-m spacing was also found to produce a reasonable but not perfect distribution of mist and it was subsequently decided to use a 1.5 m interval. However it was not clear whether a

single row of nozzles would be sufficient for mist to reach the full width of the DMT tunnel.

Simulations were subsequently carried out with combustion and heat transfer models activated. The two figures below illustrate the writers' results:

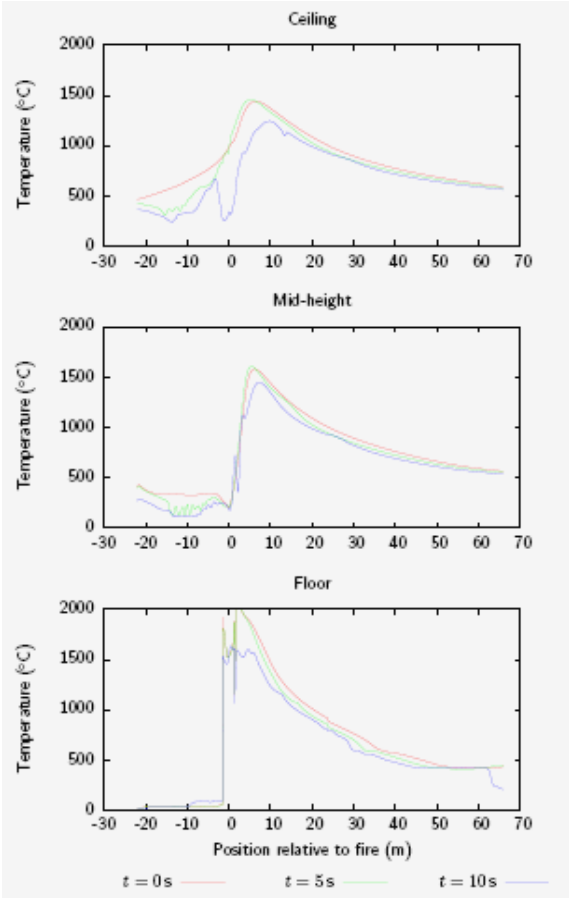

Figure 7: Reduction of temperature with time for a ventilation velocity of 0.8 m/s with single row of nozzles at 1.5 m spacing

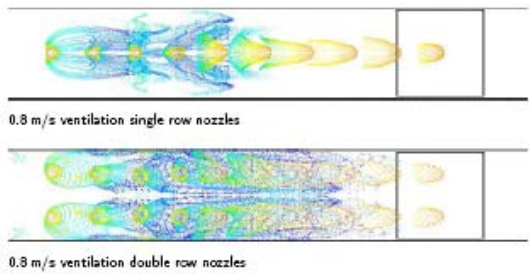

Figure 8: Distribution of mist in the tunnel 5s after activation for a ventilation velocity of 0.8 m/s

The findings from this series of simulations lead to the following conclusions:

- The action of mist on a tunnel fire varies significantly with ventilation velocity, and entirely distinct behaviors are seen for sub-critical and supercritical flow.
- For sub-critical flow, there is a strong interaction between the mist and the modeling of water mist fire suppression hot backflow. This can be beneficially as it caused a significant reduction in oxygen levels at the fire, however it also resulted in a reduction in the concentration of mist reaching the fire.
- For super-critical flow, the smooth flow regime upstream of the fire leads to a well distributed descending layer of mist that eventually reaches the seat of the fire. Downstream of the fire, there is significant evaporation within the fire plume, slightly reducing temperatures, but not otherwise influencing the fire.
- Complete extinction of the fire is not observed in any case considered. A reduction in the rate of reaction of around 40% is observed.
- Use of two parallel rows of nozzles gives a better distribution of mist in the tunnel than a single central row.
- Doubling the mist flow rate, increases the overall cooling due to the mist, but has only a minimal effect on the fire itself.

REFERENCES

DMT, (2004), "Fire fighting systems – Technical report 2" ESCC Coal RTD Programme. Contract 7220-PR/094.

HART, R., (2006) "Numerical modelling of tunnel fires and water mist suppression", PhD thesis, University of Nottingham.

HENDERSON, J. B., WIECEK, T.E. and TANT, M.R. (1985), "A model for the thermal response of polymer composite materials with experimental verification.", Journal of Composite Material, 19, 37, 575-595.

HENDERSON, J. B. and WIECEK, T.E. (1987), "A mathematical model to predict the thermal response of decomposing, expanding polymer composites.", *Journal of Composite Material*, 21, 373-393.

KIM, S.C., RYOU,H.S., (2003), "An experimental and numerical study on fire suppression using a water mist in an enclosure", *Building and Env*,, 38, 1309-1316.

LOWNDES I. S., SILVESTER S A., GIDDINGS D., PICKERING S.J. and HASSAN A.,(2005) "The computational modelling of conveyor belt fires in mine galleries", *Proc. 8th Int. Mine Vent. Cong*., AusIMM, Brisbane, Australia, 299-308.

LOWNDES I. S., SILVESTER S A., GIDDINGS D., PICKERING S.J. and HASSAN A., (2006), "The computational modelling of flame spread along a conveyor belt", Manuscript submitted for Publication in the *Journal of Fire Safety*

WU, Y., BARKAR, M.Z.A, (2000), "Control of smoke flow in tunnel fires using longitudinal ventilation systems – A stud of critical velocity", *Fire Safety Journal*, 35, 363-390