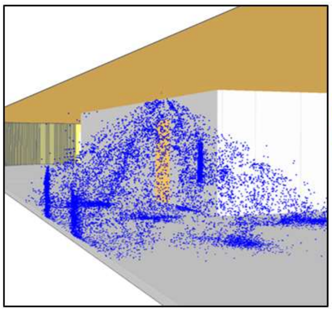
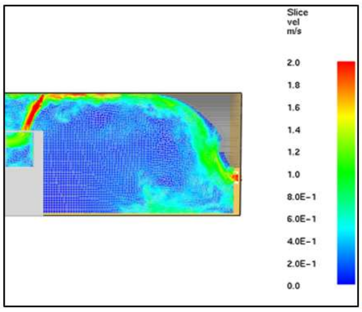
IWMA POSITION PAPER ON

CFD and water mist fire fighting





V.0.2. 21.08.2024

Imprint

Title:
Position Paper on CFD (Computational Fluid Dynamics) and Water Mist Fire Fighting

Version:
0.2 dated 21.8.2024

Published by:
International Water Mist Association (IWMA), Hamburg, Germany, 2024

Prepared by:
Max Lakkonen, Institute of Applied Fire Safety Research (IFAB)

Contributors:
Alex Palle, VID FireKill Aps
Ann Micheli, Ultra Fog
Bogdan Raciega, Baltic Fire Laboratory
Dirk Laibach, Johnson Controls
Jonathan Carpenter, FM Approvals
Kati Laakkonen, Marioff Corporation
Kemal Arsava, RISE Fire Research (Norway)
Luciano Nigro, JensenHughes
Markku Vuorisalo, Marioff
Ning Ren, FM Global
Rüdiger Kopp, Fogtec GmbH
Siaka Dembele, Kingston College

Language check by:
Ann Micheli, Ultra Fog

Table of content

1 Introduction

1.1 Background

Computational Fluid Dynamics (CFD) has gained widespread popularity across various fields, including fire engineering and water mist firefighting. The ability of CFD to simulate complex fluid flow phenomena has made it an invaluable tool for understanding fire dynamics and optimizing fire protection systems.

Alongside its growing use, many concerns have been raised about the misuse of CFD in water mist firefighting. For example, there have been cases where CFD has been used to solely to justify system designs without any experimental tests. This, along with other bad practice, has encouraged the International Water Mist Association (IWMA) to express concerns regarding the suitability, accuracy and reliability of CFD simulations in this context. If a single mist system fails, it can cast shadows on the whole water mist industry.

In response to these concerns, the IWMA has undertaken to publish this position paper. This paper aims to provide a brief overview of the current status of CFD modelling of water mist systems. By highlighting both the potential benefits and limitations of CFD in this context, the IWMA seeks to clarify the existing challenges and promote responsible use of CFD for water mist applications.

1.2 Purpose and scope

The objective of this document is to provide insight into the application of Computational Fluid Dynamics (CFD), with a specific focus on water mist systems. It outlines the advantages, limitations, and recommended practices associated with using CFD for modelling water mist systems. This paper does not serve as a technical guideline, standard, or code, but rather as an informative resource on the topic.

1.3 Disclaimer

The information presented in this document is derived from available research and expert opinions within IWMA membership. The authors and publishers of this document cannot be held responsible for any errors or omissions in the information provided.

2 Computational fluid dynamics (CFD) in fire engineering

2.1 Introduction

Computational Fluid Dynamics (CFD) is a powerful numerical modelling technique used to analyse the movement of gases and liquids in various environments, governed by the Navier-Stokes equations. These equations describe how substances flow within a given field, accounting for mass, momentum, energy (enthalpy), and chemical species. In fire dynamics, CFD resolves all these quantities.

While often unnoticed by the public, CFD significantly impacts daily life. For instance, weather forecasts rely on a combination of CFD and experimental data to predict atmospheric conditions. Additionally, CFD plays a crucial role in designing aerodynamic vehicles and aircraft. It is instrumental in developing renewable energy technologies like wind turbines and improving energy efficiency in buildings through HVAC system adjustments. Furthermore, CFD simulations contribute to understanding physiological processes and optimizing medical developments.

2.2 Use of computational fluid dynamics (CFD) in fire engineering

The most widely used application of CFD is in smoke control for fire engineering. CFD simulations allow engineers to study the movement of smoke within buildings and evaluate the effectiveness of smoke control systems with a good degree of accuracy. These simulations analyse temperature and visibility for occupants and may even incorporate evacuation modelling in conjunction with CFD. The heat release rate (HRR) is typically prescribed for these simulations and such simulations are often mandated by authorities for public spaces.

Temperature control is also crucial for structural fire resistance. Simulations can assess the effectiveness of fire protection systems, like high-pressure water mist, in controlling the temperature of structures and potentially reducing building costs.

In addition to practical applications, CFD aids research and development efforts in fire engineering by providing insights into fundamental fire dynamics phenomena and exploring innovative fire safety solutions. It is also instrumental in planning fire test programs, allowing for the simulation of basic phenomena and the development of fire test protocols.

Furthermore, CFD serves as a forensic analysis tool and has been employed in analysing various fire incidents and disasters.

2.3 Benefits of CFD fire modelling

Benefits of CFD analysis are following:

- Saving time and cost: Conducting physical experiments to study fire behaviour can be costly and time-consuming. CFD offers a cost-effective alternative by allowing engineers to explore a wide range of scenarios quickly and relatively efficiently, without the need for large-scale testing facilities. Sometimes experimental testing is not even possible as the building might be complicated. Note! CFD is easier and more reliable for modelling simple fire & smoke scenarios without any suppression systems.
- Fire propagation is an inherently highly non-linear process. In fire tests, small changes in initial or environmental conditions can lead to significant differences in the result. CFD allows for constant conditions for each simulation case eliminating erroneous conclusions due to unintended test variation.
- CFD provides reliable data for the temperature distribution and smoke behaviour. Some systems like smoke extraction systems can be well adjusted for every project.

2.4 Challenges in CFD fire modelling

One of the most challenging aspects of CFD lies in accurately modelling the heat release rate of various fuel sources. Visible flames predominantly result from the burning of gaseous fuel, where the release rate is accurately known only if the fuel is readily in gaseous form. However, with liquid or solid burning materials, the actual gaseous fuel release rate depends on the fire itself, posing a significant challenge for modelling. Despite existing test procedures, the vast array of combustible materials complicates matters further.

Even when the fuel is identified, chemical reactions occur in multiple steps involving hundreds of different reactions between instantaneous chemical species. However, due to limitations in computing power, CFD often models only one fuel species and one or two reactions. Therefore, for simulations, the choice of the combustion model used in any CFD code should be clearly understood by the user.

Moreover, accurately modelling flame sheets in typical applications is hindered by computing power limitations. While grid resolution in large geometries is on the centimetre scale, flame sheets are often less than a millimetre thick. This inability to properly model flame sheets presents challenges in radiation modelling: radiation depends on temperature difference and the highest temperatures are found within the flame sheet, meaning the radiative source from fire can only be estimated. Another complexity for radiation modelling in fires is due the gaseous species from the combustion products (e.g. water vapour, carbon dioxide, carbon monoxide etc.) that emit and absorb thermal radiation in specific spectral bands unlike for example soot particles that emit continuously.

When water or any other suppressant is introduced into the combustion equation, the complexity of modelling increases significantly. There is an ongoing debate regarding the accuracy of CFD in predicting fire suppression and extinguishment. For instance, when using water, a portion of the suppression process takes place on the fuel surface, while another part occurs in the gaseous space.

The precision of fire simulations is always constrained by computing power. To achieve realistic results, computations often require high-performance clusters and weeks of computing time. Balancing the trade-off between model complexity and computational efficiency is a key challenge in CFD.

Validating and verifying CFD models presents its own set of challenges, stemming from the lack of comprehensive experimental data and the complexity of real-world fire scenarios. Comparing simulation results with experimental measurements and empirical correlations is crucial but challenging due to uncertainties in boundary conditions, material properties, and modelling assumptions.

2.5 Typically used codes in fire protection

There are two prominent codes dedicated to fire dynamics: Fire Dynamics Simulator (FDS), developed by the National Institute of Standards and Technology (NIST) and others, and FireFOAM by FM Global. Both are widely utilized in the field, offering valuable tools for fire engineers and consultants.

FDS, initially released in 2000, is designed to handle low-speed, fire-driven flows using rectilinear computational grids. It serves as a robust solver tailored for practical applications in fire protection engineering. FDS is extensively employed by fire consultants worldwide due to its reliability and effectiveness in simulating complex fire scenarios. FDS is open-source code and has an extensive validation data basis that is continuously extended.

FireFOAM is built upon the open-source Computational Fluid Dynamics solver OpenFOAM. Launched in 2008, FireFOAM utilizes non-structural grids, offering flexibility in modelling various geometries and scenarios. However, this flexibility comes with a trade-off, as FireFOAM typically requires more computing power compared to FDS due to its different solution method. This can be improved using extensive parallel computing for large scale geometries. OpenFOAM solver offers a multitude of modelling choices depending on the user capability.

In addition to these dedicated fire dynamics solvers, there are also commercial codes commonly used for smoke modelling and fire simulation. These codes provide further options for fire engineers and researchers to analyse fire behaviour, evacuation strategies, and smoke propagation in buildings and other structures.

3 Using CFD with water mist systems

3.1 Modelling water mist systems in CFD

Understanding the intricacies of modelling water mist systems in CFD requires several considerations. It typically involves input parameters such as droplet mean size, size distribution parameter, spray opening angle, and initial velocity, all derived from experimental data, because it is currently impossible to fully model the initial atomization process of droplet formation. These values, crucial for accurate simulation and performance of water mist, vary depending on the specific water mist nozzles used, which are often determined by manufacturers, based on application requirements and fire testing. Additionally, heat activation of water mist sprinklers is simulated similarly to other heat-activated devices, involving estimating bulb temperature through a differential equation model. The model requires experimental values for parameters RTI and C, that describe limitations of heat transfer from gas to bulb and heat conduction from bulb to sprinkler body, respectively. These values are tested in FM Approval process and usually can be given by automatic nozzle manufacturer.

The impact of water mist on fire combines various effects, including flame and gas cooling, surface cooling, radiation attenuation, and oxygen depletion. These effects are incorporated into simulations through models that account for water heating and evaporation on droplet surfaces, as well as the formation of vapour and the energy consumed in the process.

3.2 Challenges of CFD with water mist systems

3.2.1 Modelling

Water mist droplets are typically smaller than 300 microns with a volume average size of 100 microns. Due to the huge number of droplets, only a fraction of them can be actually computed. The rest are accounted for by scaling the effects to represent the correct amount of water. When a sufficient number of droplets are modelled, the results become insensitive to the introduced droplet number, minimizing inaccuracy. Modelling droplets requires high computer memory requirements and extended simulation times to resolve Lagrangian particle equations of discrete elements in a continuous phase. Water mist simulations are thus much slower to run than free-burn simulations. To achieve optimum accuracy and calculation time, the user should understand and study the model's sensitivity.

The accuracy of spray prediction depends on gas flow prediction, which is heavily influenced by turbulence modelling and grid resolution. Although most commonly used solvers typically employ Large Eddy Simulation (LES) turbulence modelling to mitigate the challenges of fully time-averaged turbulence modelling, they are still susceptible to grid resolution issues. The spatial scales in CFD applications are typically too large for the mesh resolution required for LES.

Moreover, there are indications that the cone spray model may not accurately predict radial droplet size development in small-scale simulations, potentially leading to inaccuracies in large-scale simulations. Additionally, the interaction between droplets after leaving the nozzle orifice cannot currently be modelled. Modelling the initial spray formation for a given nozzle remains one of the major challenges. .

3.2.2 Combustion

The most significant challenge in water mist modelling, as with any fire suppression modelling, arises from the lack of precise knowledge regarding the details of the combustion process, thus limiting the accurate modelling of the extinguishing effect of water. Consequently, only estimations can be made. Moreover understanding of gaseous combustion extinguishing remains limited. For these reasons, the development of robust extinction models is needed in fire modelling.

A commonly used approach is to utilise semi-empirical models, which may be highly conservative, assuming no suppression effect at all. Alternatively, combustion heat can be modelled based on data measured during full-scale fire tests. In certain scenarios, e.g., enclosure fires, modelling flame quenching may be less challenging if there are significant oxygen depletion effects.

3.2.3 Geometry

While certain CFD codes allow for accurate modelling of geometries, enabling representation of intricate details at the cost of larger grid sizes, other codes relying on structured grids may be constrained to depicting geometries with stair-step characteristics. Moreover, the process of grid refinement in some CFD software can be difficult, resulting in a significant loss of geometric details during simulations.

3.2.4 Validation

Validating fire modelling has proven to be challenging. The nature of fire as a phenomenon is highly nonlinear, to the extent that conducting the same fire test protocol with precisely identical known parameters can result in significant variations, such as in extinguishing time. Consequently, developing a reliable validation set needs repeating tests multiple times to identify average fire progression.

Another difficulty in validation lies in instrumentation. Frequently, essential quantities for CFD validation, such as the heat release rate of the fire, cannot be measured accurately or at all. These innate limitations of CFD, suggest they cannot replace experimental fire tests as validation for the proof of effectiveness.

If the models are validated against experimental tests, the CFD can be used as an additional validation tool in certain cases. The CFD modelling should only be done by organisations that know the experimental fire testing and performance of water mist systems.

3.2.5 Convergence

The convergence of governing equations in transient simulations is assessed by examining the residuals of key quantities. The introduction of additional gas phase from evaporated particles can amplify local fluctuations in flow properties, potentially leading to unexpected pressure variations within the computational domain. To mitigate numerical instability, it is advisable to increase the number of iterations of the pressure solver and carefully monitor the reduction in errors between iterations. The iteration count should be incrementally increased, considering the associated computational cost.

3.2.6 Cost and time

The simulation of water mist systems using CFD is a complex process, requiring significant investments in both time and financial resources. Moreover, multiple simulations are often necessary to achieve accurate results. In some cases, experimental validation data may need to be generated to ensure the reliability of the simulations.

3.2.7 User experience

Successfully simulating water mist systems demands not only access to reliable validation data but also experienced users. This often presents a significant challenge, as many organizations or individuals lack both validation data and hands-on experience with water mist systems. In some cases, they may not have witnessed full-scale fire tests to understand how water mist systems operate. Consequently, even conducting basic plausibility checks can be challenging for such users. There are situations where CFD users validate their models against small lab-scale fires and then apply them to large-scale fires. The users should aware that sometimes large-scale fires vary significantly from small-scale fires. The user should find available large-scale fire data and scenarios to understand how fire develops for better model validation.

4 Best practice

4.1 Applications

Water mist systems are increasingly utilised across various applications for their effectiveness in extinguishing fires while minimising water damage and harm to occupants. Complementary to this, CFD simulations offer valuable insights into water mist behaviour during different fire scenarios, aiding in system design and optimisation.

One significant application of CFD in conjunction with water mist systems is in enclosed spaces such as buildings, tunnels, and industrial facilities. By simulating the dispersion and interaction of water mist with fire-induced airflow, fire engineers can determine optimal nozzle placement within system approval specified tolerances. Additionally, CFD simulations help in understanding the cooling effects of water mist on surrounding surfaces and structures, crucial for mitigating thermal radiation and preventing fire spread. By accurately predicting temperature distributions, engineers can design water mist systems within approved tolerances, effectively protecting critical structures and minimising property damage.

Furthermore, CFD can serve as an assessment tool in cases where the real installation does not fully meet the specified requirements achieved from fire tests. Authorities may accept the use of CFD to demonstrate technical alterations aimed at improving system efficiency. Generally, CFD serves as an additional tool to enhance system design and implement various technical alterations, subject to approval by authorities and while considering the limitations of technology and related approvals.

4.2 Practical limitations

Despite its advantages, CFD modelling of water mist systems encounters several practical limitations as explained in chapter 3. The limitations can be technical, computational power or user related. All these can act as major limitations. Often the user experience is the most limiting factor and also very challenging as CFD can produce information that looks fine, but in reality, contains false results. Therefore, it is essential that only users experienced with CFD shall do the simulations. Technical limitations are dealing with the capabilities of CFD. These cannot be changed, but need to be recognised. In practice the CFD requires semi-empirical modelling together with the validation. Computational power, especially with water mist systems, is also a limiting factor. This can partly be compensated by multiple cell size meshes and using cluster simulations.

4.3 Reliability

Validation is important for ensuring the reliability and accuracy of CFD simulations. While CFD offers valuable insights, experimental validation remains essential for verifying simulation results and reducing uncertainties.

An iterative approach, guided by adjustments based on experimental findings, is key for enhancing the credibility of CFD predictions. Semi-empirical models, incorporating experimental data and empirical correlations, have demonstrated a practical solution to the modelling of water mist systems.

It is also important to acknowledge the limitations of validation, particularly regarding the complexity of fire scenarios. Oftentimes, all the necessary data, e.g. heat release rate, is not measured during the fire test programmes. Validation simulations should however be incorporated to every water mist simulation report.

It is good practice for any CFD study to carry out a mesh independence study to better understand how the mesh size affects the results and the rationale for selecting a particular mesh size and grid resolution. Such analysis should be reported in the CFD simulation study.

CFD simulations currently cannot replace experimental fire tests as the primary design and approval method, but rather serve as an additional tool.

4.4 User competence

The competence of CFD users significantly influence the quality and reliability of simulations in fire engineering. Fire engineers utilising CFD must possess specialised knowledge in fire dynamics, numerical methods, turbulence models, and combustion processes.

User experience and competence are critical in navigating the complexities of CFD modelling, including parameter selection, model validation, and result interpretation. However, these factors are often underestimated, leading to potential inaccuracies and misinterpretations in simulation outcomes.

As user competence may vary, it is crucial to provide detailed reporting of the modelling process. This ensures that authors and other potentially involved parties can understand the user's actions and the foundational and critical data used as the basis for the model.

5 Conclusions

Fire simulations using Computational Fluid Dynamics (CFD) have become standard practice in fire engineering, driven by advancements in computational capabilities and CFD codes. In certain applications, such as dimensioning smoke management systems, CFD is now exclusively employed. However, the increasing use of CFD has also raised concerns relating to its application to water mist systems, prompting the International Water Mist Association (IWMA) to address these varying approaches in a position paper, highlighting both limitations and possibilities.

CFD simulations with water mist provide valuable insights into thermal analysis and temperature distribution, enabling engineers to optimise system parameters for enhanced efficiency. Despite the benefits CFD offers, it also presents several technical limitations, particularly in water mist modelling. Therefore, CFD cannot replace fire testing, and experimental validation remains crucial for the reliability. An iterative approach, guided by experiences and data on experimental tests, is essential for reducing uncertainties in predicting water mist system performance.

Semi-empirical models have demonstrated accuracy in predicting temperatures and cooling. These models either disregard the impact of water mist on the heat release rate or replicate it from experimental fire tests. Due to their reliability, these models are primarily recommended for use in simulations involving water mist systems.

Accurate modelling requires careful consideration of various factors, including codes, simulation methods, turbulence models, combustion models, droplet size distribution, and especially the skills of the CFD user, who needs specialised fire engineering knowledge. User experience and competence are often underestimated factors. It is recommended that simulations be conducted by organisations with both theoretical understanding and experimental knowledge of how water mist systems operate in real applications and fire tests.

Annex – Typical Q & A

Q1: Can CFD used for providing the design basis for water mist system?

A1: Whether CFD can be utilised to establish the design basis for water mist systems depends on the local Authority Having Jurisdiction (AHJ). However, IWMA does not endorse this practice. The acceptable design basis should be derived from full-scale fire tests in accordance with respected hazard classifications. CFD may be used as an additional tool for further optimisation and assessment, depending upon the availability of validation data.

Q2: What are generic droplet characteristics to be used in CFD modelling?

A2: There are no universally applicable "generic" droplet characteristics for CFD modelling. While some CFD codes may offer basic default characteristics, these must be adjusted to reflect the specific data of each nozzle. Water mist nozzles vary significantly depending on factors such as manufacturer, design pressure, nozzle design, and application. For instance, water mist nozzles used in data centres differ greatly from those used in road tunnels. Therefore, assessment and customization of droplet characteristics are essential for accurate CFD modelling in each specific scenario.

Q3: How can the results of CFD simulations be validated to ensure their accuracy and reliability in predicting the performance of water mist systems?

A3: The accuracy and reliability of CFD simulations for water mist systems can be validated by comparing the results with experimental data from full-scale fire tests or laboratory experiments. Sensitivity analysis (or parameter study), which assesses the impact of different input parameters on the simulation outcomes, can also be done provide insights into model robustness. Some validation data is publicly available, but manufacturers have lots of data from the approval fire tests.

Q4: How accurate are CFD simulations in predicting the effectiveness of water mist systems especially in terms of air flows and temperature distribution?

A4: The accuracy of CFD simulations in predicting the effectiveness of water mist systems varies depending on factors such as the complexity of the fire scenario, the quality of input data, and the assumptions made in the simulation. The combustion and suppression modelling is most difficult. But the combined use of experimental data and semi-empirical models have shown relatively good accuracy in the analysis of gas flows and temperature distribution.

Q5: Are there any case studies or real-world examples where CFD simulations have been successfully used to design or optimise water mist systems?

A5: Yes, there are numerous real-world examples where CFD simulations have successfully aided in the design and optimisation of water mist systems. It is an acceptable method, especially when performance based design methods are used.

Q6: What are the potential cost savings and benefits of using CFD in conjunction with water mist systems compared to traditional design methods?

A6: *Using CFD with water mist systems offers potential cost savings and benefits as it allows for more accurate analysis of fire scenarios, helping optimise system parameters like nozzle placement and protection concept (within approvals). CFD can provide additional data which identifies design flaws early,* *reducing the need for costly modifications later. CFD can also help in the development of the fire test protocols e.g. when ad-hoc testing is used.*

Q7: Are there any regulatory standards or guidelines that govern the use of CFD in designing water mist systems, and how do these standards ensure the reliability of the simulations?

Q7: *To our knowledge, there are currently no specific regulatory standards and guidelines for using CFD in designing water mist systems. However, there are general guidelines available for conducting reliable CFD modelling, along with supporting documents provided by CFD code developers. In cases where specific questions arise, the IWMA Scientific Council can be consulted for guidance on fundamental principles.*