SIEMENS

www.siemens.com/datacenters

Using CFD for optimal thermal management and cooling design in data centers

Introduction

As the power density of IT equipment within a rack increases and energy costs rise, energy efficiency and optimal thermal management become increasingly important for data centers. Operational and energy costs can be reduced by implementing simple layout modifications, cooling optimization, or improved infrastructure planning. These can be done based on experience, key measurements on site, or based on accurate predictive tools like computational fluid dynamics (CFD) analysis.

This white paper focuses on how CFD can benefit data centers. It will help you understand how CFD helps you save energy costs and reduce risks by gaining knowledge from what-if simulations. It begins with an overview of CFD and describes how data center components can be represented using these numerical methods. This white paper also shows some advantages of using CFD for the analysis of data centers and how this can be used for optimal thermal management and efficient cooling design.

Defining CFD

Computational fluid dynamics (CFD) is a set of numerical methods and algorithms that are used to solve the equations governing fluid dynamics – the motion of fluids, such as air or water. It predicts the fluid velocity, temperature, and many other variables of interest in a wide variety of applications and domains, including, aeronautics, aerospace, energy, electronics and data centers. Fluid dynamics is often an integral part of product design, complementing computer-aided design (CAD) and computer-aided engineering (CAE) software packages for design and analysis. CAD is used to build virtual 3D models of objects and assemblies. These models are used as input for CAE, where stress analysis, heat transfer, or fluid dynamics can be simulated.

"Disrupted or inadequate airflow is a frequent culprit in data center cooling inefficiencies. But since you can't see airflow, it is a problem that is often overlooked. To truly optimize your data center, airflow and pressure monitoring is necessary to ensure there is enough plenum pressure to get the needed airflow to the racks. Inadequate cable management can also contribute to plenum airflow challenges. CFD analysis can be part of the solution." [1]

The myth of real-time CFD

While discussing the topic of CFD with many data center stakeholders, we often hear the term real-time CFD. CFD, as defined above, resolves millions of Navier-Stokes equations computationaly along with associated turbulence model equations, and that alone cannot be done in real time unless you are prepared to use hundreds of CPU cores to resolve all equations in parallel, and even then it is not possible to compute and refresh the display of the CFD results on millions of points in real time.

What most people refer to as real-time CFD is in fact real-time environmental monitoring. Real-time environmental monitoring shows on a CFD-like plot the temperature values read in real time from environmental sensors. Due to the few hundreds or thousands of sensor points, both the monitoring and displaying the values can be done in real time.

Both CFD and real-time environmental monitoring provide value to data center operators. CFD results provide the data center operators with forecasting and predictive simulation results to evaluate what-if scenarios before making any physical changes. The CFD method provides a virtual data center environment to try out various configurations and find the optimal configuration. It also helps in simulating failure such as computer room air conditioner (CRAC) unit failure, and properly estimate the time it will take for the thermal characteristic of the room to become risky to the IT or storage equipment. CFD also provides the operator with the best locations to place their real-time monitoring sensors to capture the critical hot spots. The real-time environmental monitoring on the other hand provides up-to-the-minute hot spot detection.

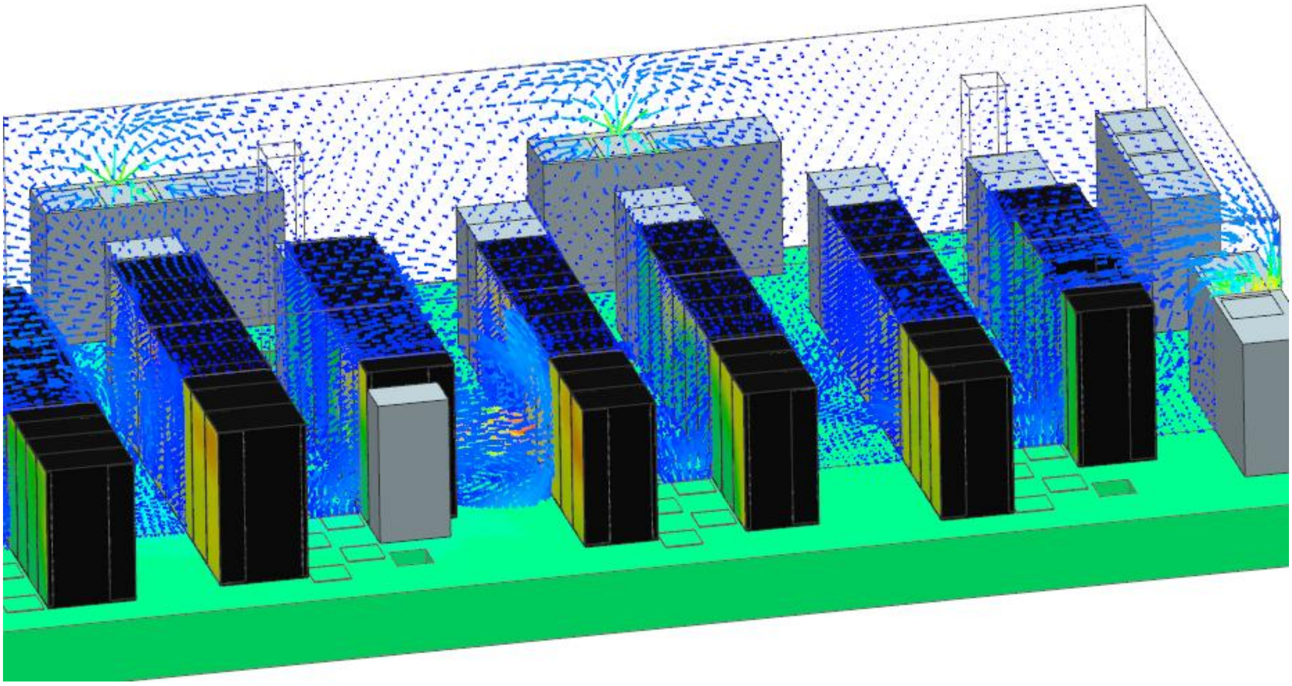
[1] DCIM Solutions: How do they do that? Gartner research note August 2014

CFD for data centers

CFD results for a data center

For data centers, CFD simulations predict the temperature, humidity level, and velocity of the air in the room, through the racks and through the servers. Figure 1 shows the air temperature distribution within the cabinets, as well as within the subfloor. The airflow in the main air room is represented with arrows. It shows the air flowing through floor tiles, entering and exiting the racks and servers, and entering the CRAC units located in the back and to the right.

Figure 1: Example of CFD results for a data center



Differences between CFD and real-time environmental monitoring

CFD and real-time environmental monitoring both provide data for temperatures or other parameters of interest for a data center. There are, however, significant differences between CFD and real-time monitoring (RTM). RTM provides specific parameters read by sensors or on assets being monitored in the data center. These parameters are measured physically at specific locations in the room. These can be stored, used to calculate other metrics, or linked to alarms.

CFD, on the other hand, provides predictions (rather than measurements) of the temperature and airflow distribution throughout the entire data center. It gives results even in locations that are not being monitored by the RTM system. As the next section will illustrate, CFD can be based on RTM measurements for a high level of accuracy.

In contrast to RTM that reports fixed measurements, CFD can be used as a flexible tool that predicts the impact of any possible change, modification, or event in the data center on the temperature and the cooling of the assets. CFD and real-time environmental monitoring are integral parts of a complete data center infrastructure management strategy.

Benefits of running a CFD analysis for a data center

Locate high temperatures and reduce the risk of equipment failure

It is well-known that high temperatures increase the risk of equipment failure. A CFD analysis of a data center can predict the locations of the highest temperatures, or hot spots. These high temperatures are the result of equipment power dissipations and insufficient cooling. The equipment located in these warm regions faces a higher risk of downtime than the other equipment in the data center.

CFD simulations can be used to find the best strategy by trying various configurations in a virtual way (without physically changing anything) to reduce these warm temperatures and mitigate the risk of equipment failure. A simple solution would consist of adding open floor tiles near the warmest assets, which increases the cooling capacity in this region. Figure 2 illustrates how CFD can predict the temperature reduction associated with adding open tiles near warm assets.

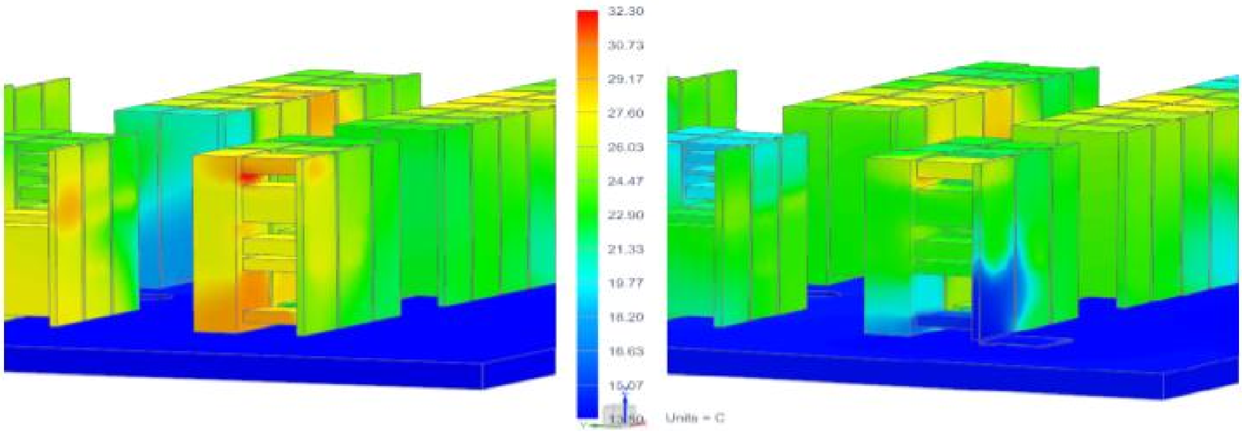


Figure 2: Temperature distribution in the cabinets for a given data center layout (left) and after addition of a tile (right)

CFD simulations can also suggest where to place or relocate high-power servers, by predicting regions with colder temperatures or higher cooling capacity. Figure 3 illustrates the temperature reduction associated with this modification for a given data center layout.

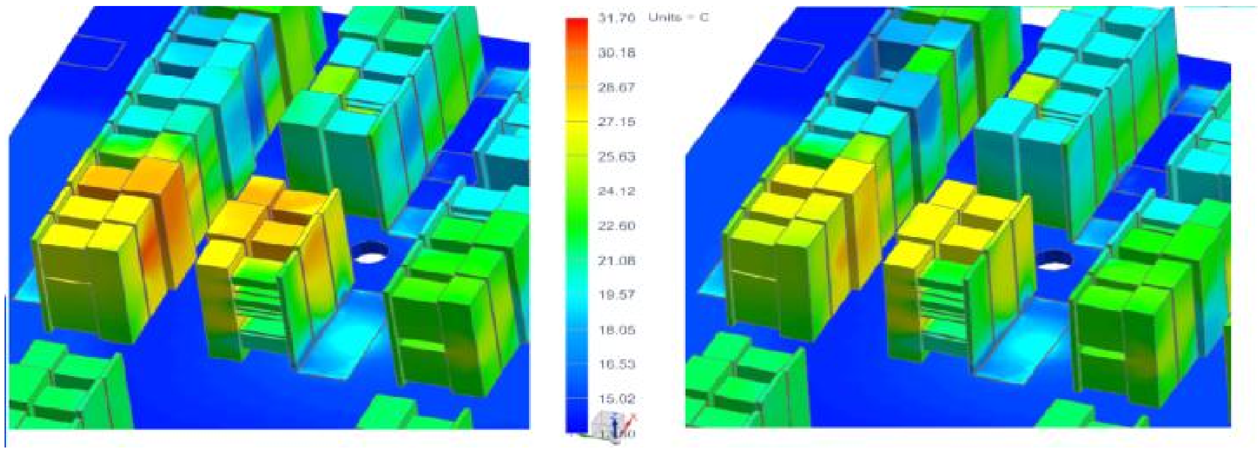


Figure 3: Temperature distribution in the cabinets for a given data center layout (left) and after relocation of two servers (right)

These are examples of low-cost modifications that can help increase equipment reliability. The optimal modifications can be identified by means of simulations prior to implementing them physically in the data center.

Find the optimal position of a new asset

Optimal positioning of new assets not only depends on space availability and power consumption, network availability, business grouping, but also on the local cooling capacity. CFD analysis can be fully integrated in data center asset management tools to predict this metric.

Figure 4 shows the Rack Cooling Index™ (RCI) for each rack in the Data Center Infrastructure Management (DCIM) solution from Siemens, Datacenter Clarity LC™ [2]. The RCI was computed using the predicted inlet temperatures of the assets from a CFD analysis. New, high-power assets should be placed in locations with high cooling capacity (i.e. racks in green). Over-cooled and under-cooled racks are easily identified by showing the RCI-HI index and RCI-LOW index.

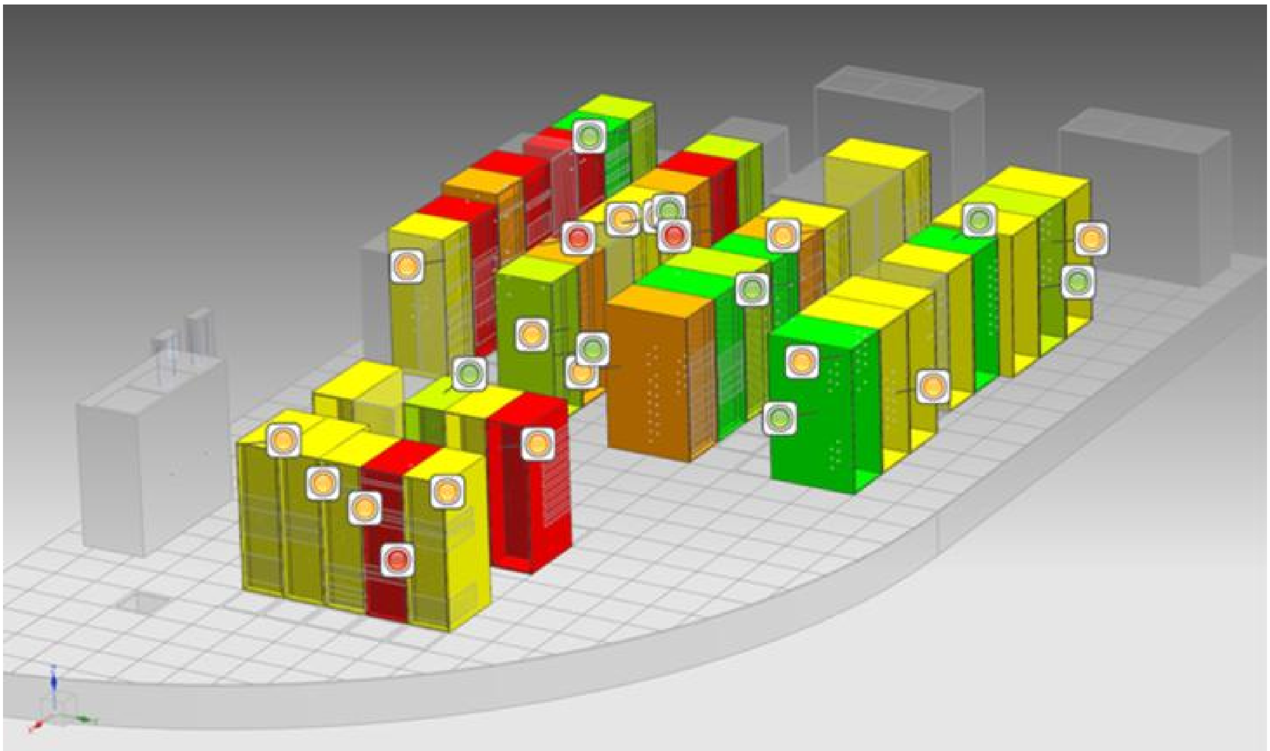


Figure 4: Rack-based cooling capacity (based on the asset inlet temperatures in a simple Datacenter Clarity LC model)

The impact of adding assets on the overall cooling capacity can subsequently be simulated to ensure sufficient cooling is still available and no additional hot spots are being introduced.

[2] Datacenter Clarity LC™ is a trademark owned and licensed by MAYA Heat Transfer Technologies Ltd.

Reduce costs by optimizing airflow

A high-efficiency cooling strategy can only be achieved with optimal airflow design. CFD simulations of a data center predict the airflow distribution and help identify inefficiencies in that airflow design. In the example shown in figure 5, the warm air exhausted by some servers is drawn in as cooling by the other servers, instead of what should be cold air from the subfloor. This is caused by an incomplete hot aisle. Adding racks to the row located between the hot outlet and the cold inlet of servers would be one of the possible solutions to mitigate this issue.

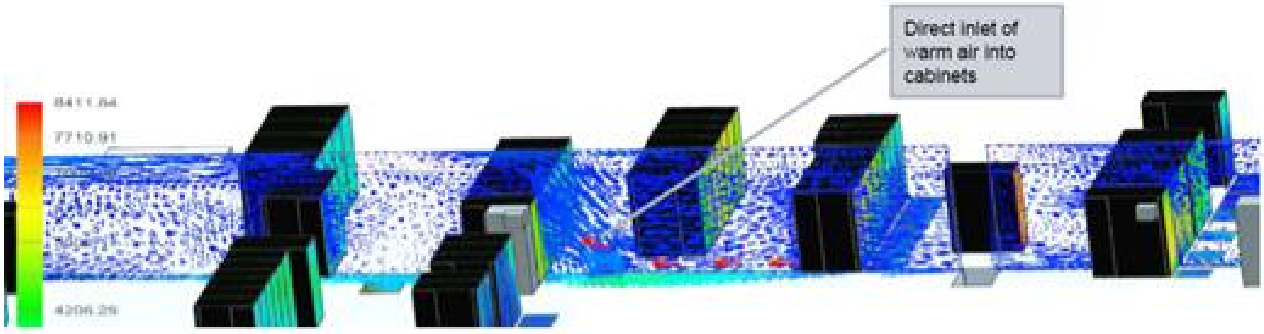


Figure 5: Inefficient airflow design in which warm air enters the intake of other servers

In figure 6, warm air recirculates from the top of the racks and reenters the same servers. In this case, a baffle at the top of the racks would be beneficial and would reduce the inlet temperatures of the servers.

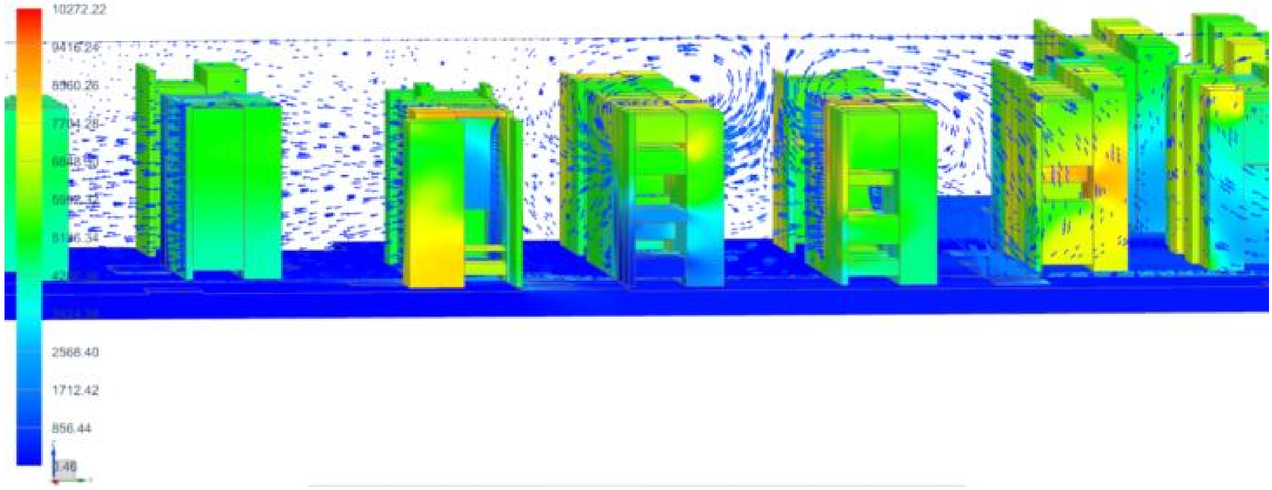


Figure 6: Inefficient airflow design in which warm air recirculates at the top of the racks

Airflow distribution can be optimized by, among other things, organizing the data center into hot and cold aisles, modifying the open floor tile positions, or isolating the cold aisles from the rest of the data center. Since high operational costs of making these changes in the data center can be incurred with these modifications, it is prudent to establish their cost effectiveness prior to implementation by means of simulations. The modifications can be virtually implemented in the CFD model, and the simulations can predict the associated server inlet temperature reductions and whether the temperature set point of the data center air conditioning units can be increased. These types of simulations are referred to as what-if analyses.

Reduce costs by optimizing operational conditions of cooling units

Since the inlet temperatures of the servers, as well as their power and airflow consumption, are typically unknown or guess-estimated from ideal manufacturer nameplate values, data centers often operate their cooling units by using the maximum or somehow derated manufacturer's cooling requirements. This results in overestimated cooling requirements, increases the electrical consumption and, hence the operational costs of the data center. The inlet temperatures of the servers, however, can be predicted with CFD; and, if linked with RTM, the model can account for the real physical power and airflow consumption of the servers. The data center can then be optimized virtually using CFD simulation and the accuracy of the results can be extremely accurate when real-time server data is collected and used as input to the CFD model. Optimization would balance maximizing the temperature set point of the cooling units, minimizing the fan rate of the cooling units, and minimizing the cooling equipment costs, all while keeping the inlet temperatures of the servers within their required range.

Prevent equipment failure by simulation

CFD analysis can be used to identify the most critical cooling equipment within a data center and establish the consequences of a potential failure in the cooling equipment. When a cooling equipment fails, temperatures start to increase with time. The failure of the most critical cooling equipment will result in the most severe temperature excursions, and these can be simulated using a CFD model. Such temperature increases challenge the long-term reliability of the IT equipment and should be avoided by deploying, for example, backup cooling units available in case of emergency. Dimensioning and positioning of the primary and backup cooling units can be based on CFD simulations.

CFD analysis can also be used to determine how long one has to react when a piece of cooling equipment has failed. For example, there may be multiple Computer Room Air Handlers (CRAHs) running at 60 F discharge. The failure of one CRAH may only allow 10 minutes to react before the temperature passes the critical threshold.

How to implement CFD for a data center

This section explains how a CFD model can be built to virtually simulate a data center.

Build the 3D model

The first step consists of building the 3D CAD geometry of a data center. The model must include all the key components and assets of the data center, such as the floor and floor tiling, subfloor plenum (if any), walls, pillars, racks, servers, Power Distribution Units (PDUs), and CRAC units.

Some geometric details, such as small gaps between racks or details in the walls, can be neglected for CFD analyses since these details are largely irrelevant to the overall airflow distribution. Figure 7 shows an example of 3D CAD geometry of a data center created by Datacenter Clarity LC.

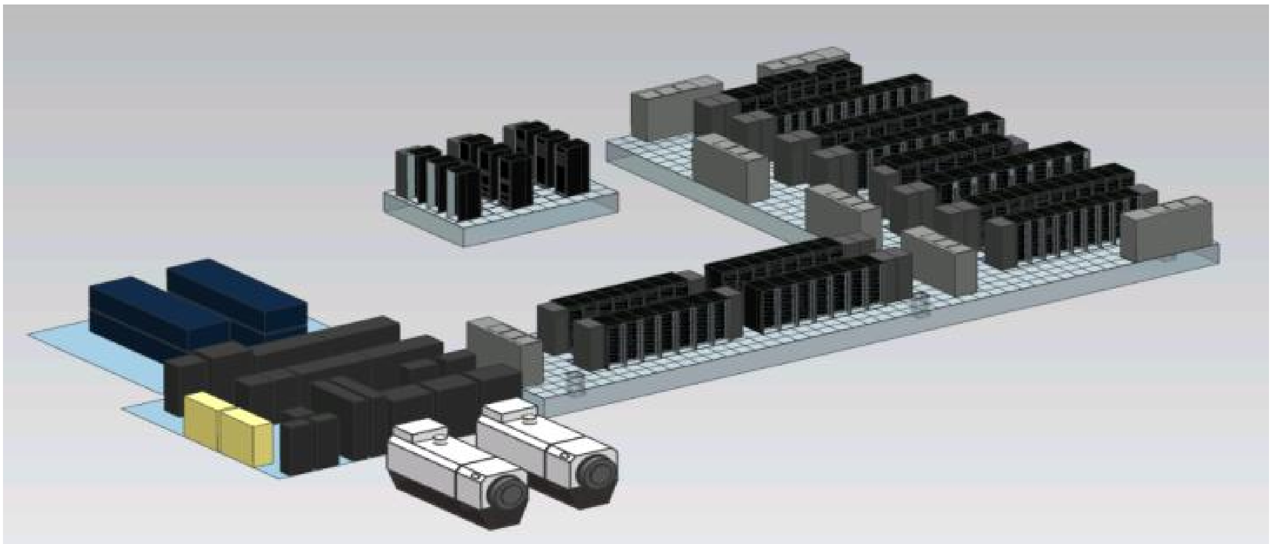


Figure 7: Example of a CAD model created by Datacenter Clarity LC

Mesh the air volume

CFD will solve conservation equations over small cells that fill the air volume. The air volume, including the subfloor plenum and the air flowing through the racks, must therefore be divided into many small cells defining the mesh. The mesh size will impact the solution accuracy. Robust meshing techniques must therefore be used to ensure an appropriate representation of the air in the data center. Fortunately, mesh generation is an automated process in modern CFD packages and can be further optimized in systems such as data centers (see figure 8 as an example).

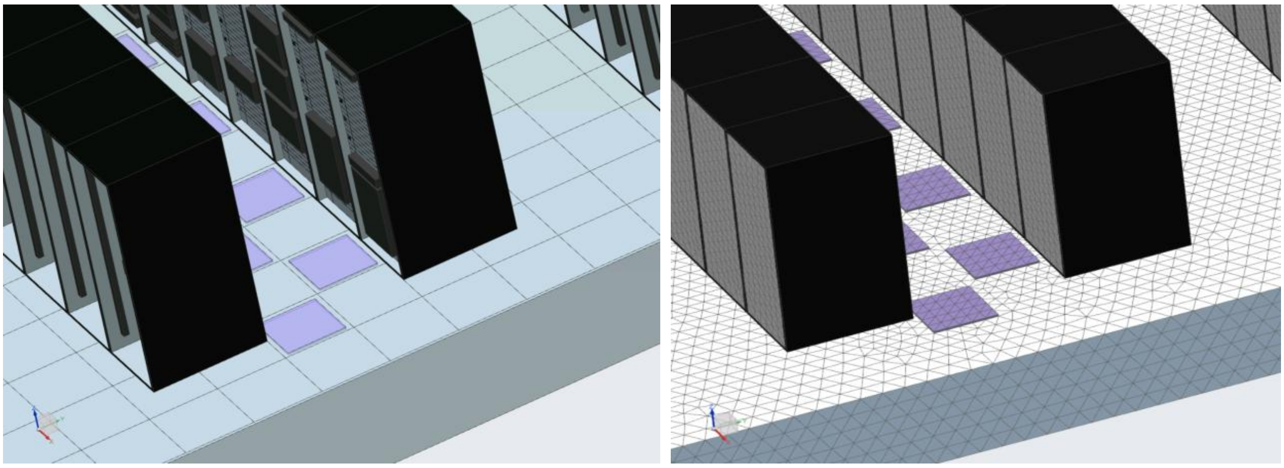


Figure 8: CAD model of a data center before (left) and after meshing (right) of the subfloor and the air in the cabinets

Include heat loads and heat sinks

The airflow and temperature distributions are the result of the heat (heat loads) generated by the servers or the other assets in the data center and the cool air provided by the cooling units (heat sinks). Loads are entered in the model as boundary conditions using one of the following methods:

- Using manufacturer data

  The most common approach consists of using the manufacturer maximum or derated powers and volumetric flows. This approach is suitable for design, where the real physical consumption of each asset is unknown, for example, simulations of future changes. However, in existing data centers, the server power consumption depends on its usage, and uncertainty can be introduced by using manufacturer data. It is prudent to validate the CFD simulation results with temperature and flow measurements captured in the data center and adjust heat loads as necessary.

- Using real-time data

  This is the recommended approach for existing data centers and involves using the real-time or time-averaged power consumption and airflow of each IT asset in the data center. This more reliable method is feasible if RTM of the assets is implemented as part of a DCIM strategy. It is advantageous and highly recommended to have a CFD model directly embedded into the DCIM tool, in order to directly use the monitored data as inputs.

Advantages of asset management and CFD under the same software

DCIM-embedded simulation enables engineers to conduct CFD analysis in the same user environment as the operation's environment. More important, any update on the 3D virtual data center model gets propagated to the CFD model (one central asset management system). Therefore, engineers always use the latest 3D virtual model for their CFD analysis to conduct multiple configurations and what-if studies and evaluate how the modifications will influence the performance. Hence, the CFD can be done in a fraction of the time it would take if you use a CFD tool which is not embedded within your DCIM solution. The main reason for this is you can get to your CFD result far faster without the hassle of data translation, 2D or 3D geometry clean-up, CFD model setup, 2D and 3D meshing setup, and reapplying boundary conditions and material properties every time you want to conduct a new CFD analysis of the current state of affairs. Historically CFD in data centers is not being conducted regularly due to the high costs associated with the constant regeneration of the CFD model. The advantages of CFD, as highlighted earlier in this paper and in many other publications, can be fully realized with a much lower cost to get the CFD results, and the much higher accuracy due to the centralized database and 3D virtual environment provided by Datacenter Clarity LC.

Conclusions

This white paper introduced computational fluid dynamics and described how it may be used to understand what occurs thermally inside a data center. CFD model accuracy is significantly improved when embedded in a DCIM tool and linked to RTM.

Some of the benefits of using CFD for data centers were presented. CFD simulations predict temperature and airflow distributions, which helps determine high temperature locations and airflow deficiencies in the data center design. CFD simulations can be used to study the cost effectiveness of possible modifications to the data center layout, to suggest ways to increase the cooling efficiency, or to find the optimal location of new assets.

Computational fluid dynamics is becoming increasingly important for data center infrastructure management and efficient cooling design. It should be used as a key tool in support of decision-making, troubleshooting, design, and optimization of data centers.

Siemens Industry, Inc.
Building Technologies Division
1000 Deerfield Parkway
Buffalo Grove, IL 60089
United States
Tel +1 847 215 1000

Siemens Switzerland Ltd
Building Technologies Division
International Headquarters
Gubelstrasse 22
6301 Zug
Switzerland
Tel +41 41 724 24 24

www.siemens.com/datacenters