Optimization of water spray system of the Frigel LDK cooler by means of Computational Fluid Dynamics (CFD)

ENSURING PERFORMANCE AND SUSTAINABILITY THROUGH SIMULATION-DRIVEN R&D

ABSTRACT

How the use of CFD simulations allowed Frigel to optimize water consumption and cooling of the LDK

The high performance and environmental sustainability demanded to industrial coolers require nowadays to adopt a change of pace in design strategies compared to the past. The introduction of advanced simulation techniques already in earliest stages of product conception represents a way for persecuting the continuous technical improvements needed.

Frigel has embraced a simulation-driven approach for the design of the LDK cooling system and, with particular reference to the water nozzles used for air humidification, obtained considerable benefits.

1. Spray systems as efficient method for limiting water consumption

Spray systems are commonly used in industry on a wide range of applications and for various purposes (e.g., humidification, cooling, fuel injection, application of coatings, cleaning, lubrication, etc.).

Sprays represent an efficient way for introducing and redistributing a given quantity of liquid into a gaseous medium. In general, a **continuous stream** of liquid is elaborated by pressurized systems (usually referred to as *nozzles*) and discharged in form of **small droplets**.

A wide range of **nozzle types** exists on the market and they can be categorized as a function of some fundamental parameters: flow rate as a function of feeding pressure, shape of the jet (e.g. full cone, hollow cone, mist/fog, flat fan), droplet size (mean diameter and diameter range), opening angle of the jet.

A typical usage of sprays is **humidification** and **cooling of ambient air**[1]. For such applications, the conversion of a water stream into an enormous number of droplets allows to obtain a large interfacial surface between water and air in order to maximize heat transfer and evaporation. In this way, the proper level of humidification and cooling can be achieved efficiently by also **reducing** the impact on environment in terms of **water consumption**. Frigel has adopted a spray system on the LDK cooling machine and optimized it with the use of *Computational Fluid Dynamics* (CFD).

2. What is CFD? How can it be used for optimizing water spray systems?

CFD is a generic term referring to the solution of problems involving fluid dynamics by means of complex numerical algorithms resolved by computers. Currently, in most of the cases, such algorithms implement and solve the *Navier–Stokes* equations that are mathematical expressions that allow to reproduce the movement of a viscous fluid (e.g., air or water) in a defined space called *computational domain*.

The solution of Navier–Stokes equations provides distributions of velocity, pressure, temperature and density of the flow. Unfortunately, such mathematical expressions are as accurate as complex and it is not possible to derive a solution in closed form, except for some extremely simple cases that are not always applicable to the huge variety of industrial problems.

This is the reason why the solution of fluid dynamic problems is numerical, thus demanded to computer algorithms. **CFD has continuously evolved** in the last sixty years[2], also thanks to the enormous increase of computers performance, becoming an important instrument in the hands of design and R&D engineers and is also able to accurately reproduce the behavior of spray flows.

Sprays are characterized by **complex physical phenomena** governed by several parameters (mainly physical properties of liquid and gaseous media involved,velocity of injection, velocity of the gas phase, temperature, pressure, relative humidity and droplet size).

One of the most effective methods to model a flow characterized by an **injection of droplets** is the so-called "*Eulerian-Lagrangian approach*".

In this latter, the gaseous phase (e.g., air) is treated like a continuum (Eulerian approach) for which conservation equations of mass, momentum and energy are resolved. On the contrary, **droplets** are treated as single particles or, more in detail, as "parcels" representative of average conditions of groups of particles.

In this way, the trajectories of the groups of particles are tracked **by solving a force balance** on them (Lagrangian approach). Moreover, reciprocal interactions between gaseous phase and droplets is solved in terms of mass (evaporation and condensation), momentum (forces) and energy exchange. Depending from local conditions, droplets can be subjected to breakup mechanisms[3] (fragmentation of a droplet into smaller droplets) or coalescence.

A further complication is given by the presence of walls on which droplets can come in contact. In such cases, specific models can be used for predicting rebound, splashing or formation of a wall film[4].

As underlined above, the **complexity** of modelling flows characterized by the presence of **sprays** is remarkable. However, with the proper knowledge, **engineers can extract useful information from simulations** (e.g., droplet trajectories, evaporated flow rate, effective amount of wetted surface, droplet velocity, etc.) and exploit them for **improving performance** and **sustainability** of the products.

3. How does product development benefit from the use of CFD simulations?

A wide range of fluid dynamic problems can be approached through CFD, e.g., steady or transient flows, turbulent or laminar flows, single or multiphase flows, compressible or incompressible flows, heat transfer problems (including thermal convection, conduction and radiation), droplet transport, combustion and so on. To address this broad panorama of scenarios, specific models have been developed over the years to reproduce the underlying physics.

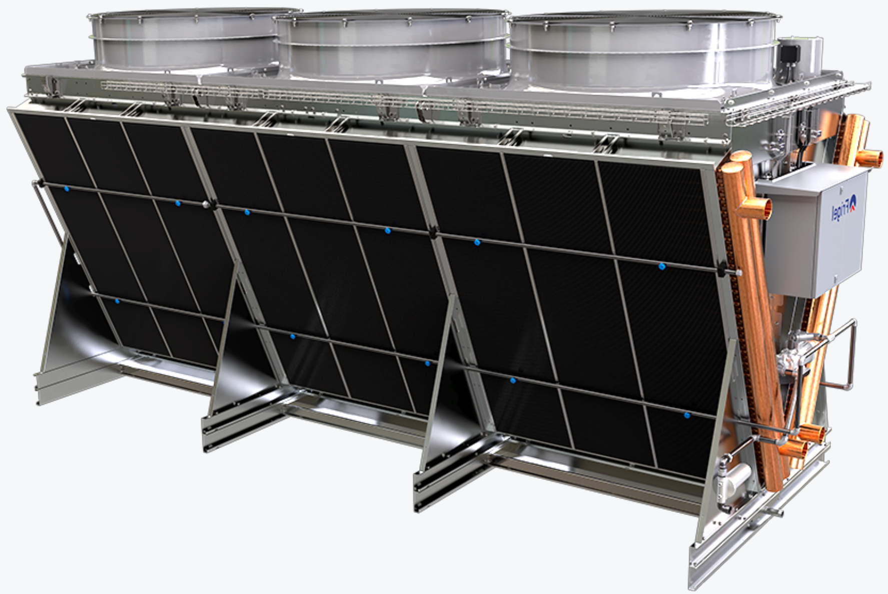
Modern CFD codes are powerful tools that, if put in the hands of technicians with a proper knowledge, are able to provide a deep insight on physical phenomena with a noticeable accuracy and affordable costs.

For such reason, companies who rely on CFD as an instrument to be used during early/mid design stages can significantly reduce costs associated with **massive experimental campaigns**, avoid potential problems arising at **prototype levels** and speed up considerably the time-to-market. Moreover, the reproduction of the physics in a *virtual environment* offers the possibility of **exploring multiple design scenarios** during the design phase so as to naturally tend towards optimized designs.

Frigel has embraced this type of approach, managing to guarantee continuous improvement of its products through ***simulation–driven R&D***, with particular reference to the development of the *LDK cooling system* **Figure A** and to the **optimization of its water spray system**.

Figure A

3D model of the Frigel LDK cooling system with spray technology



4. Application of CFD to the water spray system of the Frigel LDK

During the development of the Frigel LDK cooling system, the **reduction of water consumption was a key target** for the designers. Bearing in mind water saving, the spray system adopted on the LDK for humidification purposes has been extensively studied with the aim of optimizing its behaviour in terms of **proper redistribution of water** and **effective evaporation**.

4.1 Spray model validation

In order to obtain reliable results from simulations, a validation of the computational models used is fundamental. The first step of the activity consisted in **reproducing a simple case** where a spray is injected downward in quiescent air.

This configuration is a typical experimental setup used by nozzle manufacturers for investigating quantitative parameters of the spray. From the computational point of view, it is a useful reference for validating the parameters of the spray model.

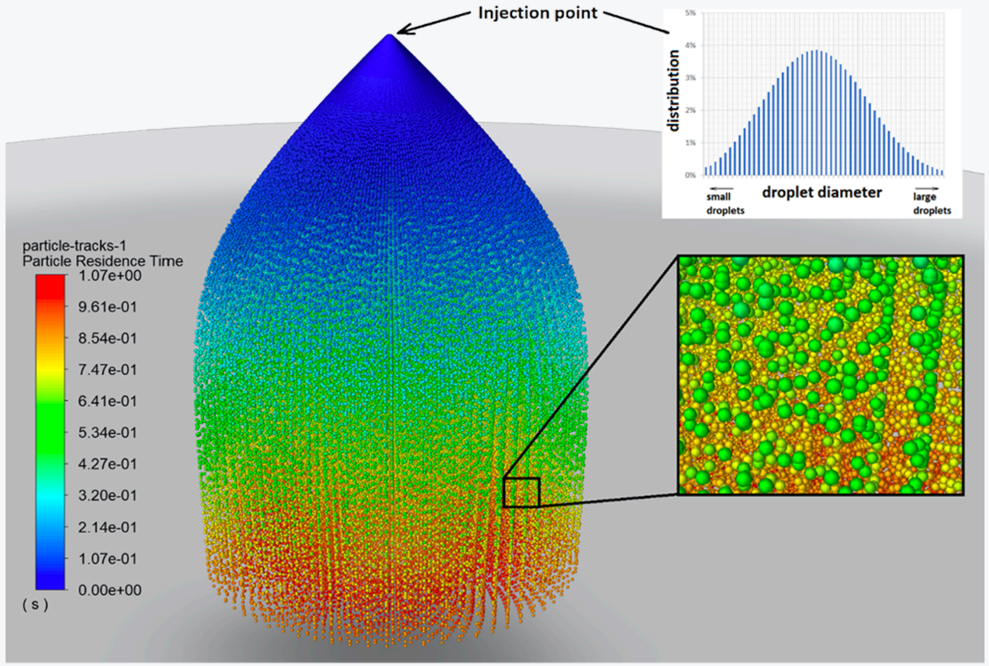
Figure B shows a visualization of the **water droplets injected from the top and gradually deflected by gravity**.

As it is possible to appreciate from the figure, a very large number of parcels is modelled in order to reproduce the realistic behavior of the flow. The size of the injected droplets is a fundamental parameter for a spray simulation. Particular attention was dedicated to reproducing a **realistic distribution of diameters** by means of the calibration of the so-called "Rosin-Rammler" function that describes droplet size.

The spray opening at a given distance from the injection point (this parameter is in fact provided by most of the manufacturers as a function of nozzle type and feeding pressure) is one of the simulation outputs used for model validation.

Figure B

Visualization of water droplets of a vertical downward hollow cone spray in quiescent air. Droplets are colored by the so-called "residence time" (time passed by droplets inside the computational domain). It is possible to appreciate the deflection of droplets trajectories due to the action of gravity. The distribution of water droplet diameters is shown in the top-right part of the figure.



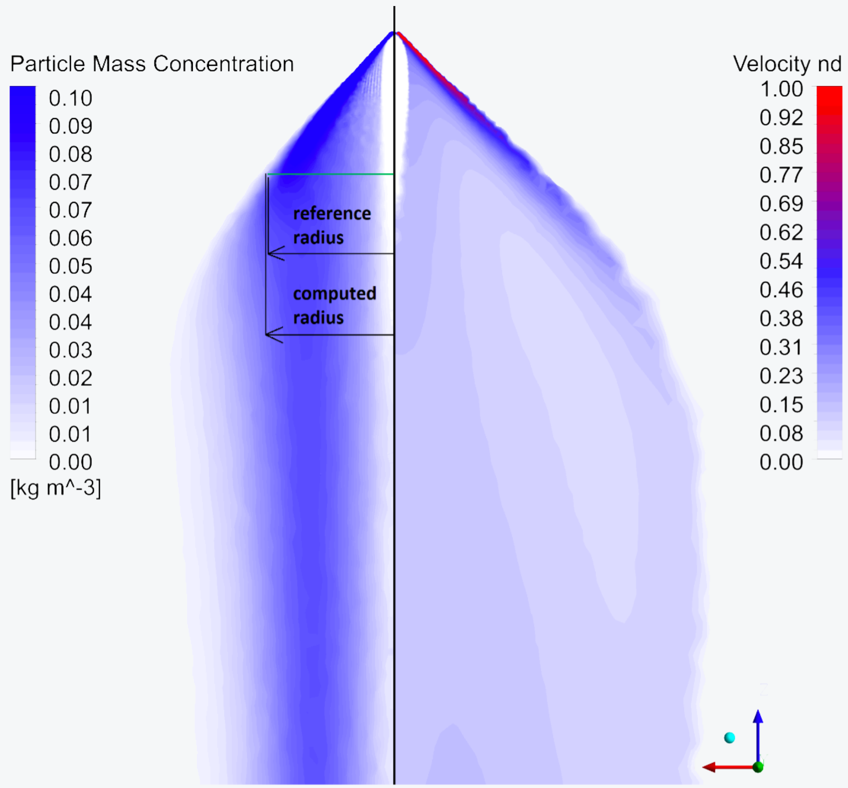
Optimization of water spray system of the Frigel LDK cooler by means of Computational Fluid Dynamics (CFD)

Figure C reports a **section of the spray** of **Figure B on a vertical plane**. On the left part of the figure, where the particle mass concentration is reported, it is possible to see the good agreement between the predicted radius of the spray and the reference one (obtained from manufacturer's data).

The right part of **Figure C** shows the non-dimensional distribution of droplet velocity. From this latter part, it is possible to appreciate how a **high-speed flow is present in proximity of the injection point** while particle velocity gradually decreases moving away from the nozzle.

Figure C

Visualization of a section of the spray of Figure B on a vertical plane. Left part: mass concentration of droplets and comparison between the reference radius of the spray and the computed one. Right part: non-dimensional distribution of droplet velocity.



4.2 Modelling of the spray system on the LDK machine

The preliminary validation activity described above allowed for obtaining a reliable model able to reproduce the actual spray behavior in isolated conditions.

The calibrated model was then used in subsequent activities **for optimizing the spray system on the LDK machine**. The target of the activity was to control air humidification without wetting excessively the coils.

This aspect is fundamental from the point of view of both **hygiene and safety**. Several water nozzles are present along the LDK coils.

For each injection point, the spray characteristics have been reproduced numerically as shown in § 4.1. In this case, **water jets** describe more complex trajectories since nozzles are mounted with **inclined axis** with respect to vertical direction and are **immersed in an airflow** entering the coils.

Simulations have been used to analyze the jet behavior under different working conditions and to **properly adjust water flow and nozzle orientation**.

Figure D shows a visualization of the spray “clouds” by means of isosurfaces of droplet mass fraction. It is well observable from the figure how the sprays, having an initial hollow-cone shape, are gradually deformed by the action of gravity and the incoming air flow on the coils.

From CFD simulations it has been possible to predict the **complex dynamics of droplet trajectories** from the injection points to the coils and consequently to understand how to **calibrate the spray** system in order to obtain an **adequate coverage** of the coils themselves.

Figure D

Visualization of the spray clouds along the coils of the LDK by means of isosurfaces of water droplet mass fraction. It is possible to observe how the water jets, with initial hollow-cone shape, are transported on the coils by the air flow. Air flow exiting from the top of the fans is visualized through the red streamlines.

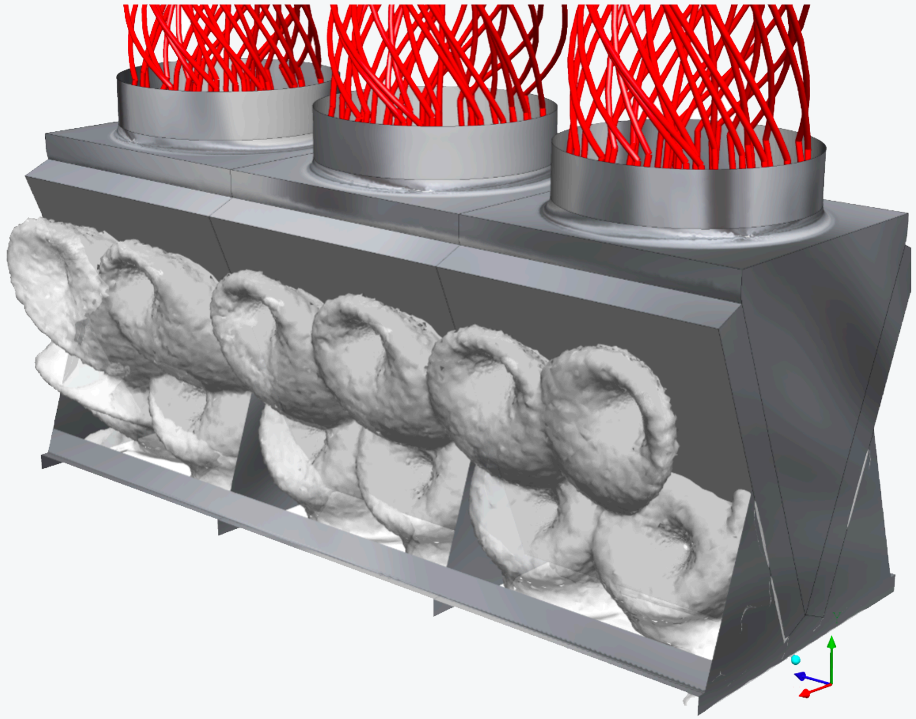


Figure E shows a visualization of the portions of the coils reached by the water sprays (blue areas). It is possible to note how, in the zones where water nozzles are positioned, coils are effectively covered by water jets.

Through the use of the CFD simulations is has also been possible to **minimize the overall water flow consumption** by means of the reduction of the portion of injected water that, under the effect of the air flow and gravity, did not directly reach the coils. Such quantity has been limited to a very small portion of the total injected water, obtaining a **remarkable result in terms on environmental impact**.

Figure E

Qualitative visualization of the coil portions interested by water flow (blue regions). Where water jets are present, the coils are uniformly covered by water droplets.

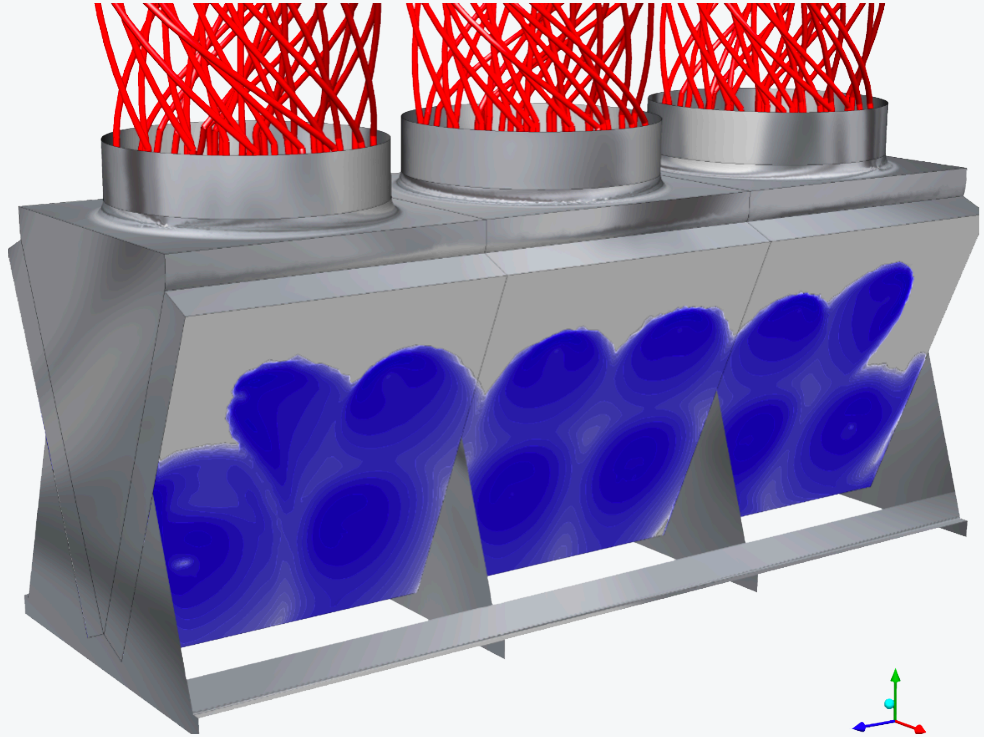
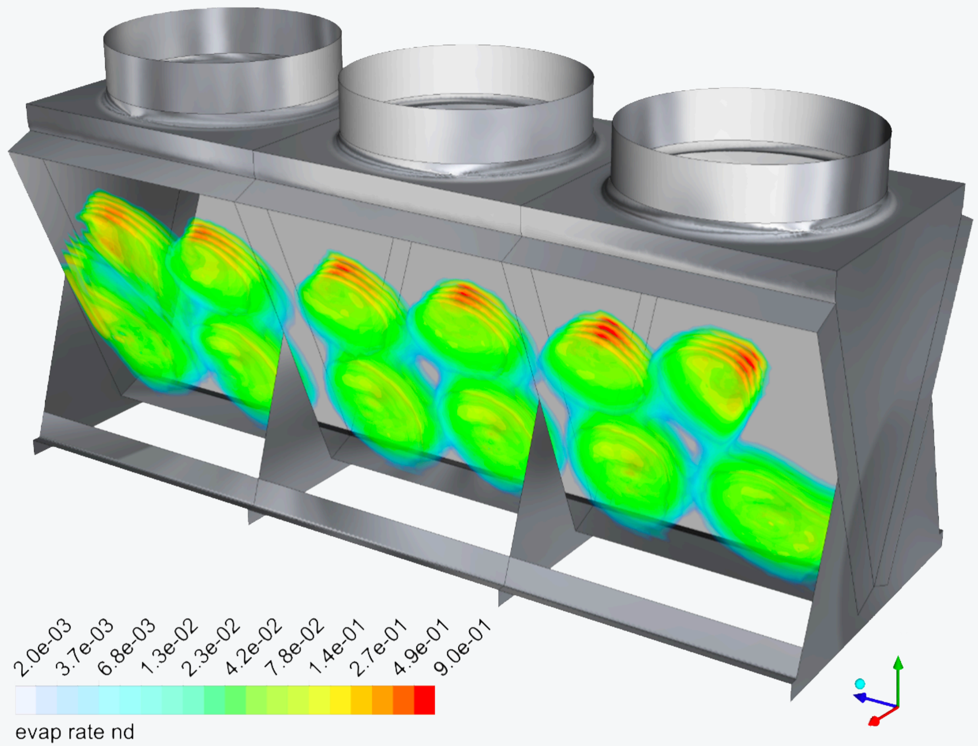


Figure F shows a 3D visualization of the non-dimensional evaporation rate on the coils. To ensure a high evaporation rate means to have a **faster transition of water** from the liquid phase (droplets) to the gaseous one (humid air). This aspect is strongly correlated to ambient conditions (temperature and relative humidity) as well as to the spray characteristics and, in more detail, to the **droplet velocity and their size** (smaller droplets tends to evaporate faster due to the higher interfacial area between liquid and gas phase).

Simulations allowed for understanding the effects of the aforementioned parameters and to optimize the system for **obtaining the proper air humidification** and consequently an efficient cooling. Moreover, **CFD calculations allowed for avoiding excessive wetting** of the coils in order to **minimize water consumption** and prevent operational problems.

Figure F

Visualization of the non-dimensional water evaporation rate on the coils. Red color means faster passage of water from the liquid phase (droplets) to the gas phase (humid air).



Optimization of water spray system of the Frigel LDK cooler by means of Computational Fluid Dynamics (CFD)

CONCLUSIONS

To ensure performance and environmental sustainability of products is nowadays fundamental not only for the success on the market but, mostly, for ethic reasons.

CONCLUSIONS

For industrial cooling systems, performance is directly correlated to **energy and water consumption** and both have an **impact on environment** as well as on **operating costs of a plant**.

Frigel knows very well this strict correlation and has always paid great attention to the aforementioned aspects during the development phases of its products. However, the high performance demanded requires now to adopt a change of pace in **design strategies** compared to the past by introducing **advanced simulation techniques** already in earliest stages of machine conception.

Frigel has embraced a *simulation-driven approach* for the design of the **LDK cooling system**, with particular focus on the **technical aspects** that make the difference: **aerodynamics**, **heat transfer**[5] and **water management.** **CFD simulation** has been adopted by Frigel as **fundamental** support for helping engineers in taking the proper technical decisions.

In particular, thanks to the use of CFD simulation, the **water spray system** of the LDK has **been optimized** for ensuring an **efficient cooling** and **minimizing** the **water quantity needed**, thus ensuring high **performance and environmental sustainability**.

Authors

Isacco Stiaccini, PhD
Frigel R&D Dept

Giovanni Galoppi, PhD
Frigel R&D Dept

Marco Terenghi
HRS Business Development Unit

Massimiliano Insinna, PhD
CReAI – Centro Ricerche e Attività Industriali

References

[1] Montazeri H., Blocken B., Hensen J.L.M.. Evaporative cooling by water spray systems: CFD simulation, experimental validation and sensitivity analysis. Building and Environment (2014).

[2] Shang J.J.S.. Landmarks and new frontiers of computational fluid dynamics. Advances in Aerodynamics. 1, 5 (2019).

[3] Ashgriz N.. Handbook of Atomization and Sprays - Theory and Applications. Springer US (2011).

[4] Adeniyi A.A., Morvan H. P., Simmons K.A.. A coupled Euler Lagrange CFD modelling of droplets-to-film. The aeronautical journal (2017).

[5] Stiaccini I., Galoppi G., Terenghi M., Insinna M.. Use of Computational Fluid Dynamics (CFD) for performance optimization of the Frigel LDK adiabatic cooling system.

Use of Computational Fluid Dynamics (CFD) for performance optimization of the Frigel LDK adiabatic cooling system

WHITE PAPER