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

ACHIEVING HIGH-EFFICIENCY COOLING THROUGH SIMULATION-DRIVEN R&D

ABSTRACT

CFD has become a powerful instrument in the hands of design and R&D engineers.

With specific expertise in the fields of fluid dynamics and heat transfer, CFD can provide a strong acceleration to the design process of cooling systems and can help in guarantee high efficiency of the product. The present paper describes how Frigel has exploited the benefits of CFD for the design of its LDK adiabatic cooling system.

1. What is CFD?

Computational Fluid Dynamics 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, not 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[1], also thanks to the enormous increase of computers performance, becoming an important instrument in the hands of design and R&D engineers. The basic approach consists in the following steps: geometry preparation, spatial discretization of the computational domain (mesh generation), simulation setup, run, and post-processing of the results. Each of these phases requires specifical expertise, with which it is possible to exploit the full potential of CFD.

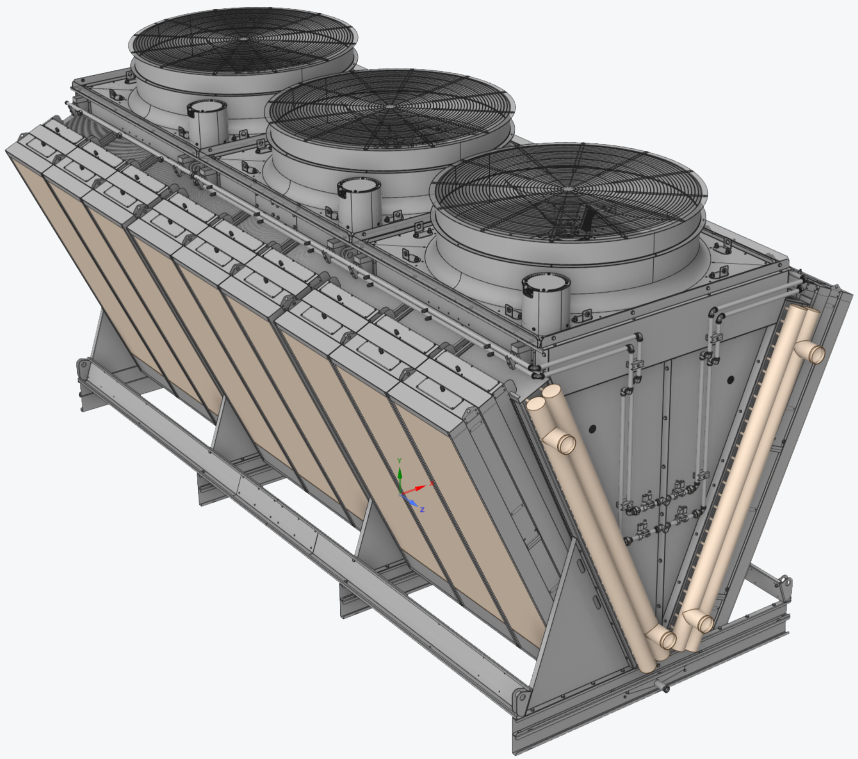
2. What are the benefits of using CFD during product development?

A wide range of fluid dynamic problems can be approached through CFD, e.g., steady or transient flows, turbulent and laminar flows, single or multiphase flows, compressible or incompressible flows, heat transfer problems (including thermal convection, conduction and radiation), particle 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. The result is that, 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 relies on CFD as an instrument to be used during early/mid design stages can significantly reduce costs associated to 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 adiabatic cooling system* (**Figure A**).

Figure A

3D model of the Frigel LDK adiabatic cooling system



3. Application of CFD to the development of the Frigel LDK adiabatic cooling system

Multiple aspects are involved in the development process of an industrial cooling system: aerodynamics and heat transfer through heat exchangers, aerodynamics of internal volumes, aerodynamics and aeroacoustics of fans, and aerodynamic interaction with the external ambient.

3.1 Heat exchangers optimization

Frigel has dedicated a strong effort in the selection and optimization of the heat exchangers for the LDK adiabatic cooling system.

The heat exchanger is one of the main elements that determine the overall efficiency of the system. A proper design of a fin and tube heat exchanger requires to *maximize the thermal power exchanged and contemporary minimize pressure losses* encountered by the fluids on both air and liquid side.

A multitude of parameters is involved in the optimization of a heat exchanger, e.g., number of tube ranks, shape and thickness of the tubes, pitch and thickness of the fins, shape of the fins[2]. CFD plays an important role in guiding the right choice of such parameters and can provide accurate performance estimations of various types of geometries[3] [4].

Figure B and **Figure C** show some examples of the results obtained by Frigel on various heat exchanger configurations. The level of insight offered by CFD in understanding heat transfer and mechanisms of pressure loss generation is much more powerful respect to the very limited number of information obtainable through experimental testing. For such reasons, the expensive experimental tests can be restricted to the final validation phase of the development process that can be carried out through CFD simulations.

Figure B

Non-dimensional pressure distribution on a portion of fin and tube heat exchanger. CFD allows to accurately reproduce the pressure drop across the fins and to optimize the heat exchanger geometry in order to increase the overall machine efficiency.

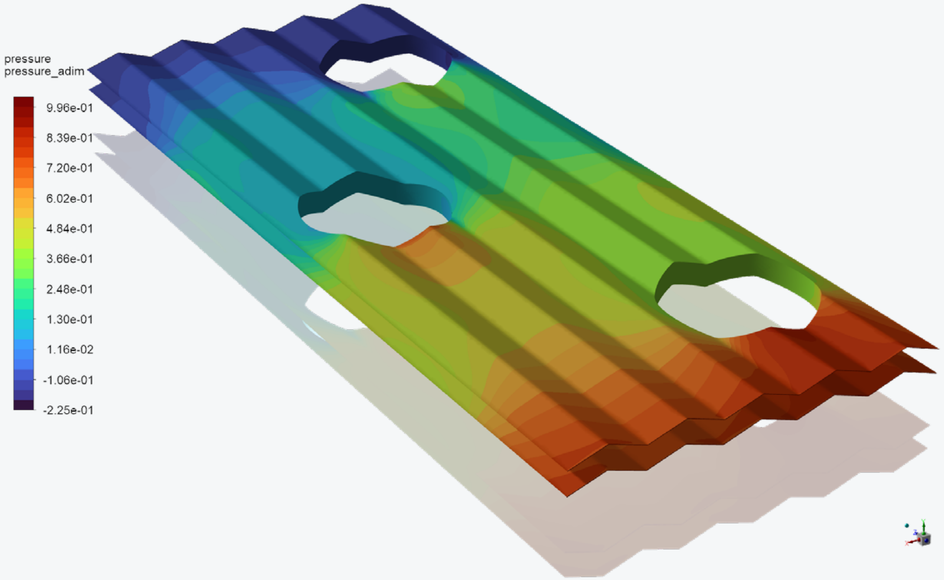
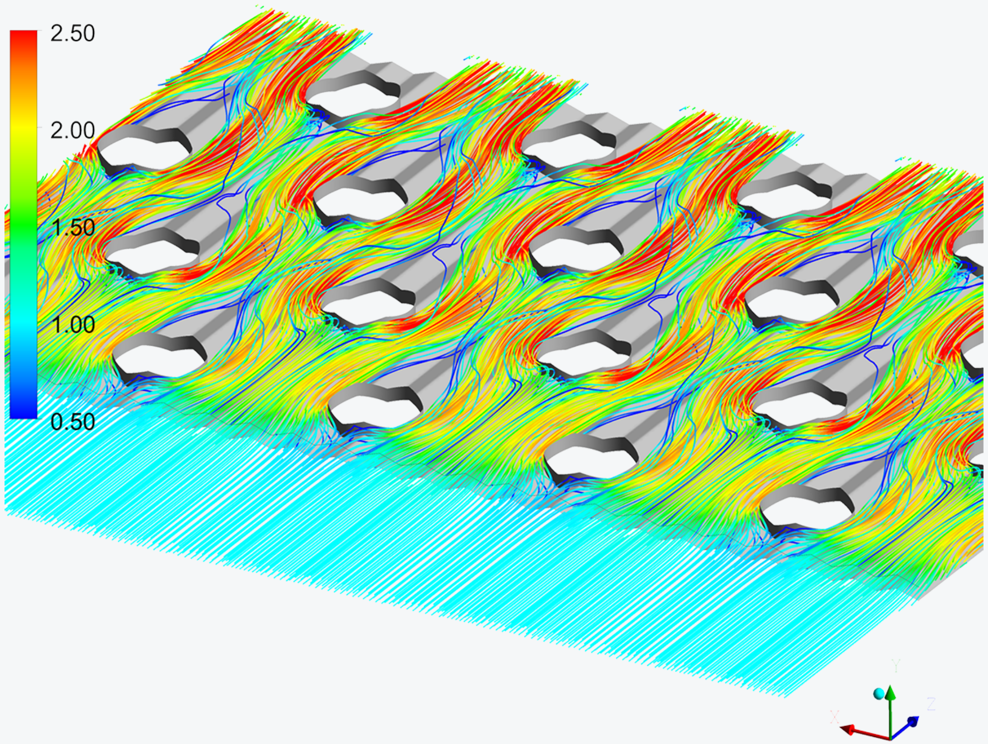


Figure C

Streamlines through a portion of fin and tube heat exchanger colored by non-dimensional velocity magnitude. It is possible to appreciate the acceleration of the flow due to the presence of the obstacles represented by the tube ranks.

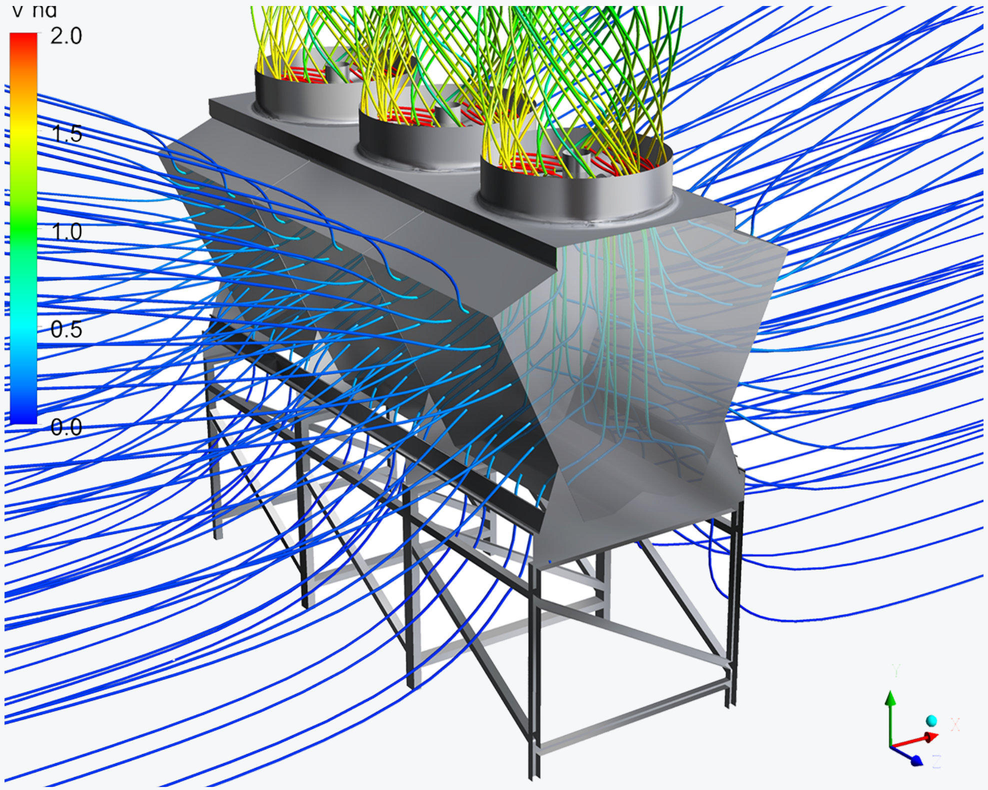


3.2 Study of plenum configuration

V-shaped adiabatic coolers are characterized by inclined heat exchangers with respect to the vertical plane. Such arrangement allows to minimize the overall machine volume and to contemporary increase the overall amount of heat transfer surface. Moreover, the heat exchangers angle also defines the shape of the plenum located immediately upstream of the extraction fans which is extremely important for minimizing the power consumption of this latter. The proper choice of the inclination angle of heat exchangers is not straightforward. Frigel has performed a simulation campaign with the use of CFD in order to find the best configuration (**Figure D**) and managing to *minimize the formation of flow non-uniformities upstream of the fans.*

Figure D

Streamlines through the Frigel LDK adiabatic cooler colored by non-dimensional velocity magnitude. The flow crosses pads and heat exchangers thanks to the optimized pressure rise provided by the fans.



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

3.3 Fan performance optimization

Extraction fans, located on the top of the machine provide the pressure rise needed to reach the design air flow rate. Unoptimized fans are potential source of reduction of overall efficiency, vibration and noise generation[5]. The fan performance strongly depends from the upstream elements and, in particular, from the uniformity of the flow inside the plenum. For this reason, particular attention has been dedicated by Frigel in *optimizing the interaction between fan and plenum as well as in optimizing shape of the fan blades and the fan supporting structure* (**Figure E**). Concerning this latter, both aerodynamics and mechanical resistance have been simultaneously considered during the design phase, leading to high efficiency, reliability and limited noise emissions.

3.4 Study of the effects of the proximity of multiple machines

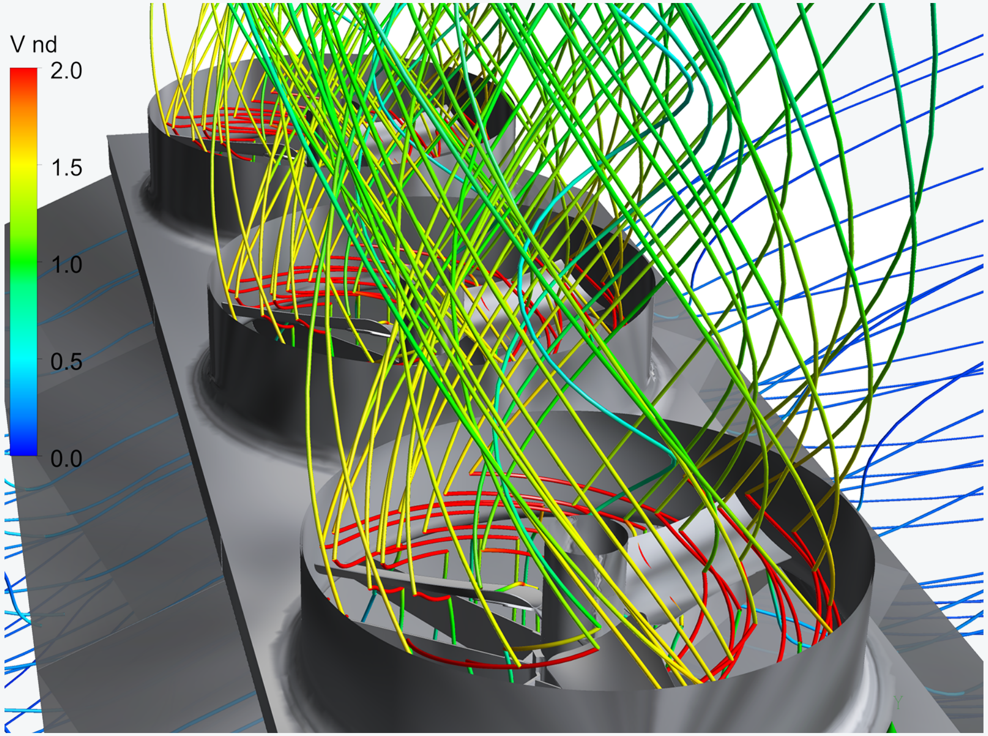
It is not only necessary to guarantee high performance of the single machine but it is also fundamental to think about the configurations in which they will be installed. For this reason, Frigel performed studies about the effects of multiple flanked LDK adiabatic coolers. Such studies, developed with the support of CFD simulations (**Figure F**) allowed to *understand the optimal distance at which the machines should be positioned* in order to contemporarily minimize both overall installation footprint and power consumption.

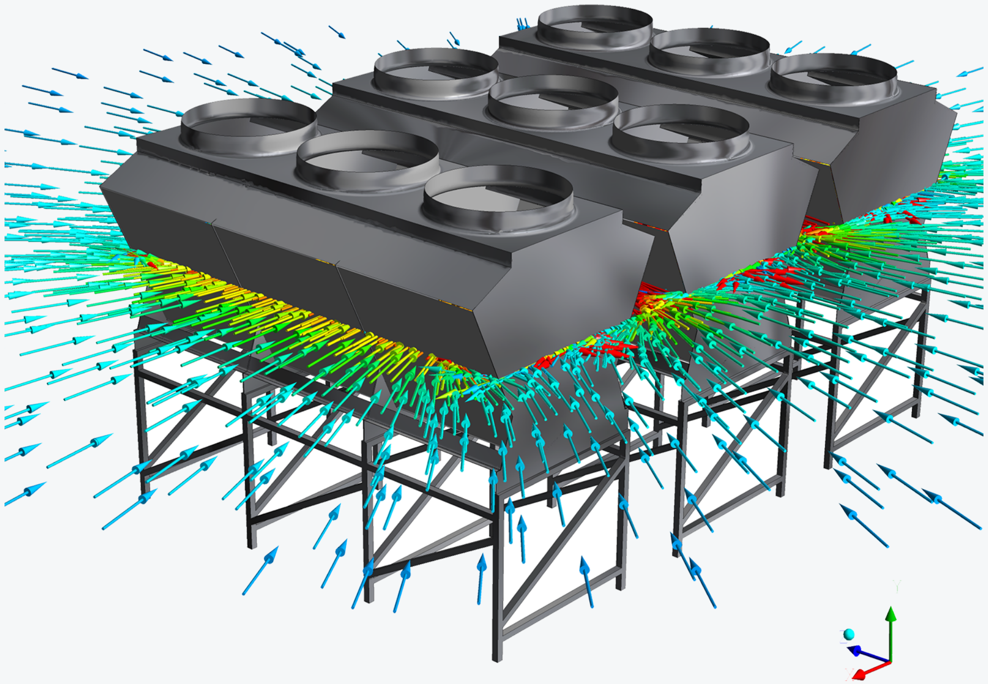
Figure E

View of the flow exiting the fans positioned on the top of the Frigel LDK adiabatic cooler. Streamlines are colored by non-dimensional velocity magnitude. A proper control of the discharge flow allows to reduce noise emission as well as to increase the overall machine performance.

Figure F

Velocity vectors of the air flow entering three flanked modules of the LDK adiabatic cooling system. Distance between modules is optimized for minimizing overall footprint and power consumption.





CONCLUSIONS

Understanding aerodynamic and heat transfer phenomena is the key point for exploiting potential performance improvement of cooling systems.

The use of modern powerful CFD simulation tools, combined with the expertise of design and R&D engineers, allowed to have a wide overview on a multitude of design aspects involved in the development process of the LDK cooling system (e.g., heat transfer and pressure losses inside heat exchangers, internal aerodynamics of the plenum, fan aerodynamics and fluid dynamic aspects correlated to the machine installation). The deriving knowledge on the fundamental points that determine the overall machine performance, allowed Frigel to reach a high level of efficiency with the LDK adiabatic cooling system.

Frigel

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

[2] Raja B.D., Patel V., Jhala R.L.. Thermal design and optimization of fin-and-tube heat exchanger using heat transfer search algorithm. Thermal Science and Engineering Progress (2017).

[3] Ó Cléirigh C.T., Smith W.J.. Can CFD accurately predict the heat-transfer and pressure-drop performance of finned-tube bundles? Applied Thermal Engineering (2014).

[4] Bacellar D., Aute V., Radermacher R.. CFD-based correlation development for air side performance of finned and finless tube heat exchangers with small diameter tubes. 15th International Refrigeration and Air Conditioning Conference at Purdue, July 14-17, 2014.

[5] Yang L., Hua O., Zhao-Hui D.. Optimization design and experimental study of low-pressure axial fan with forward skewed blades. International Journal of Rotating Machinery (2007).