Modelling fire suppression by water sprays in CFD: a review of progresses and challenges

Dr Siaka DEMBELE

Associate Professor / Leader Fire, Explosion & Fluid Dynamics Research Group

Kingston University London, UK

Outline

- Background
- Aims/Objectives
- Spray atomisation modelling (progress & challenges)
- Water spray and fire interaction modelling: some remaining challenges
- Conclusions & Future research needs for the water mist industry

Background (1)

- In past two decades, major advances in water mist/spray technology for fire suppression by the watermist industry

- Many application and experimental studies on water mist/spray have been carried out

- Theoretical studies based on numerical and computational modelling for fire suppression by water mist/sprays are very limited

Background (2)

- Computational Fluid Dynamics (CFD) well established for fire growth, spread and smoke simulations (e.g. FDS, FireFoam). Solves conservation equations for mass, momentum and energy (Navier-Stokes equations).
- Major advances in spray modelling have been achieved in the context of automotive applications (diesel injectors). With diesel cars being outlawed in many countries, negative impact on spray research expected
- Progress on the development of CFD tools to simulate fire suppression by water mist/sprays very slow
- Water mist relatively new technology - the development of robust CFD tools for spray formation and fire suppression by watermist positive step for performance based design and mist industry. Can also help demonstrate that water “mist” criterion met without testing.

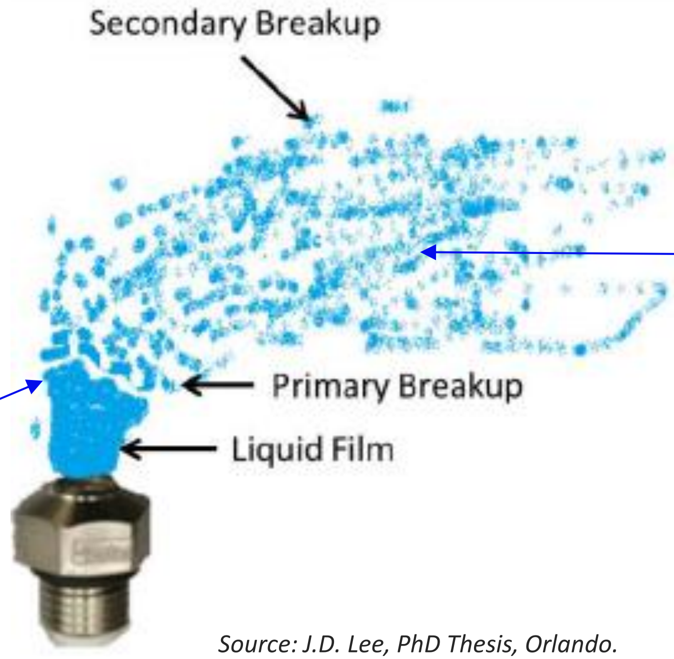
Aims/Objectives

To provide an overview of the research advances and needs for CFD fire suppression by sprays.

Scope:

- Spray formation/atomisation
- Spray interaction with fire during suppression

Spray atomisation modelling in CFD (1)



Dilute zone (Lagrangian approach for discrete droplets & Eulerian approach for continuous gas phase)

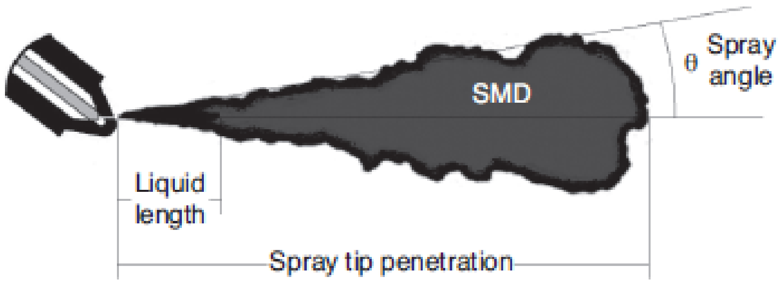
Dense spray zone near nozzle exit

Source: J.D. Lee, PhD Thesis, Orlando.

- Purpose of all spray atomizer nozzles types (pressure, impingement, twin fluid etc..) is to transform a continuous liquid phase to dispersed/discrete droplets
- Major advances in modelling spray primary and secondary break-ups achieved for automotive applications (fuel injectors in compression ignition engines e.g diesel)

Spray atomisation modelling in CFD (2)

0D models (empirical & semi-empirical correlations)



Model	Correlation	References
Reitz and Bracco	$\tan(\theta) = \frac{4\pi}{A}\left(\frac{\rho_g}{\rho_l}\right)^{0.5} f(\gamma)$	Reitz and Bracco, 1979
Reitz and Bracco Simplified	$\tan(\theta) = \frac{4\pi}{A}\left(\frac{\rho_g}{\rho_l}\right)^{0.5} \frac{\sqrt{3}}{6}$	Heywood, 1988

SMD = Sauter Mean Diameter D32

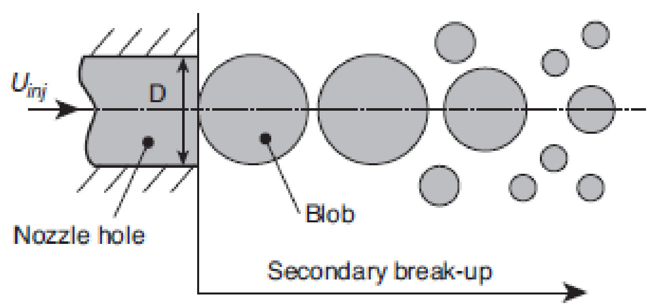
Source: Dos Santos, Le Moyne, IFP Energies Nouvelles 2011.

- ❖ Based on experimental test data – extensively used in the automotive industry for diesel engine models (advantage reduced computational cost).
- ❖ 0D models can be applied to water mist but requires extensive experimental data for calibration and reformulations of aerodynamic break-up models
- ❖ 0D models too rigid, interaction spray and gas phase not modelled, no spatial dependence accounted for, needs calibration and mostly too crude
- ❖ Multidimensional CFD approach more reliable, robust and flexible

Spray atomisation modelling in CFD (3)

Primary Atomisation (Blob Model)

- Goal of primary break-up model is to determine the initial conditions of droplets (size, velocity etc…) at nozzle exit. These conditions are influenced by the flow conditions **inside and outside** the nozzle. Measurements in dense spray zone near nozzle exit difficult task.
- Blob model most popular – Reitz & Diwakar (1987) (e.g. KIVA CFD automotive code)



No detailed modelling of primary atomisation. Spherical droplets of same diameter as nozzle injected, which are subjected to secondary breakup (Lagrangian).
Drop size and velocity predicted.

- Simple model, but physics of primary atomisation (e.g. effects of nozzle geometry) not included – major limitation. Suitable for single fluid pressure nozzles.
- Blob model good starting point for implementation in CFD codes for water mist atomisation

Spray atomisation modelling in CFD (4)

Primary Atomisation (Distribution function - DF)

- ❖ This approach assumes that the water jet is already atomised at the nozzle exit, the distribution of droplet sizes is predicted by a mathematical distribution function.

- ❖ DF approach has so far been the most popular used in fire CFD codes (FDS and FireFoam) and employed for suppression by water sprays in FDS, a combination of log-normal and Rosin-Rammler distribution used

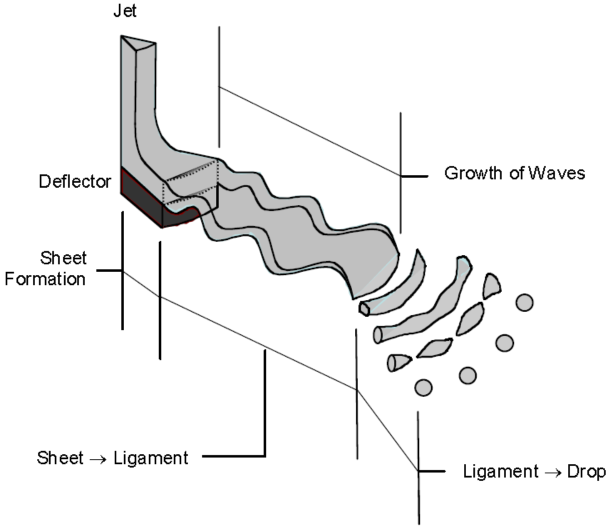
$$F_v(D) = \begin{cases} \frac{1}{\sqrt{2\pi}} \int_0^D \frac{1}{\sigma D'} \exp\left(-\frac{[\ln(D'/D_{v,0.5})]^2}{2\sigma^2}\right) \, \mathrm{d}D' & (D \le D_{v,0.5}) \\ 1 - \exp\left(-0.693\left(\frac{D}{D_{v,0.5}}\right)^{\gamma}\right) & (D > D_{v,0.5}) \end{cases}$$

Main limitations distribution functions: empirical parameters (σ, γ and Dv0.5) are needed, obtained usually from experiments

Spray atomisation modelling in CFD (5)

Primary Atomisation (Kelvin-Helmholtz instability break-up – KH model)

Ref. Zhou & Yu, Fire S J, 46 (2011)



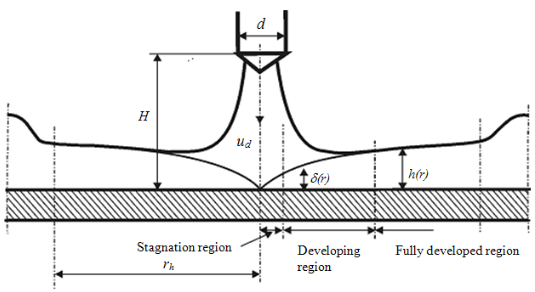
- ❖ Liquid breakup due to aerodynamic instabilities growing on the surface of the jet stream/liquid sheet
- ❖ KH model extensively used in automotive industry (Reitz, Huh & Gosman)
- ❖ For low-pressure water mist / sprinkler impingement type nozzles the University of Maryland and Kingston University have extensively worked on the development of KH models

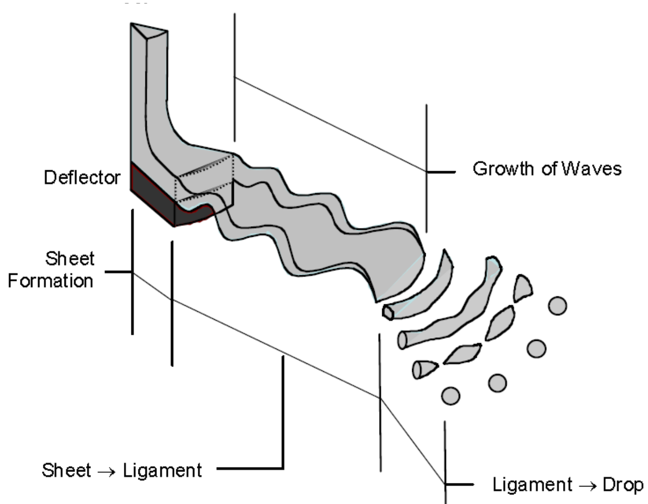
Spray atomisation modelling in CFD (6)

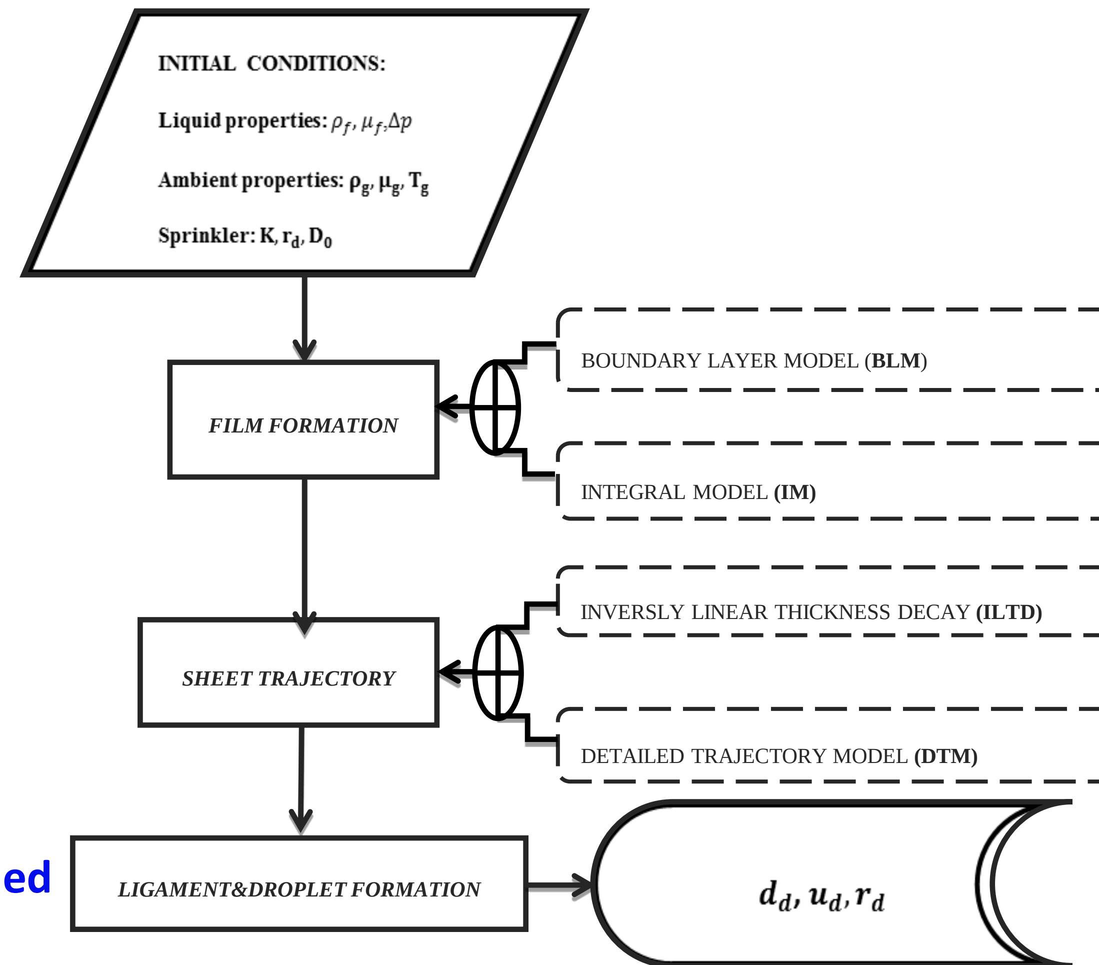
Primary Atomisation (Kelvin-Helmholtz instability break-up – KH model)

Model developed at Kingston University for water spray

Aghajani, Dembele and Wen (2014) *Analysis of a semi-empirical sprinkler spray model.* *Fire Safety Journal, 64, pp. 1-11.*







Relatively good predictions obtained for simple geometry flat plates.

Spray atomisation modelling in CFD (7)

Primary Atomisation (Kelvin-Helmholtz instability break-up – KH model)

- KH approaches is well suited for low pressure water mist/sprinkler type initial spray predictions

- At the current state of the research, it is possible to predict the initial spray characteristics for SIMPLE GEOMETRY NOZZLE (i.e. water jet impinging a circular plate without tines/holes)

- Future research needs: include the effects of boss, arm, holes and tines in spray model

Spray atomisation modelling in CFD (8)

Primary Atomisation (other advanced approaches)

❖DNS (Direct Numerical Simulation) of sprays is impractical due to high computing costs involved

❖ Eulerian RANS (Reynolds-Averaged Navier-Stokes) spray primary atomization model have been developed by Vallet *et al*. (2001) from generalizing Kolmogorov hypothesis on turbulence to characteristic scales of spray.

❖ Eulerian LES (Large Eddy Simulation) based on VOF (Volume of Fluid) developed by Villiers et al. (2004)

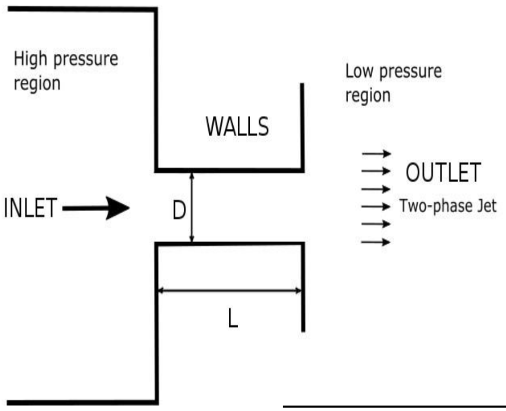
Current research trend: LES with Eulerian frame (e.g. VOF) to accurately model primary atomisation in dense spray zone & Couple with Lagrangian approach in dilute zone of spray to track droplets.

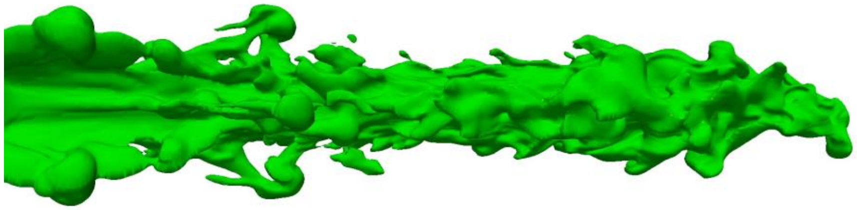
e.g. ELSA (Eulerian-Lagrangian Spray Atomization) approach

Spray atomisation modelling in CFD (9)

Research at Kingston University

ELSA approach has been extended to LES frame to predict initial water spray for primary atomisation in flash atomisation – promising results for simple geometry





Snapshot of iso-contour Y_l for LES simulations at the first 5mm of spray motion downstream the nozzle exit. A mean cell size equal to $4\mu m$ was used close to the nozzle exit.

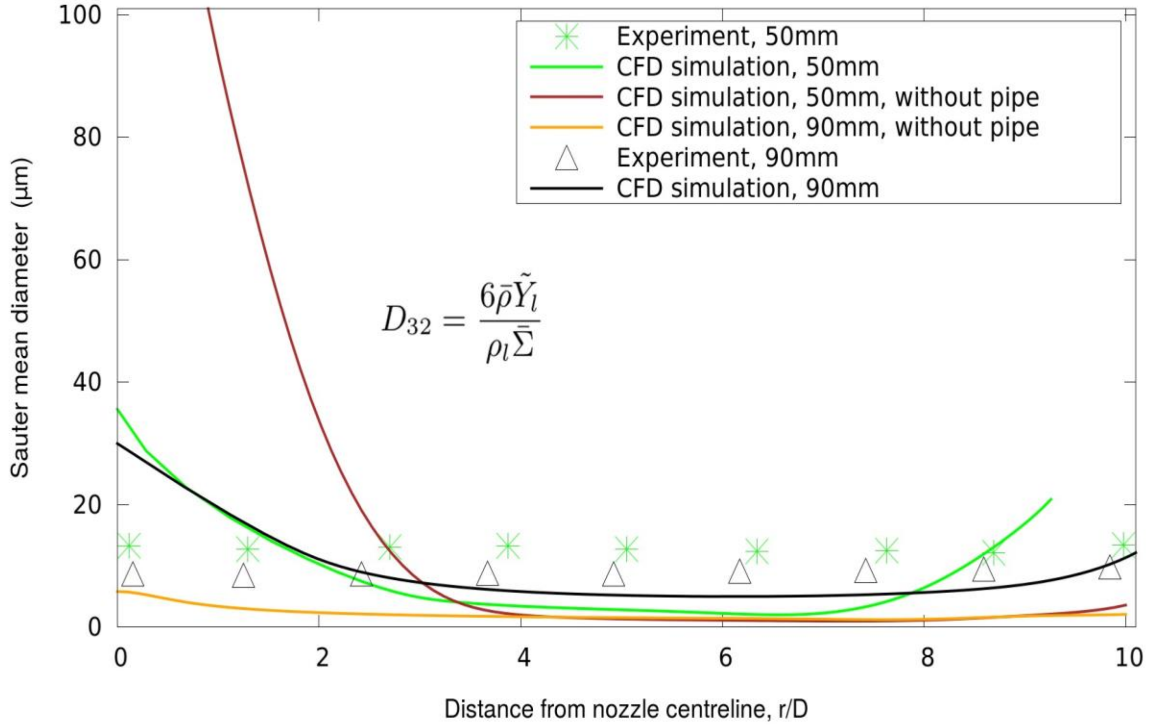
Table1:Physical properties for simulations.

	Physical parameters for simulations
Inlet pressure	700 kPa
Inlet temperature	247 K
Outlet pressure	100 kPa
Outlet temperature	298 K
L/D	78.4
Nozzle diameter	0.81 mm
Thermodynamic conditions	Saturated

Spray atomisation modelling in CFD (10)

Research at Kingston University

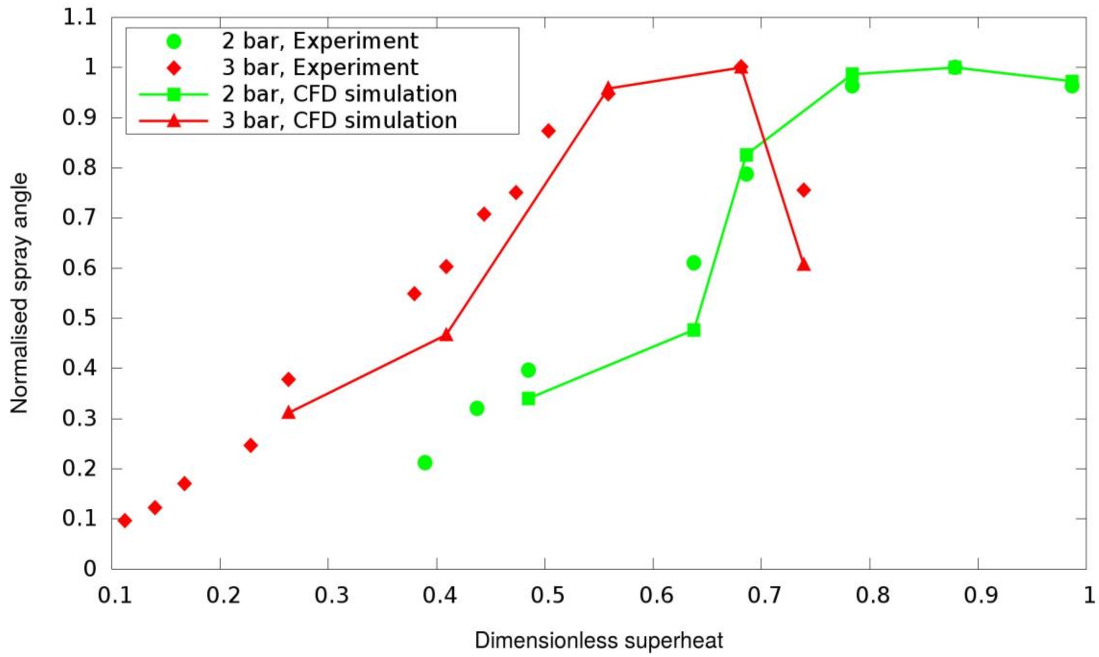
Lyras, K., Dembele, S., Schmidt, D., Wen, J., 2018. Numerical simulation of subcooled and superheated jets under thermodynamic non-equilibrium. International Journal of Multiphase Flow 102, 16–28.



Sauter mean diameter (SMD) at x=50, 90mm. Comparison with Zhou et al. (2012).

Spray atomisation modelling in CFD (11)

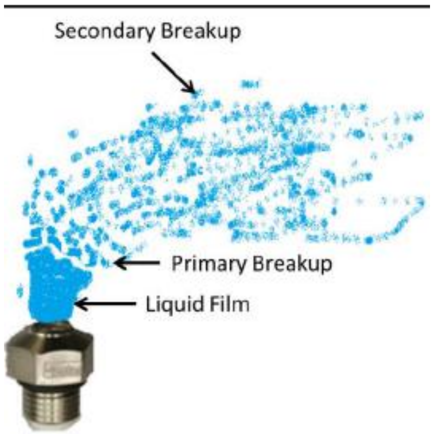
Research at Kingston University



Normalised spray angle with respect to the dimensionless superheat $\Delta\ \check{T}_{sh}$ at 50mm (x/D=33.3) distance downstream the nozzle exit. Comparison with Park and Lee (1994).

Spray atomisation modelling in CFD (12)

Secondary Atomisation



- Modelling secondary break-up less problematic than primary breakup
- Well established models that provide relatively good predictions have been developed in literature
- Taylor-Analogy Break-Up Model (TAB Model) most popular and used in many codes.

Water spray and fire interaction modelling: challenges (1)

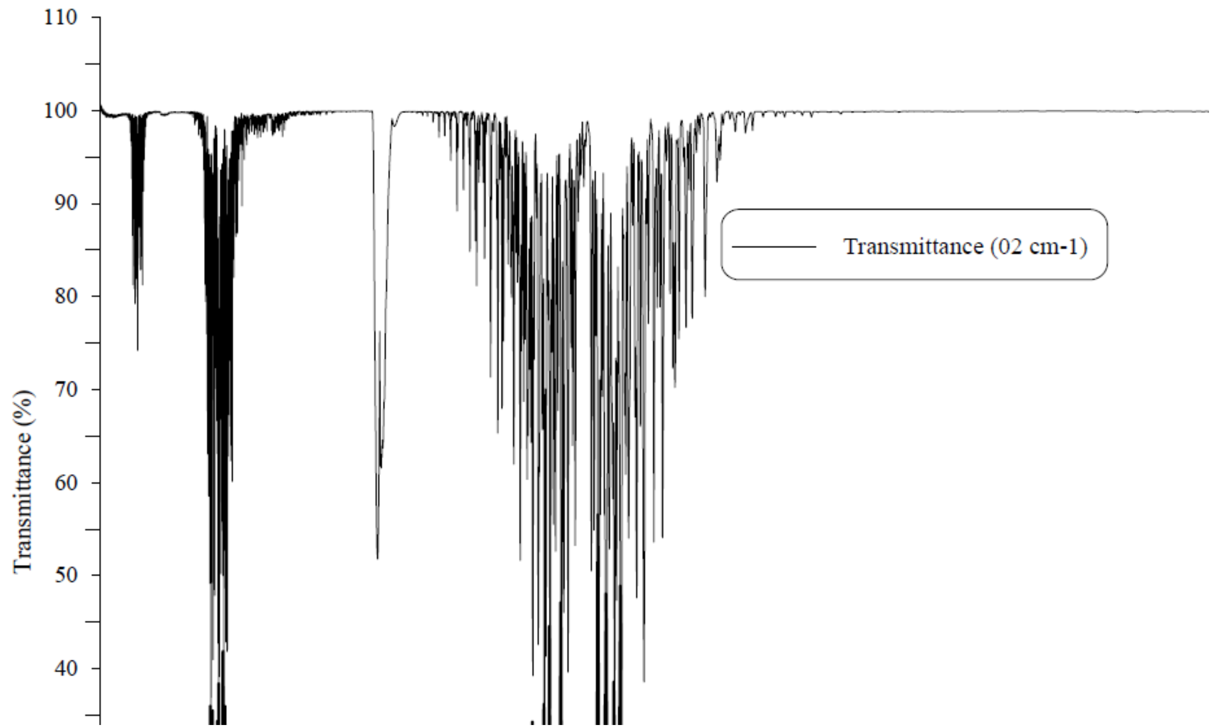
- Thermal radiation remains the dominant mode of heat transfer in a high temperature fire environment

- Once a water mist spray interacts with fire, steam (water vapour) resulting from droplets vaporisation is produced.

- Modelling the interaction between thermal radiation from the flame and water vapour (gas phase) with acceptable computing times remains one the main challenges to be addressed

Water spray and fire interaction modelling: challenges (2)

Spectral transmissivity of a water spray (droplets and water vapour)



Source: Dembele, S., Delmas, A. and Sacadura, JF (1997) A method for modeling the mitigation of hazardous fire thermal radiation by water spray curtains. Journal Of Heat Transfer, 119(4), pp. 746-753.

- Emission/absorption of H_2O in specific bands (not continuous like liquid water) Significant bands centred around 6.3 microns, 2.7 and 1.9 microns.
- Large number of vibration-rotation transition lines in H_2O spectrum (thousands lines):

LINE-BY-LINE SPECTRAL GAS MODEL IMPRACTICAL DUE TO LARGE COMPUTING TIME

Water spray and fire interaction modelling: challenges (3)

Modelling radiative heat transfer in water sprays

$$\mu \frac{dI_\lambda(x,\mu)}{\beta_{d\lambda} dx} + I_\lambda(x,\mu) = (1-\omega_{d\lambda}) I_{b\lambda}[T_{spray}] + \frac{\omega_\lambda}{2}\int_{-1}^{1} \varphi_{d\lambda}(\mu,\mu') I_\lambda(x,\mu') d\mu' = S_\lambda(x,\mu)$$

RADIATIVE TRANSFER EQUATION (RTE)

- Thermal radiation attenuation by water droplets due to absorption and scattering (Mie Theory)
- Major developments in gas radiation models in recent years: Narrow Bands, Full-Spectrum Correlated-K (FSCK), SLW (spectral line weighted sum of gray gases etc…
- **Good alternative to handle gas radiation in water mist fire suppression is the Box Model based on the Wide band approach. Recently been implemented in FireFoam (FM Global, Warwick & Kingston Universities)**
- **Efficient RTE solvers needed - P1 as alternative to Finite Volume Method (FVM). Currently investigated in EU H2020 RAD-FIRE project led by S. Dembele**

Conclusions & Future Research Needs for the Water Mist Industry (1)

- Modelling spray atomization has made significant advances in past decades mainly driven by research in the automotive industry for diesel engines; CFD remaining future trend to better capture spray formation and dynamics
- Modelling primary atomisation in the dense zone of the spray near the nozzle exit remains a challenge.
- The majority of spray models (0D, 1D or multidimensional) have been calibrated for combustion engine fuels, new research and testing needed to rigorously adapt them to water mist sprays
- For watermist systems, primary atomisation of low pressure impingement nozzles can be well captured with Kelvin-Helmholtz instability break-up approach in simple configurations. New research needed in future to include effects of arm, tine, boss and cavitation/flashing inside the nozzle.

Conclusions & Future Research Needs for the Water Mist Industry (2)

- To keep compatibility and for future coupling with currently existing opensource CFD fire simulations tools such as FDS and FireFoam, future water mist spray models should be developed in LES (Large Eddy Simulation) turbulence framework.
- For medium/high pressure water mist nozzles, single or twin fluid future research should focus on LES with Eulerian frame (e.g. VOF) to accurately capture the spray primary atomisation taking into account the geometry and flow conditions inside and outside the nozzle.
- Good approach for future developments could be ELSA (Eulerian Lagrangian Spray Atomization) in LES frame approach where a Eulerian method is used in the dense phase near the nozzle and a Lagrangian one for droplets in the dilute zone
- Slowdown expected in automotive spray modelling (diesel ban) the water mist industry should develop own research agenda.

Dr Siaka Dembele
s.dembele@kingston.ac.uk
Associate Professor
Leader Fire Explosion Fluid Dynamics Research Group
Department of Mechanical Engineering
Kingston University London
Friars Avenue, London SW15 3DW, UK