
GÜNTNER

www.guentner.eu

Technical article from
10.12.2017

## Authors

**Andrea Belloni**
*EPC/Güntner GmbH & Co.KG*

**Dr Andreas Zürner**
*Research/Güntner GmbH & Co. KG*

Güntner GmbH & Co. KG
Hans-Güntner-Straße 2 – 6
82256 FÜRSTENFELDBRUCK
GERMANY



# *Efficient engine cooling despite high outdoor temperatures thanks to Computational Fluid Dynamics*

**Ensuring effective engine cooling in the face of high outdoor temperatures requires more information than just the average ambient temperature to be available at the planning stage. A system's design can be particularly critical if outdoor temperatures very seldom drop below 30 °C and the dissipation of waste heat becomes a real technical challenge. Computational Fluid Dynamics vividly depicts the fluidic interaction of individual components with their immediate surroundings so that the potential for design and engineering-related optimisations can be fully exploited. As such, the arrangement and design of containers of a gas engine power plant can be specifically adapted for a particular site.**

Fluid flows are complex and cannot, generally speaking, be measured from an analytical perspective. However, with the help of iterative methods it is in fact possible to calculate them and, once the findings have been properly processed, make them visible and comprehensible. Computational Fluid Dynamics (CFD) is the name given to the tool that, thanks to modern computational power, can visually depict even complicated flow conditions using mathematical and physical model calculations. Parameters such as the direction and velocity of fluid flows, temperature distributions and even conjugate heat transfer caused by flows can be visualised with this tool in two and three-dimensional representations.

With CFD, all the relevant physical variables can be displayed at once for fluids which either flow through an object or around an object. The parameters that are important for heat transfer (i.e. convection, radiation and heat conduction) can be made visible and therefore "measurable" even in complex and particularly small or large geometries which cannot be easily recreated for testing purposes. This process significantly speeds up the project planning stage as the time spent on virtual fluid dynamics is considerably shorter than with the classic trial and error approach. CFD also provides an array of data that is completely inaccessible using conventional measurement and testing procedures, that is, unless economically unjustifiable costs are incurred.

## CFD on site

The performance of air coolers can often be heavily adversely affected by areas on site where heat transfer is problematic, for example due to unfavourable conditions relating to the architecture or the regional landscape, or even installations which are particularly susceptible to windy conditions. Güntner therefore uses CFD in external areas with particularly challenging requirements in order to ascertain the influence of buildings and other constructions on wind and other air flows early on in the planning stage and test their impact on the heat dissipation of heat exchangers as part of a second step.

If problematic areas are indeed pinpointed, not only can the performance of the dry coolers and condensers be improved within the simulated environment through specific changes but the installation site and the geometry of the Güntner aggregates can also be optimised. Such simulations can be highly complex since they need to cover a broad range of parameters, for example heat transfer, pressure loss, air flows within narrow spaces and the calculation of buoyant forces and drag forces.

## Modelling, calculation and evaluation

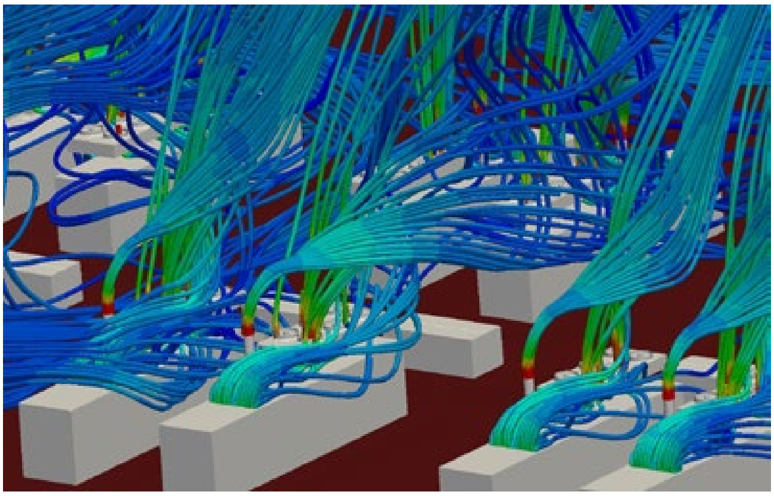
CFD typically starts with the individuals involved ascertaining the challenge or issue to be overcome and building a three-dimensional model. The appropriate computational models are then selected and an initial simulation is started. Once the results have been tested in terms of plausibility, optimisations are carried out, i.e. specific changes are made to particular variables of the three-dimensional model.

The CFD tool calculates both gradual and creeping (laminar) flows as well as turbulent flows as part of the simulation. Heat-related buoyant forces are also taken into consideration. However, non-isothermal and conjugate heat transfer and other multiphysical cases of coupling such as condensation and humidity can also be combined. The CFD tool ascertains the direction of the flow and its corresponding heat propagation, for example, as a visible and evaluable result, and displays them in a two or three-dimensional image.

## First step: assessment of the initial situation

In the past, during high outdoor temperatures, slumps in performance were a constant problem at an existing gas power plant of an MTU engine partner in Myanmar as, in extreme temperatures, the engine heat could not be sufficiently dissipated from the “container-engine-ambient air” system, even though the heat exchanger capacity was more than sufficient from a purely mathematical perspective. At this gas engine power plant, flat bed fluid coolers dissipate the heat from 100 enclosed gas generators (at 1.8 MW) into the environment.

The containers with the coolers positioned on their roof are arranged on the premises in two long rows perpendicular to the predominant wind direction. Each of the containers has a stack for the engine exhaust gases and an outlet in the roof to dissipate the engine heat from the container.
In order to get to the bottom of the issue of the slumps in performance and visualise exactly what was going



**Picture 1:**
More information than just the average ambient temperature is required in order to properly design and plan engine cooling.

wrong, Güntner initially carried out a precise assessment of the initial situation on site and recorded temperatures at defined measurement points as part of a field test. At exactly the same measurement point, changes in air inlet temperatures under the coolers were recorded of up to 10 K over the measurement time and differences at adjacent measurement points under a single fluid cooler of up to 15 K at the same point in time. However, the average ambient temperature in the measurement range changed only by a maximum 6 K within the measurement timeframe. This meant that the only explanation was that exhaust air that had already been warmed up was being aspirated, the origin of which needed to be ascertained using Computational Fluid Dynamics.

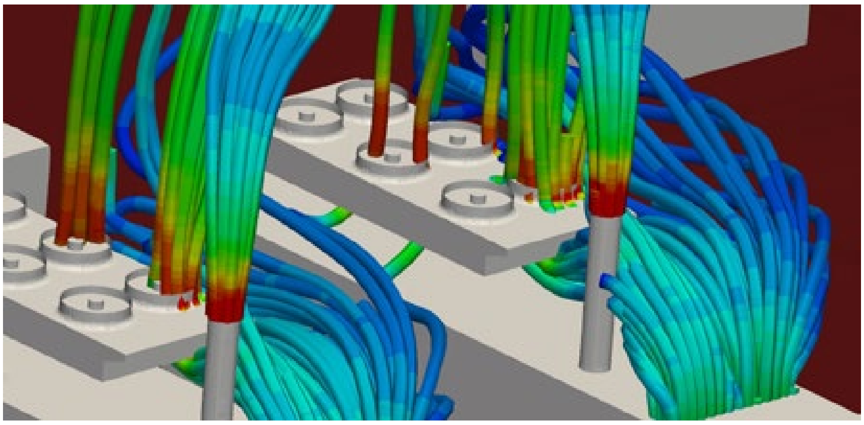
Based on the actual conditions at hand, Güntner developed a CFD model which took both the set-up of all the gas power plant's 100 generators and the design of the containers and air coolers as well as the exhaust gas routing and the weather data in the measurement timeframe into consideration. The CFD software calculated the relevant variables, such as pressure, temperature and air velocity for each of the approx. 36 million cells of the model. The findings were verified by comparing them to the measurement data collected in the field test.

## Visualisation of the initial situation

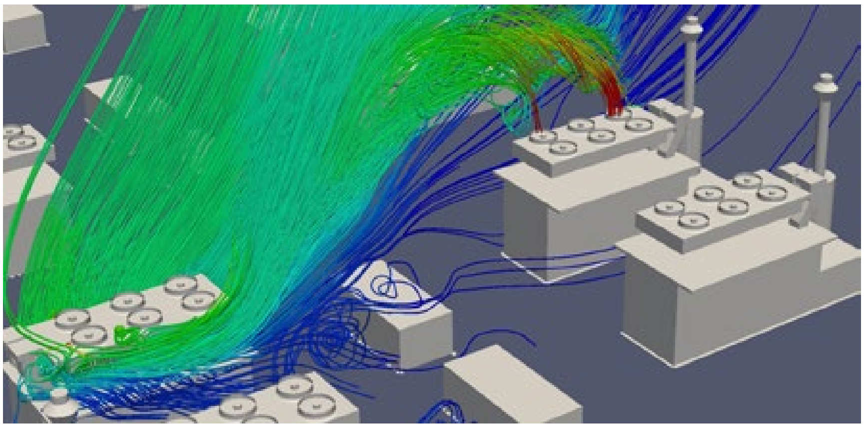
The initial graphical CFD assessment showed that to some extent the 60 °C exhaust air from the containers was being aspirated by the fans of the fluid cooler, which went some way to explaining the capacity losses. However, the approx. 430 °C engine exhaust gases from the flues were being aspirated, in some places, by the fans as well and were warming up the supply air.

What's more, exhaust gases and container exhaust air were gathering in the centre of the power plant, which led to an even greater rise in the supply air temperature for the internally positioned fluid coolers. To make matters worse, the exhaust gases of the external units were also warming up the supply air of the internal units, which altogether led to slumps in performance for almost all the internally positioned generators.

Following these findings, more simulations should firstly unlock potential for optimising the existing situation and secondly result in optimum set-up and design for upcoming similar projects. Several powerful servers were utilised over weeks for the corresponding calculations and several hundred gigabytes of data were generated as a result.



**Picture 2:**
Fault no. 1: The 60 °C exhaust air of the containers is partially aspirated by the fans.



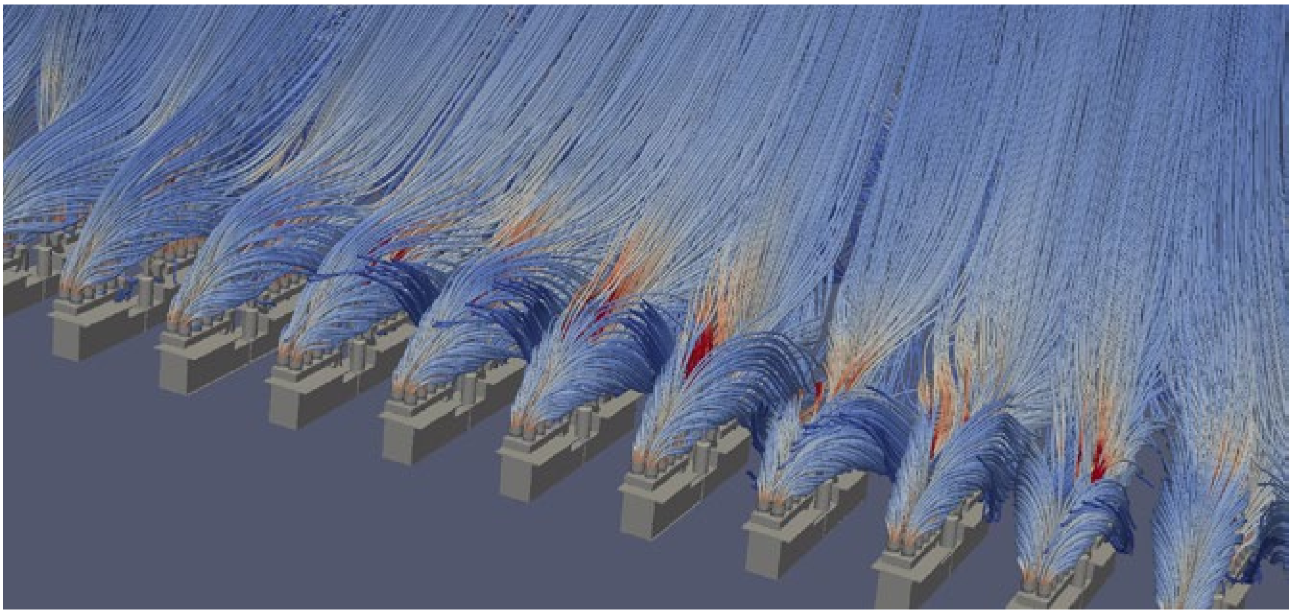
**Picture 3:**
Fault no. 2: The exhaust air of wind-facing generators warms up the supply air of wind-averted units.

## Design-related improvements to the existing situation – new set-up arrangements for subsequent project

By modifying only one parameter at a time in the database of the assessment of the initial situation and simulating the respective effects of this modification, it was possible to improve the efficiency of the existing power plant with design-related changes to the individual components. The measures included adapting the shape of the fan covers and adapting the engine exhaust gas routing.

A surface area with the same dimensions as those of the existing power plant was also examined for future power plants. In addition to the design-related improvements, as made to the existing plant, with the help of the CFD the arrangement of the generators and the design of the new containers were also optimised in order to prevent as many air short circuits as possible in the future. This very customised power plant design with the new arrangement of generators was again optimised in a final simulation step with weather data.

New power plants with the improved design will therefore also have considerably lower voltage fluctuations even in the event of temperature peaks and unfavourable wind conditions.



**Picture 4:**
An optimised arrangement and design achieved with the help of CFD will very effectively prevent air short circuits and increase engine cooling performance.