1. INTRODUCTION

Spray cooling is widely applied in metallurgical processes. In the spray cooling of the gas, the nozzle ejects the water droplets into the hot gas flow where the droplets are dispersed and evaporated and the hot gas is then cooled by providing the latent heat for vaporization of droplets and the apparent heat for air, water and vapour. In this process, the gas flow is influenced by evaporation cooling and aerodynamic acceleration due to the spray. The behaviour of droplets from the spray is in return influenced by the main flow. The interaction between the gas flow and the spray makes the spray cooling processes very complex.

In spray cooling of metallurgical systems it is usually desirable to have all water droplets completely vaporized to avoid blockage problems in subsequent equipment and it is not desirable to have too many water droplets hitting the duct wall because this may cause corrosion problem. While the performance of the spray cooling systems depends on the nozzle behaviour such as droplets size distribution and nozzle capacity, nozzle configuration such as nozzle location and injection direction, a good understanding of the spray cooling behaviour is important for designing and operating the spray cooling systems applied in metallurgical processes.

Modelling of the spray cooling processes using CFD techniques can provide detailed information of flow and thermal fields, from which the mechanisms of the processes can be clarified and optimization of the configuration of the spray system can be achieved. Numerous studies have been carried out for spray systems. Xia and Kankaanpää [1] presented a comprehensive review on the spray and droplet performance. Water spray cooling system has been used in gas turbine to improve power output and efficiency [2,3]. Spray cooling has also been used in incineration processes [4,5]. Misting and sprinkler sprays are often used in fire fighting [6,7]. In spray forming molten metal is atomized and accelerated by a high velocity gas and the resulting spray of metal droplets is directed onto a substrate where the droplets solidifies and consolidates to form a high density preform [8,9]. In spray drying process, the feed is atomized into a chamber where the resulting spray mixes with a hot gas, and the liquid component of the spray is then evaporated to produce dried particles. There have been numerous experimental and CFD studies on the spray drying [10,11]. However, in the author’s knowledge there appears sparse CFD modelling related to the present work treating with spray cooling of the off-gas. The present paper aims to numerically investigate the cooling of the off-gas by a water spray in a straight duct so as to obtain detailed spray cooling behaviour and examine the effect of the nozzle capacity on the flow in the case of cross flow situation with nozzle mounted at the duct wall.

2. MATHEMATICAL MODEL AND SOLUTION

The spray cooling case considered is the cooling of the off-gas by a water spray in a straight duct, as shown in Fig.1. The duct is 9.55 m long and 2.1 m in diameter. The nozzle is located at the duct wall, 1.55 m downstream of the inlet, and the spray is in the gravitational direction. Cross flow is formed between the spray and the off-gas. The gas cooling flow by an air atomizing nozzle is modelled by using the Eulerian-Lagrangian approach. In this procedure, a steady state gas flow in the flow domain without droplets is first solved. Then, the discrete-phase spray injection by air atomizer is created and the inter-phase coupling is initiated. Thereafter a coupled, unsteady cooling problem by water spray is solved. The droplet performance is computed by the discrete phase model (DPM). A number of droplet streams...
are chosen and each stream represents a mass flow rate and thus a droplet number per unit time. Each of these droplets is tracked through the flow using local time-averaged gas velocity predicted by the gas phase model and the droplet positions are iterated within each time step and updated as the solution advances in time.

The dispersion of the droplets due to turbulence is modelled using a stochastic discrete-particle approach. The trajectory equations for individual particles are integrated, using the instantaneous fluid velocity, along the particle path during the integration. In this way, when the number of representative particles is sufficient enough, the random effects of turbulence on the particle dispersion may be taken into consideration.

The effect of droplets on gas phase is taken into account by the particle-source-in-cell method. Within each control volume a droplet passes through, the amount of mass, momentum, and heat transferred from the droplet to the gas phase is calculated, and the sum of the amount transferred by all droplets tracked constitutes the total source terms, which are included into the gas phase calculation. This process including the gas phase, droplet behaviour, and source term calculations and gas phase flow recalculation is repeated until the criteria for convergence is satisfied.

2.1 Gas Flow

The cooling of the off-gas with a water spray in a horizontal straight duct is simulated. The gas phase is treated as a continuous phase and the droplets as a dispersed phase. The ideal gas law is applied to the off-gas phase. The conservation of mass, momentum, and energy of the continuous phase, the gas phase can be described as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = M_n
\]

\[
\frac{\partial \rho \mathbf{V}}{\partial t} + \nabla \cdot (\rho \mathbf{V} \otimes \mathbf{V}) = -\nabla \rho_p + \rho \mathbf{g} + \mathbf{F}_n
\]

\[
\frac{\partial \rho \mathbf{h}_p}{\partial t} + \mathbf{V} \otimes (\rho \mathbf{h}_p) = \mathbf{V} \cdot (-\mathbf{f} + \rho \mathbf{g} + \mathbf{F}_n)
\]

Here \( C_p \) is the specific heat, \( g \) the gravitational acceleration, \( h \) the enthalpy, \( k \) the thermal conductivity, \( V \) the velocity, \( \mu \) the turbulent viscosity, \( \rho \) the density, and \( \tau \) the stress tensor, respectively. The subscript \( p \) denotes the gas phase. \( M_n \) is the source term of mass coming from the dispersed phase due to droplets evaporation and is calculated from the change of mass of a droplet when it passes through each control volume. \( F_n \) is the overall external force resulting from the interaction between continuous gas phase and dispersed phase and is calculated from the sum of the changes in the momentum of droplets which pass through each control volume. \( h_n \) is the heat exchanged between the gas phase and the droplets. The details about these source terms can be referred to refs. [12-14].

For the case considered, the estimated Reynolds number reaches about \( 4 \times 10^5 \), the flow is then turbulent. Turbulence is modelled using the \( k-\varepsilon \) turbulence model [15].

2.2 Spray Dispersion and Evaporation

The spray may be presented by a number of droplets tracked by Lagrangian method [16,17]. The motion equation of a droplet is as follows:

\[
\frac{dV_p}{dt} = \frac{1}{2} C_d A_p \rho (V_f - V_p) V_p - V_p \mid + m_r g + F_r
\]

The drag coefficient for the droplet, \( C_d \), depends mainly on the Reynolds number based on the droplet slip velocity:

\[
Re = \frac{dV_p}{\mu}
\]

(4)

The equation of heat balance of a droplet may be presented by:

\[
m_r \frac{dT_p}{dt} = h_A (T_f - T_p) + h_b \frac{Dm}{Dt}
\]

(6)

The heat transfer coefficient, \( h \), is calculated by the correlation of Ranz and Marshall:

\[
Nu = 2.0 + 0.6 Re^{0.8} Pr^{0.3}
\]

(7)

When the droplet temperature reaches its boiling temperature, a boiling rate equation is applied and the droplet remains at a fixed temperature \( T_{bd} \):

\[
\frac{Dd}{Dt} = \frac{4k}{\rho C_{pd} d} (1 + 0.23 \sqrt{Re}) \left[ \frac{C_{pd} (T_f - T_p)}{h_b} \right]
\]

(8)

The turbulent dispersion of particles is modelled by a stochastic discrete particle approach, in which the turbulent dispersion of droplets is predicted by integrating the trajectory equations for individual droplets, using the instantaneous fluid velocity along the droplet path during the integration. When sufficient representative droplets are tracked, the random effect of turbulence on the droplet dispersion may be considered for. To do this, the discrete random walk model is employed. In this model, the fluctuating velocity components are discrete piecewise constant functions of time and their random value remains unchanged over a time interval given by the characteristic lifetime of eddies.

The droplets are allowed to collide and breakup during their motion. The droplet collision is modelled using O’Rourke model [19]. In this algorithm, it is assumed that two droplets may collide only if they are in the same continuous phase cell and only coalescence and bouncing outcomes are considered. The probability of each outcome is calculated from the collisional Weber number and experimental data and the properties of the two colliding
parcels are modified based on the outcome of the collision. The droplet breakup is modelled by the Taylor analogy breakup (TAB) model proposed by O’Rourke and Amsden [20]. In this approach, the droplet distortion is assumed to be caused by a forced, damped harmonic oscillator, in which the forcing term is given by the aerodynamic drag, the damping term is a result of the liquid viscosity, and the restoring force is from the surface tension.

### 2.3 Boundary Conditions

An air-blast atomizer with a spray angle of 55° is used. The water spray and the compressed air temperatures at the nozzle exit are set at 23 °C. The results of three typical water spray flow rates will be described:

a) the spray water mass flow rate $m_w=0.2523 \text{ kg/s}$, the corresponding air volume flow rate $m_a=131 \text{ Nm}^3/\text{h}$, the droplet diameter is between 10 µm and 140 µm with a Sauter mean diameter of 44 µm and the relative velocity is 86.5 m/s.

b) $m_w=0.504 \text{ kg/s}$, $m_a=94 \text{ Nm}^3/\text{h}$, $d=10$–$167 \mu m$ with a

![Figure 2](image.png)

Figure 2  History of droplet trace colored by droplet diameter
Sauter mean diameter of 56 µm and the relative velocity is 56.4 m/s.
c) \(m_{\text{w}}=1.0095 \text{ kg/s}, m_{\text{w}}=41 \text{ Nm}^2/\text{h}, d=15-210 \mu \text{m} \) with a Sauter mean diameter of 64 µm and the relative velocity is 12.91 m/s.

At the inlet, the off-gas volume flow rate is 68000 Nm³/h and its temperature is 800 °C. The turbulent kinetic energy, \(k\), and its dissipation rate, \(\varepsilon\), at the inlet are given as \(k_{\text{in}}=0.01V_{\text{in}}^2\), and \(\varepsilon_{\text{in}}=2.0k_{\text{in}}^{3/2}/\text{D}\), respectively. Here \(V_{\text{in}}\) is the inlet velocity and \(\text{D}\) is the duct diameter. At the outlet, the pressure is specified to be atmospheric. At the duct wall, a no slip condition is set for the gas flow and an adiabatic wall, for thermal conditions. When the droplets are in contact with the wall, it is assumed that they are reflected or evaporated.

2.4 Numerical Method

The commercial code Fluent 6.1.22 [13] is used. All the space derivatives are discretized using the QUICK scheme [21], and the pressure-velocity coupling is solved using the SIMPLEx algorithm [22]. A refined mesh, with about 300,000 tetrahedral/hybrid Tgrid type elements, is used. First, the steady state gas flow is simulated. Then droplet injection from the atomizing nozzle is initiated and unsteady state simulation of the interactions between the gas and the droplets are carried out. The time step used is between \(10^{-5}\) and \(5 \times 10^{-4}\), and 25 iterations within each time step are set, with which the scaled residuals of smaller than \(10^{-3}\) for all variables can be reached.

The spray is modelled by 56 droplet streams. Every stream represents a number of droplets. This number of droplet streams has been proved to give accurate enough solutions from trial calculations in a range of 15-120 droplet streams.

3. MEAN OUTLET TEMPERATURE

In designing the spray cooling system of gas, it may be usually desirable to obtain a certain gas temperature at the exit. The gas temperature at the outlet is usually non-uniform, depending on the mixing between the gas and the vapour. However, the mean outlet gas temperature should be an important parameter. In order to estimate the mean gas temperature at the outlet, the following assumptions are made:

1) neglect the heat loss from the duct wall,
2) assume that all the droplets vaporize completely before they flow out of the duct exit,
3) assume constant physical properties.

Then the decrease of the internal energy of the gas should be balanced by the latent heat for evaporation of all droplets and the increase of the internal energy of air injected from the nozzle, water droplets and vapour. For one nozzle, the mean outlet gas temperature can be calculated by

\[
T_{\text{out}} = \frac{m_{\text{g}}C_{p,g}T_{\text{g,in}} + m_{\text{w}}C_{p,w}T_{\text{w,in}} - m_{\text{s}}(h_{\text{g}} + 373(C_{p,g} - C_{p,v}) - C_{p,v}T_{\text{w,in}})}{m_{\text{g}}C_{p,g} + m_{\text{w}}C_{p,w} + m_{\text{s}}C_{p,v}}
\]  

Here \(C_p\) is the specific heat, \(h_{\text{g}}\) the latent heat, \(m_{\text{s}}\) the mass flow rate of the off-gas at the inlet, \(m_{\text{w}}\) the water flow rate of the nozzle, \(m_{\text{a}}\) the air mass flow rate of the nozzle, \(T\) the temperature. The subscript, a, g, w, and v denotes air, gas, water, and vapour, respectively. The subscript in is at the inlet. The specific heat \(C_{p,g}\) is a value taken at the mean gas temperature between inlet and outlet. \(C_{p,v}\) is the vapour specific heat at the mean temperature between 100 °C and the outlet temperature. \(C_{p,a}\) is the air specific heat at the mean temperature between the nozzle inlet and the outlet. It should be pointed out that Eq. (9) can be used to decide the nozzle capacity, \(m_{\text{a}}\), once a certain outlet temperature is given for the design purpose of a spray cooling system, then

\[
m_{\text{a}} = \frac{m_{\text{g}}C_{p,g}(T_{\text{g,in}} - T_{\text{out}}) - m_{\text{w}}C_{p,w}(T_{\text{w,in}} - T_{\text{out}})}{h_{\text{g}} + C_{p,v}(373 - T_{\text{out}}) + C_{p,a}(T_{\text{g,in}} - 373)}
\]  

In industrial spray cooling system, it may be necessary to arrange a series of spray nozzles to cool the off-gas to a certain exit temperature. For the arrangement of \(N\) nozzles, the mean outlet gas temperature may be estimated as follows:

\[
T_{\text{out}} = \frac{m_{\text{g}}C_{p,g}T_{\text{g,in}} + \sum_{i=1}^{N} m_{\text{g}}C_{p,g}T_{\text{g,in}} - \sum_{i=1}^{N} m_{\text{s}}(h_{\text{g}} + 373(C_{p,g} - C_{p,v}) - C_{p,v}T_{\text{w,in}})}{m_{\text{g}}C_{p,g} + \sum_{i=1}^{N} m_{\text{g}}C_{p,g} + \sum_{i=1}^{N} m_{\text{s}}C_{p,v}}
\]  

Because there is no experimental data available, the simplified analytical solution for the outlet temperature from Eq. 9 is compared with that of numerical one to verify the CFD predictions.

4. RESULTS AND DISCUSSION

In order to obtain the transient behaviour of the spray cooling of the hot gas in a duct, un-steady state simulation has been carried out. Figure 2 shows the droplet traces coloured by the droplet diameter for stream one of the 56 streams at various transient times for the case \(m_{\text{a}}=0.2523 \text{ kg/s}\). Initially the droplet size distribution ranges from 10 to 145 µm in diameter. Once the droplets enter the main flow, they coalesce obviously because some droplets can be 400-500 µm in diameter. For a droplet evaporating in stagnant surroundings, the evaporation time may be estimated by \(t=\frac{\rho_{\text{d}}\pi d^2}{(\text{kg/s})(\text{Tave})}\). Here \(d_{\text{d}}\) is the droplet diameter, \(\text{kg/s}\), the thermal conductivity of gas, and \(\text{Tave}\), the logarithmic mean

![Figure 3 Velocity vector plot in the plane across the nozzle for \(m_{\text{a}}=0.2523 \text{ kg/s}\)](image)
temperature difference. It is seen that the evaporation time is proportional to the square of the droplet diameter. Therefore, the large droplets due to coalescence may escape out of the duct outlet without being evaporated completely for a certain length of the duct if the residence time of the droplets is not long enough. This affects negatively the cooling efficiency and other practical aspects. Such coalescence should be minimized in the design of spray cooling systems. For example, when a number of nozzles are installed in a spray cooling system, the configuration of the nozzles should be such that the spray front from each nozzle should not be in touch with each other. It can be seen from Fig. 2 that the spray front against the main flow is depressed because of the effect of oncoming gas flow, and the spray front moves towards the duct exit. The spray can penetrate deeply into the main flow and some un-evaporated droplets during their motion may reach onto the opposite wall of the duct, as can be seen in following section. The droplet front approaches the outlet within about 0.4 s, which is equivalent to the convection time. Later on, the flow and the droplet trace become quite similar at different transients and a quasi-steady state is reached.

The velocity vector in the plane across the injection nozzle near the nozzle exit is depicted in Figure 3. It is seen that the main gas flow is affected by the spray and the droplets are dispersed or evaporated in the main flow. As expected, the gas flow drives the spray towards the exit and the conical shape of the outside surface is distorted. The simulations reveal that the flow fluctuates with time even when it reaches a quasi-steady state. The slight velocity and temperature fluctuation results from different evaporation times for different sizes of droplets.

The temperature contours at various transients are shown in Figure 4 for different planes. Before the spray front reaches the exit, a lower temperature zone appears (for example, at t=0.2 and 0.4 s). This is because it takes some time for droplets to evaporate (the evaporating time is different for different droplet sizes) just after the droplets are injected into the main flow, and a swarm of droplets which enters at first are then probably evaporated at quite similar time interval and in the same zone. It is seen that the spray penetration becomes deep toward the opposite wall of the duct as the spray moves forward. It is also seen that after about t=0.4 s the temperature distributions are quite similar (see Figure 4c-d), indicating that a quasi-steady state is reached. The temperature is not uniform in a cross section and the lower temperature zone enlarges gradually as the length of the duct increases. The nonuniform temperature distribution at the exit indicates that a perfect mixing between the gas flow and the spray is not achieved just by one injection nozzle.

Figure 5 depicts the mass fractions of H2O in various planes and cross-sections at different times for m_w=0.2523 kg/s. At t<0.4 s, a zone with relatively high mass fraction of H2O appears, explaining a lower temperature zone as shown
in Figure 4 at t=0.2 and 0.4 s. At t>0.4 s, the distribution of the mass fraction of H\textsubscript{2}O becomes similar with time, and the high mass fraction zone is in the bulk region. Higher mass fraction of H\textsubscript{2}O leads to lower gas temperature, as seen in Figure 4.

The contours of the mass source of H\textsubscript{2}O on the duct wall at t=4 s (in quasi-steady state) are shown in Figure 6 for three nozzle capacities. At m\textsubscript{w}=0.2523 kg/s, only a small number of droplets will hit the opposite duct wall. When the nozzle capacity increases, the DPM source of H\textsubscript{2}O becomes higher, indicating that more droplets hit the duct wall. The starting point for the droplet hitting the wall is not just the opposite of the nozzle, but a distance downstream (about 0.6 m from the injection point), because of the influence of the main gas flow. With increasing the nozzle capacity, the droplet-hit area becomes larger and the length is longer, as can be seen from Figure 6. At m\textsubscript{w}=1.01 kg/s, too many droplets will hit the duct wall, which may cause undesirable corrosion problems. It is seen that the hitting area of the droplets is on the lower (opposite the nozzle) part of the duct wall and there are no droplets hitting the upper part of the duct wall.

Figure 7 shows the time histories of the mass-averaged gas temperature in the flow domain, T\textsubscript{g}, and the mean gas temperature at the outlet, T\textsubscript{gout}, for two nozzle water capacities m\textsubscript{w}=0.2523 kg/s and m\textsubscript{w}=1.01 kg/s (the subscript cf denotes cross flow with nozzle located at the duct wall). It can be seen that the mean temperature T\textsubscript{g} decreases rapidly at the initial stage during the transient. At 0.4 s<t<1.4 s, the mean gas temperature recovers slightly. At t>1.4 s, T\textsubscript{g} remains more or less unchanged. The history of the mean gas temperature at the outlet follows a similar trend, except that at t<0.25 s, the outlet gas temperature remains unchanged. This is because the front of the gas flow cooled by the water spray takes a certain time to approach the exit. A slight fluctuation of the mean temperature with time at the exit occurs. This may be due to the fact that the number of droplets or the mass of liquid water escaping out of the exit without being completely evaporated may change with time. At large nozzle capacity m\textsubscript{w}=1.01 kg/s, it appears that the front of the spray reaches the exit within a slightly shorter time than at small capacity m\textsubscript{w}=0.2523 kg/s. This is because the flow velocity becomes greater at higher nozzle capacity. As expected, the mean gas temperature is lower at larger nozzle capacity due to more heat provided by the hot gas to evaporate the injected droplets and heat up the vapour.

For the spray cooling case considered there is no experimental data available. Verification of the numerical predictions is made by comparing to the simplified analytical solution proposed in the above section. Comparison of the numerical prediction with the analytical solution from Eq. 10 is shown in Figure 8 as a function of the nozzle capacity. It can be seen that the difference is rather small. This explains two aspects: first, the numerical prediction is in a good agreement with the analytical solution, indicating the CFD model can be well used to predict the spray cooling
processes. Secondly, the predicted mean temperature is slightly higher than the analytical one, indicating that a small amount of liquid water have probably escaped out of the exit due to un-evaporated large droplets resulting from coalescence, for the duct length considered. Practically it is desirable to have all droplets completely vaporized before they reach next equipment. From the present results it is recommended that the system installed downstream should be located about 9-10 m away from the nozzle location. It is also seen that the mean exit temperature decreases with increasing the nozzle capacity. However, it should be noted that the droplets hitting the duct wall also increase as the nozzle capacity increases, which may not be desirable in practical operations. It should be mentioned that the cross flow cooling by locating the nozzle at the duct wall may be preferable to the parallel flow cooling by locating the nozzle at the central axis [23].

5 CONCLUSIONS

The flow and heat transfer in a spray cooling duct with nozzle located at the wall has been numerically simulated and transient behaviour of the droplet trace, velocity and temperature has been obtained for different nozzle capacities. A simplified analytical solution for the mean exit temperature is proposed for the spray cooling. Predicted mean temperature at the outlet is verified with the simplified analytical solution.

A quasi-steady state is reached at about t=0.4 s. In the quasi-steady state, a small fluctuation in velocity and temperature remains, caused by different evaporation times of different droplet sizes.

With increasing nozzle capacity, the outlet temperature decreases. At larger nozzle capacity, droplets are more probably subject to coalesce, more droplets may hit the duct wall and more un-evaporated droplets may flow out of the outlet for the duct length considered. The system installed downstream should be located about 10 m away from the nozzle location.

The suggested reasonable nozzle capacity is about 0.25 kg/s. With nozzle located at the duct wall, only opposite part of the duct wall may be hit by the droplets.

ACKNOWLEDGEMENT

This project is sponsored by Outokumpu Technology, Research Center, Outokumpu Research Oy. The authors would like to thank Dr. A. Vartiainen, Dr. P. Taskinen and Mr. T. Ahokainen for their interest in this work and their valuable advices and Mr. T. Kankaanpää for his help. The Finnish IT center for Science (CSC) provided the computer sources.
REFERENCES


Figure 8 Comparison of the predicted and analytical mean gas temperatures at the outlet.