Author

Dr. Andreas Zürner
Research
Güntner GmbH & Co. KG

Member of Güntner Group

Use of numerical flow simulations (CFD) for optimising heat exchangers

Numerical flow simulations (Computational fluid dynamics) have already been used successfully for many years now at Güntner to handle the most varied tasks. The article to hand briefly outlines the method involved and uses a few examples to illustrate the extent to which insights and findings have been gained as a result, which would be scarcely feasible using standard measurement methods or only with a great degree of difficulty.

An increasing number of publications on the topic of computational fluid dynamics (CFD) in the area of finned, air-cooled heat exchangers can be interpreted as an indication that CFD will become an indispensable tool for developing new units in the foreseeable future. Most work at present is still coming from the university environment – generally projects running over several months or even years, which look at a specific application in detail in order to answer fundamental questions comprehensively.

Rapid progress in the area of chip manufacturing, however, is now also allowing CFD to be used efficiently for simpler tasks with little expense and effort. Numerical flow simulations of consistently high quality can meanwhile even be performed with normal desktop PCs within an acceptable time period. And if larger projects with an increased need for computing capacity ever have to be processed, then cloud computing offers the possibility of cost-effective leasing of the required resources on the web.

What is CFD?

In simple terms, computational fluid dynamics sets out to describe fluid motions with the aid of computers. The concept of fluid dynamics is based largely on the Navier-Stokes equations and their variations. In general terms, only a few very specific problems can be resolved analytically with these partial differential equations. This means that to solve the actual problems that confront us on an everyday basis, iterative approaches have to be used almost without exception.

By nature, however, these can then „only" deliver approximate solutions. The deviations in this respect should be understood relative to an analytically precise solution and are still smaller in general by orders of magnitude than would normally be interpreted as a deviation. Because iterative approaches generally involve significant computational effort, computers are usually used for these tasks.
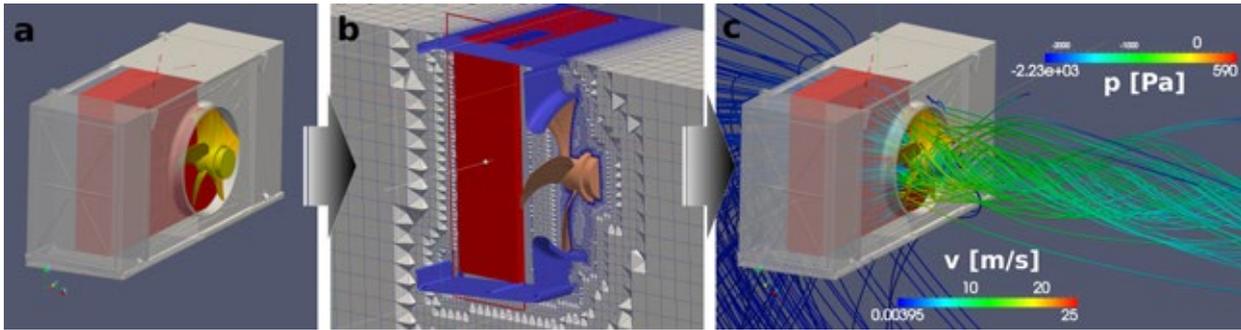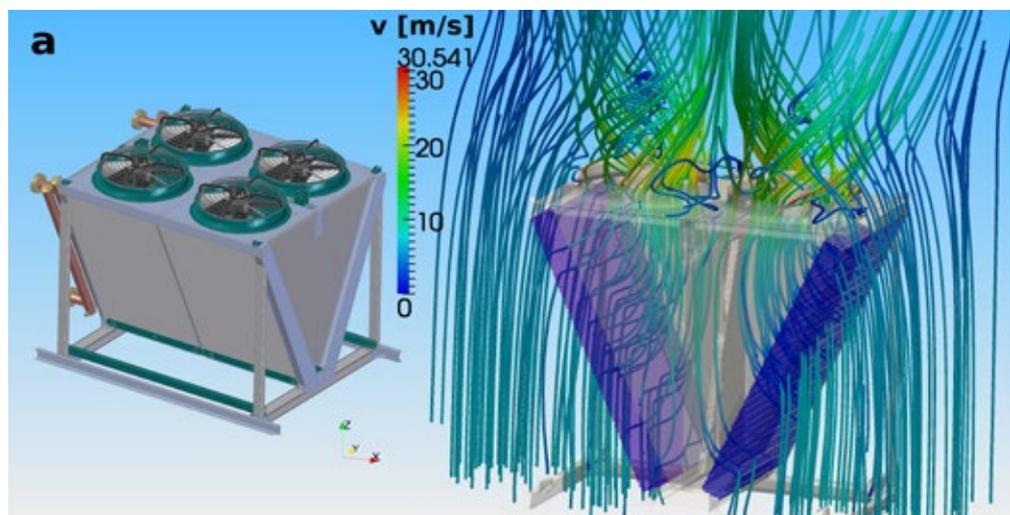
Picture 1:

a CAD drawing for a given problem (air flow through a cooler in a cold room).

b Cross-section through the computational mesh to visualise the individual cells.

c Sample evaluation of the iteratively calculated results. Flow lines indicate the path of imaginary particles.

Basic approach

A corresponding (generally 3D) drawing is first created for a given problem using a CAD program. **Picture 1a** shows an example of a cooler in a cold room. The entire area of relevance – in our example consisting of cold room, heat exchanger, casing and fan – is split into a number of small, individual cells (**Picture 1b**). This step, which is termed „meshing", determines the points at which the calculation software calculates the individual simulation variables (pressure, velocity, temperature, etc.). Meshing therefore contributes significantly to the precision of the simulation results and has to be matched to the problem at hand – a task that is still not fully automated and therefore demands a certain level of experience and basic expertise in computational fluid dynamics on the part of the user. Following on from the meshing step, additional parameters that are important for simulation are specified, such as definition of the interfaces, selection of solvers and the turbulence model but also substance-specific fluid properties.


Picture 2:

Examples of using CFD simulations for developing new products:

a Even before constructing initial prototypes, it was possible to precisely examine the air flows around and within the new container versions of the GFD and GVD. In particular, the optimum position of the spray nozzles for the new HydroSpray system was defined based on the simulation results.

b The casing of the new FLAT series was likewise examined under fluid mechanics aspects and, above all, optimised with regard to the distance between fan plate and heat exchanger coil.
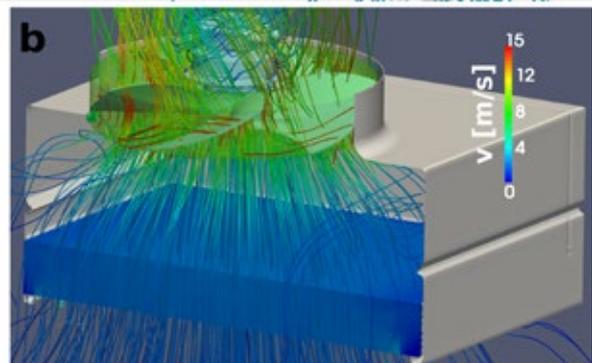
The points just mentioned are generally summarised under the term „preprocessing". We use our own internal CFD workstation for this purpose. The actual simulation, i.e. the iterative solving of the equation systems, is performed on the basis of cloud computing using servers leased on the web. Thanks to this extremely flexible solution, we are virtually unrestricted in terms of computing and memory capacity and reduce running costs to a minimum. Once the simulation is complete, the results are transferred back onto the company server and evaluated there accordingly (postprocessing). **Picture 1c** shows an example of the result of such an evaluation in which the flow is visualised with the aid of flow lines.

Existing measurement data is used where possible as a basis for validating the simulation results. The convergence of specific parameters (generally pressure or temperature), the independence of the results from the level of discretisation and the residuals of the individual iteration steps continue to be drawn on.

Possible applications of CFD

The range of applications for CFD simulations is extremely diverse in the refrigeration and air conditioning industry. The development of new products undoubtedly tops the list here. Even today, critical product-specific problems are examined in advance at Güntner using CFD, without even a single sheet having to be bent or a hole drilled. For example, the optimum position of the spray nozzles on the new HydroSpray system was determined based on simulation results (see **Picture 2a**). But also the casing of the new FLAT series was successfully optimised under fluid mechanics aspects in terms of the distance between the fan plate and heat exchanger coil – without the need to perform costly and time-consuming sets of measurements on prototypes (see **Picture 2b**).

Likewise, products that are already on the market can be investigated systematically using CFD. Are there possibilities for improvement in specific areas? How does the unit behave in specific installation locations or under special circumstances? **Picture 3** shows an example of the numerically calculated flow in an evaporator (GHN080.2), which is secured to the ceiling at different distances to the wall. It was successfully demonstrated that an inadequate wall distance can lead to more significant performance losses especially in direct evaporation mode. Good use was in turn made here of the fact that numeric simulations can be used as a comparatively fast and cost-effective means of examining a range of different variants.

Last but not least, extremely valuable information can be obtained with CFD simulations in relation to very basic questions, for example concerning flow, pressure drop and temperature distribution in finned heat exchangers
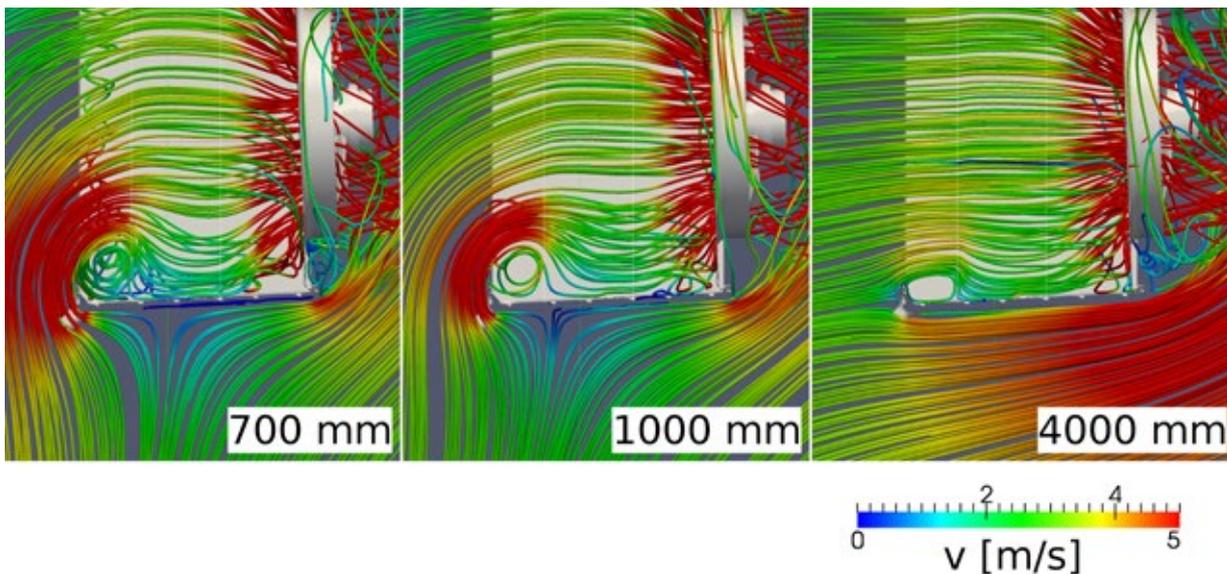
Picture 3:

Numerically calculated flow in a GHN080.2 with different wall distances:

If the distance to the wall is reduced, the air vortex before the heat exchanger increases. This vortex impairs the flow in the heat exchanger especially at very small distances (700 mm, see picture on left) and can therefore lead to considerable performance losses, specifically in DX operation.
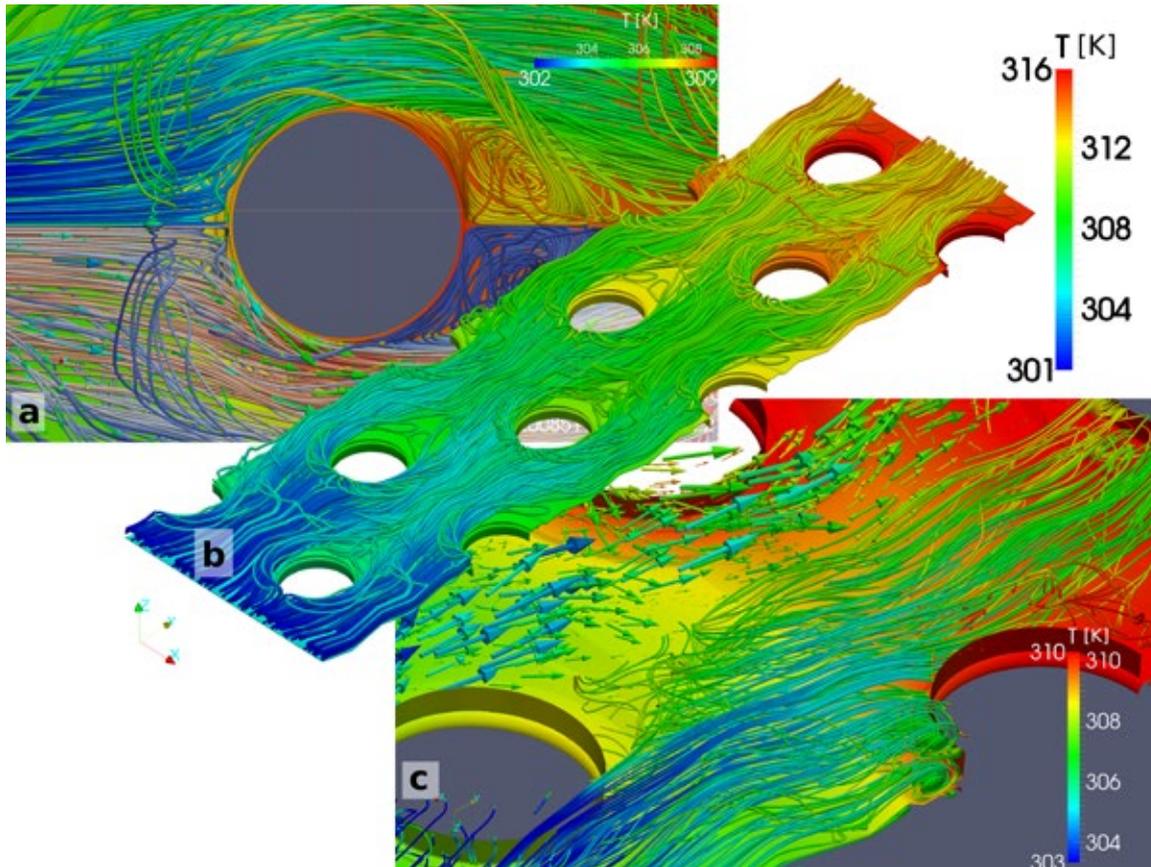
Picture 4:

Calculation of heat transfer and pressure drop in finned heat exchangers:

Owing to the symmetry properties of the fins, only a small cut-out (**b**), needs to be simulated in order to obtain detailed information on the flow, the pressure drop (not shown in picture) and the temperature distribution in the heat exchanger (**a, c**)

(see **Picture 4**). The possibility to analyse areas and/or physical dimensions that can be examined only with significant effort from a measurement perspective should certainly also be emphasised here, e.g. the air flows around the core tubes (**Picture 4a, c**) but also local temperatures and pressures. Moreover, routine calculation of downstream pressure drop and heat transfer from numerical flow simulations is now so precise as to replace actual measurements – at least in the first instance. This means that the most promising candidates can be determined from a larger number of fin variants from an optimal cost and time perspective, without the need to construct and measure a prototype specifically for each variant.

References

[1] Hwang S.W., CFD analysis of fin tube heat exchanger with a pair of delta winglet vortex generators, Journal of Mechanical Science and Technology 26 (9) 2949~2958

[2] Singh, V., DEVELOPMENT OF AN ADVANCED HEAT EXCHANGER MODEL FOR STEADY STATE AND FROSTING CONDITIONS, Dissertation, University of Maryland, 2009

[3] Suga K., Numerical study on heat transfer and pressure drop in multilouvered fins, Journal of Enhanced Heat Transfer 2 (3) 231~238

[4] Singh V., Simulation of air-to-refrigerant fin-and-tube heat exchanger with CFD-based air propagation, International Journal of Refrigeration 34 (8) 1883~1897

[5] Sun L., Evaluation of elliptical finned-tube heat exchanger performance using CFD and response surface methodology, International Journal of Thermal Sciences 75 45~53

[6] Perrotin T., Thermal-hydraulic CFD study in louvered fin-and-flat-tube heat exchangers, International Journal of Refrigeration 27 (4) 422~432

Summary

In the field of science, numerical flow simulation (CFD) has also been established for some time in the area of finned, air-cooled heat exchangers. Increasingly powerful computers additionally ensure that small and medium-sized companies in this sector can also perform their own CFD analyses cost-effectively. Comparatively cost-effective computing capacity can be leased on the web for larger projects.

This article should provide a concise overview of the different areas within Güntner where this tool is already being used successfully. The air flow in and to the unit can therefore be studied in greater detail as early as during the development stage of a new product. Moreover, the findings from numerical flow simulations are of enormous benefit when it comes to improving current products or examining specific applications in greater detail. CFD simulations also provide extremely helpful results in terms of answering fundamental questions. For example, it is also possible to routinely examine pressure drops and heat transfers in finned heat exchangers in sufficient detail and to visualise strong turbulences and reverse flows.

It goes without saying that it cannot be and also is not the aim of Güntner to develop new solvers or turbulence models for the numerical flow simulation of heat exchangers or to simulate a cooler down to the finest detail. Instead, it is much more a question of using existing tools efficiently wherever it makes sense because, for example, measurements would be too costly. Measured data always forms the basis on which the simulation has to be initially validated before further calculations can be performed with regard to the actual question.