

WHITE PAPER / CFD FOR AEC

5 WAYS CFD HELPS ARCHITECTS AND ENGINEERS IMPROVE BUILDING PERFORMANCE

This white paper investigates the application of **CFD simulation technology** in building design and the analysis of HVAC systems. The aim is to show its benefits to professionals working in the **AEC industry** who are interested in improving their skills and learning new tools for better and faster projects.

SIMSCALE



INTRODUCTION

Buildings are the human response to weather inclemencies. Be it the shining sun, heat, cold, wind gusts, rain, snow, dust or any other harmful elements of nature, our shelters allow us to rest, work or carry out almost any activity in comfort and safety.

With this functional goal in mind, architects and engineers have developed a wide variety of designs and technologies that, with time and testing, have been proven to meet the requirements of safe and comfortable buildings. From the first wooden huts, to our artificially-ventilated, earthquake-proof, high-rise buildings of today, the drive for continuous improvement has never stopped.

The advent of computer-aided engineering (CAE) in recent years has unlocked a set of powerful tools that allow engineers to develop building projects with more confidence, efficiency, and agility.
The virtual testing of designs for validation and exploration of concepts is a must-have in any innovative development project.
One of the most important entries in the category of **CAE tools** is computational fluid dynamics (**CFD**), which is a set of techniques that help to numerically **simulate physical phenomena** related to fluids and quantities that behave alike (such as heat).
With it, a given system can be studied and one can predict its behavior in areas such as flow paths, pressure drops, vortex formation, velocity distribution, energy losses, heat flow, and temperature distributions.

This is all completed in a virtual test environment, without the need for physical prototypes and complicated measurement rigs, allowing multiple design variations to be evaluated with maximum economy of resources.

01

VALIDATE HVAC SYSTEMS AND PARTS

Heating, ventilation and **air conditioning (HVAC)** systems are the principal means of guaranteeing **thermal comfort** in buildings, be it residential, commercial or industrial. For that reason, they are almost ubiquitous. Additionally, they are useful for contamination control in closed environments, such as clean rooms, underground parking structures, or subway stations. Here are some areas where the design engineer can **leverage CFD** to meet HVAC systems' performance goals:

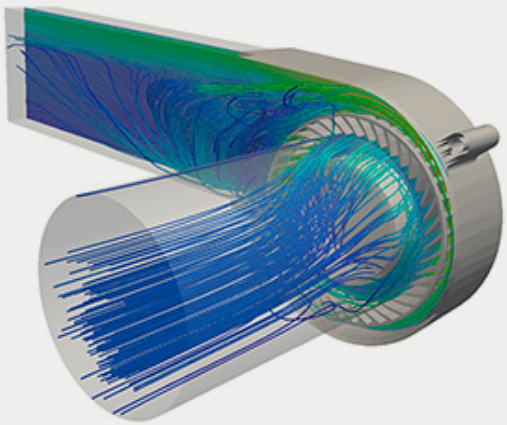
Predict performance of fans, compressors or pumps

Fans, compressors and pumps are used to generate the flow of liquids and gases. Although they have been in use for a very long time and there is a great amount of experience related to them, the designer often relies on manufacturer datasheets and curves to approximate flow rate and then try and test the integrated system to meet the performance requirements.

With CFD, the design engineer can test variations of impeller and casing designs, with realistic boundary conditions in concordance with the system where it will be performing.

When analyzed this way, the designers gain useful insights into the flow paths, pressure distributions and drops, which allows them to identify areas that need geometrical changes for performance improvement.

Changing the CAD model and re-running the simulation can be done in a matter of hours, allowing design engineers to test multiple potential designs.



CFD Analysis of a Centrifugal Fan with SimScale

Investigate flow, minimize pressure drops in ducts

Ducts sometimes seem like simple enough systems, but they are often routed according to handbooks and the experience of the designer, and then overlooked for the rest of the project. However, by virtually testing the ducts with CFD simulation, one can gain insights into the flow patterns and predict areas of high pressure drops, recirculation or vortex formation, all in operating conditions. The designer can then introduce optimizations such as longer span reductions, higher radius bends or baffles to enhance flow performance and reduce pressure drops.

Optimize heating and cooling equipment

Meeting performance targets in HVAC systems is very important, not only because of under-performance resulting in lack of comfort or safety, but also because over-performance represents wasted energy and increased capital and operational costs. With CFD, the designer can simulate the operating conditions and test for the adequate cooling or heating power required to meet the performance criteria.

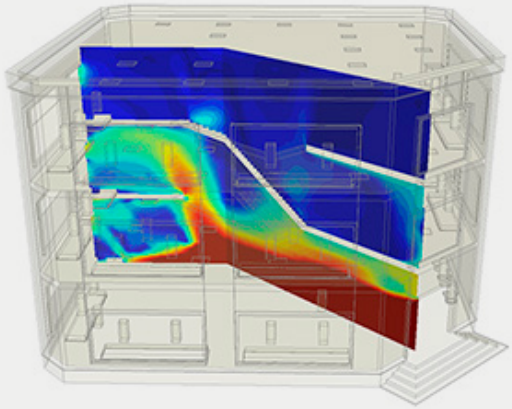
02

IMPROVE ENERGY EFFICIENCY

Energy efficiency in both residential and commercial buildings is of increasing importance as we become more focused on our **carbon footprint** and the effects of **global warming**. It is the responsibility of architects and engineers to take into account **energy consumption** and optimize the building performance from the conceptual design stage to ensure the best results are achieved. In this regard, the **virtual testing** is a useful, fast and economic tool to **compare and optimize designs**.

Ensure compliance with regulatory provisions

Entities such as ASHRAE (American Society of Heating, Refrigerating and Air-Conditioning Engineers) develop and publish specifications for the design and performance of HVAC systems and buildings [1] [2] [3].
Almost any building design project, whether it's a residential, commercial, industrial, or green building, is required to meet such a standard. Through virtual testing, the building's performance and code compliance can be guaranteed early in the design phase.



Airflow Analysis of Natural Ventilation in a Building with SimScale

Maximize flow performance in HVAC systems

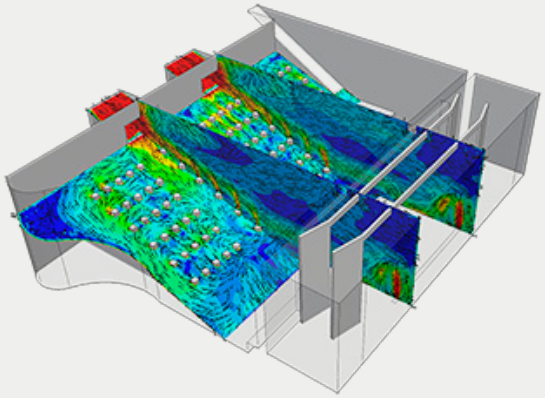
HVAC systems account for one of the biggest segments of energy consumption in both residential and commercial buildings [4] [5]. Thus, any improvement that can be made to their performance will have a big impact on the carbon footprint of a building.
One of the key areas for optimization is flow performance. By reducing pressure drops in ducts and optimizing inlet and outlet placement, the system can reduce the power requirement of fans, impacting directly the energy consumption (fans account for most of the energy consumption in HVAC systems).

Test ventilation placement to boost energy efficiency

Poor ventilation inlet/outlet placement strategies can have an impact on energy consumption if the space is not fully reached by the effect of the flow injection or suction, whether because of its large size or the presence of obstacles. Thus, high levels of reliability and capacity of the equipment are required in order to meet the variable target values for comfort. Through virtual testing, the designer can easily test multiple placements and choose the best performer for the particular conditions, ensuring optimal energy efficiency.

Improve insulation

By taking heat sources into account—both internal sources like electronic equipment and external sources like windows—the designer can visualize convective flows and temperature distributions, which allows them to accurately identify areas that do not comply with comfort requirements.
Based on this information, further iterations on the design can be carried out to test different insulation alternatives and assess their performance, before finally arriving on an optimal configuration (or a few candidates) worthy of being physically tested.



CFD Analysis of Thermal Comfort in a Theater with SimScale

03

PREDICT WIND LOADS ON BUILDINGS

High-speed wind loads on structures can be as devastating as earthquake loads. For this reason, **building construction codes** dedicate whole chapters to **wind load estimation** and include wind effects in several load combinations. Unfortunately, estimated loads are good (conservative) approximations only for regular, closed, box-like geometries, but for shapes that deviate from the basics tackled in the codes, loads must be estimated by **wind tunnel testing** or **computer simulation**.

Another factor that is simplified in construction codes is the effect of the surroundings; wind speeds can be both amplified or reduced by the presence of other buildings and bodies such as mountains, which may lead both to overestimation or sub-estimation of load levels. Codes will always be conservative, so if structural optimization is a requirement of the building project, estimation of realistic flow conditions must be carried out by wind tunnel wind-tunnel testing or computer simulation.

In comparison to wind tunnel testing, CFD offers many advantages:

- Reduced costs because physical prototyping is required in the former, and sometimes for multiple designs. Leasing of wind tunnels is rather expensive.
- More insight into the flow patterns and pressure distributions, as physical testing only offers data for specific measurement points.
- Quicker design iterations, as computer simulation is available 24/7 and model development through CAD is faster than physical prototyping.

Predict dynamic wind loads on high-rise buildings

If the architecture of the project includes factors such as non-conventional shaping, then wind load calculation methods from construction codes might not be valid—especially if the precise effects of the surroundings are to be taken into account. Computer simulation with CFD and related methods offer multiple tools to predict the loads on a building by taking into account effects such as:

- Turbulence and vortex formations
- Disturbance in the flow due to surrounding objects
- Taking into account building flexibility and deformations (fluid-structure interaction)
- Simulation of dynamical behavior to predict flutter frequencies

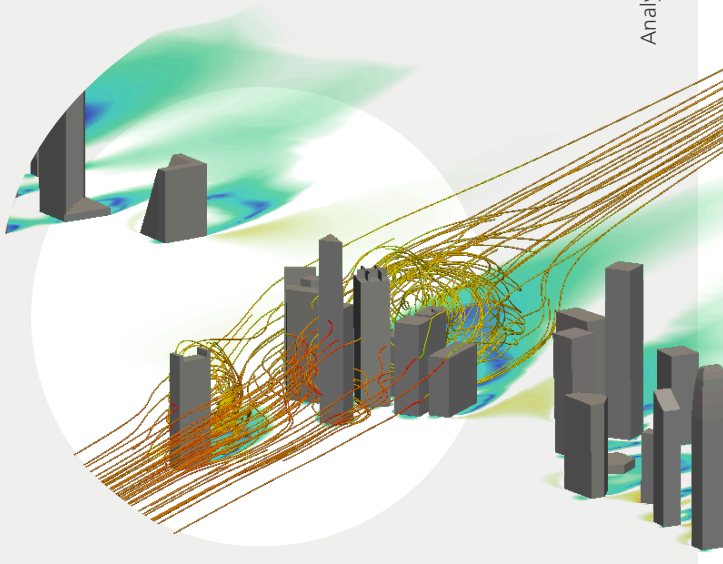
Test for bending and twisting loads on structures

Non-symmetrical architecture results in pressure differences that produce twisting loads over the structure.
Moreover, the presence of close buildings, such as in twin towers, also generates pressure differences that result in bending and twisting loads. These effects, in addition to the effect of the main lateral load, conjugate to produce rather complicated, asymmetrical and sometimes unexpected load distributions on the structural system.
Failing to correctly estimate these effects could result in unsafe conditions.
Through virtual testing with CFD, the structural engineer can be confident that all wind-induced loads and effects are taken into account and correctly estimated.

Test architectural designs and evaluate airflow over the constructions

By leveraging computer simulation, architects can gain more information to iterate on architectural designs. Both beneficial and unwanted effects can be predicted, such as:

- Using wind flow as leverage for a building's ventilation
- Predicting areas of vorticities and wind acceleration that could affect pedestrian comfort
- Test screening and wind protection for pedestrians
- Reducing or equilibrating the overall wind loads over the structure



Analysis of Wind Load on Buildings and Pedestrian Comfort in Singapore with SimScale

04

OPTIMIZE THERMAL COMFORT

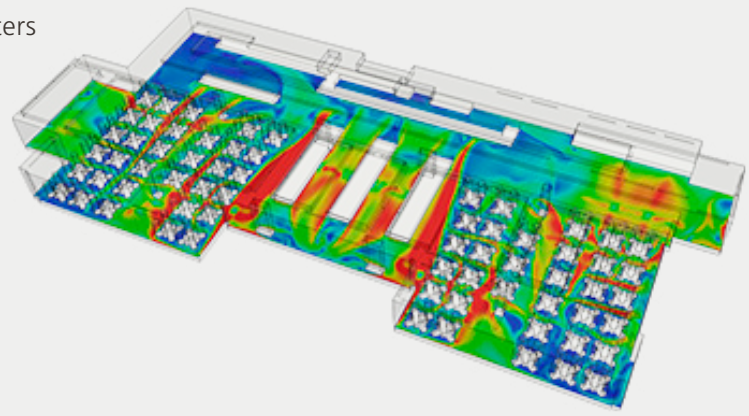
In architecture, creating an environment that is satisfactory for humans to be in is crucial. Many factors have an influence on perceived comfort, such as human metabolic rate, clothing insulation, air temperature, radiant temperature, air speed and humidity [3].
Some of these variables can be artificially conditioned to **meet target values**, both by the use of **HVAC equipment** or by **natural ventilation** strategies.
Here are some ways a designer can leverage **CDF for thermal comfort**:

Visualize airflow and heat transfer, evaluate temperature gradients, air distribution or velocity plots

With a CFD simulation, the designer gains a deep insight into the state of the room to be conditioned. For example, the designer can look at the speed and paths of air flow and identify vortex formation, recirculation areas or circulation around obstacles.
They can also generate time-temperature variation maps for the whole space or within specific objects. In this way, areas that do not meet the comfort parameters can be identified.

Test different air supply outlets and inlets

By making variations in air-supply flow rate and placement, the designer can tackle identified areas that do not meet comfort requirements.
With virtual testing of the different alternatives, the designer can quickly investigate and find the optimal design to meet the criteria.



Analysis of Airflow in a Restaurant Carried Out with SimScale

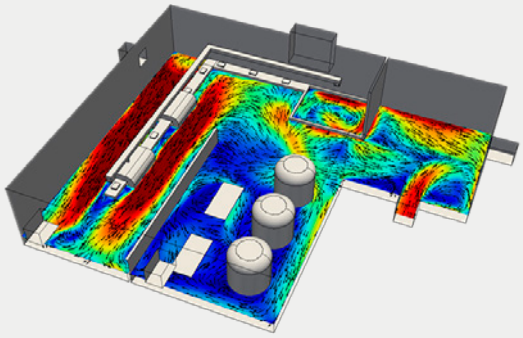
05

CONTROL AIR QUALITY AND CONTAMINATION

Air quality and contamination can be difficult problems to tackle.
Achieving precise performance through **HVAC systems** is critical for both meeting **quality criteria** and keeping costs down. **Passive scalar transport analysis** allows the designer to simulate and **visualize contaminant production** and movement with the flow.
By leveraging **CFD technology**, the designer can improve the design of air control and contamination systems.

Create a smart ventilation strategy

With virtual testing, the designer can gain a deeper insight into the behavior of the ventilation strategy and test different flow rates and inlet/outlet placement, so optimal performance and requirements compliance can be achieved.



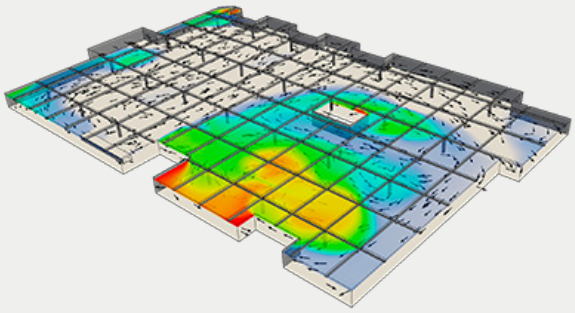
CFD Analysis of Toxic Fumes in an Industrial Building with SimScale

Ensure efficient removal of high contaminant concentrations in cleanrooms, labs or industrial buildings

The presence of obstacles and poor ventilation strategies can affect the performance of the air purification system in laboratories or cleanrooms, thus jeopardizing the safety of occupants and the process involved.
With virtual testing, the contaminant production, spread and removal can be analyzed, with precise measurements of concentration and flow variables.
Using this data, the ventilation strategy can be optimized to meet safe contaminant concentration levels.

Predict smoke propagation in underground garages, subway stations or shopping centers

Smoke production from combustion vehicles or fires in confined spaces can quickly create harmful contamination levels.
With the same techniques explained above, the designer can look at smoke propagation patterns and test variations in ventilation strategies, evacuation shafts and paths to maximize the safety of occupants.



CFD SIMULATION SOFTWARE

Until this point, CFD has been treated as a generic technology. No specific implementation or tool has been highlighted that a designer or analyst can use to leverage all the mentioned benefits in an actual project. When considering **investing in a CFD software**, many factors must be taken into account, but two of them will determine the ability to **perform successful workflows and results**:

1. Computing Power

CFD simulations are very demanding on resources. This means you will normally need machines with many processing cores and large memory to tackle decent-sized problems with reasonable accuracy and practical computing times. Hardware capital and operational costs can represent a large portion of the expenditures involved in implementing CFD in a project.

2. Capabilities vs. Ease of Use

Well-established, top-of-the-line CFD packages can tackle numerous types of problems, but they are not straightforward or easy to use for non-experienced simulation engineers and could turn out to be prohibitive for AEC designers and engineers. On the other hand, simpler simulation packages—such as those included in CAD suites—might be more accessible but often lack the capabilities or power to perform the required analysis for the applications stated in this article.

Cloud-Based CFD Tools

One very interesting, fresh approach to **CFD tools** is the **SimScale platform** (the examples in this article are taken from it). SimScale offers **cloud-computing power** to run the simulations remotely, which means the computing power and hardware maintenance issue is taken care of. It also offers a very approachable **user interface** that is deployed in the **web browser**, a **community of engineers** ready to assist, and a **library of public reference projects** that can be examined, copied, modified and used for other applications. In this way, the barriers in using CFD technology are removed..

Conclusion

Virtual testing of building design projects through **computational fluid dynamics** (CFD) simulation offers key advantages and insights that can be leveraged in areas related to **fluid flows**, such as **heating**, **ventilation** and **air conditioning**, **thermal comfort**, **safety against wind loads**, **contamination control** and **air quality.**

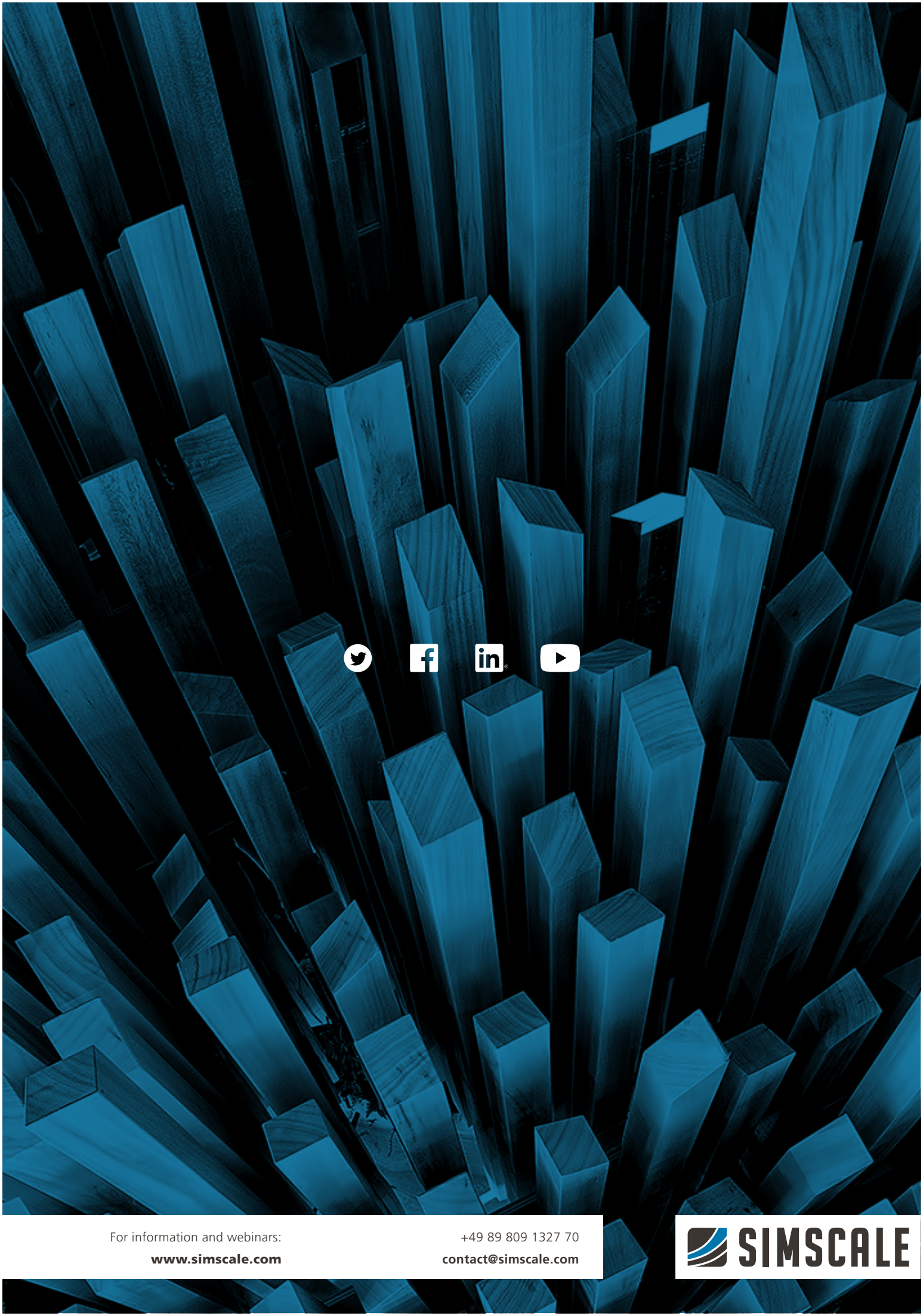
By using CFD simulation, the designer can tackle related issues from the conceptual design phase, thus **reducing unexpected deficiencies** that sometimes only become evident when construction is completed and tested. Safety and comfort **criteria requirements can be securely met**.

References

1. US Energy Information Administration, Residential Energy Consumption Survey, 2012
2. US Energy Information Administration, Commercial Energy Consumption Survey, 2012
3. AHSRAE Standard 55
4. AHSRAE Standard 61.1
5. AHSRAE Standard 61.2

Author: Guillermo Giraldo

Specialist in computer numerical simulation, Guillermo Giraldo is a mechatronics and machinery engineer. He is the founder of the engineering services company Senid and a SimScale consulting partner.



For information and webinars:
www.simscale.com

+49 89 809 1327 70
contact@simscale.com

SIMSCALE