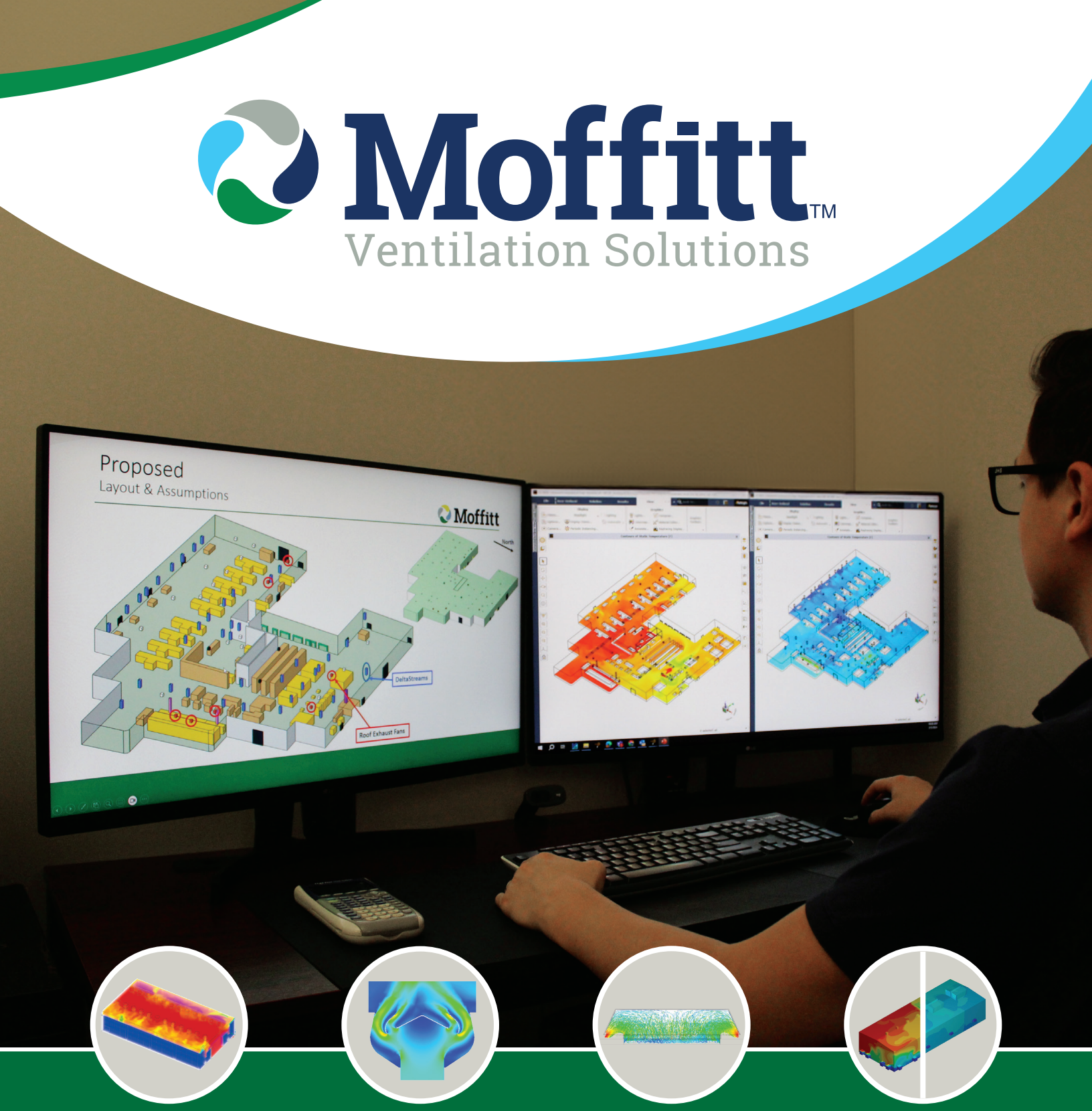
Moffitt™
Ventilation Solutions



A Computational Fluid Dynamics (CFD) ventilation model shows the flow of air and heat within a building. CFD modeling is especially useful for pinpointing temperatures at different locations and elevations throughout a facility. CFD modeling is used to design more powerful and more efficient ventilation systems.

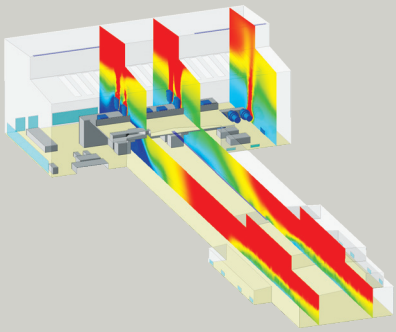
See our Project Profiles on the back to learn how CFD modeling can help you.

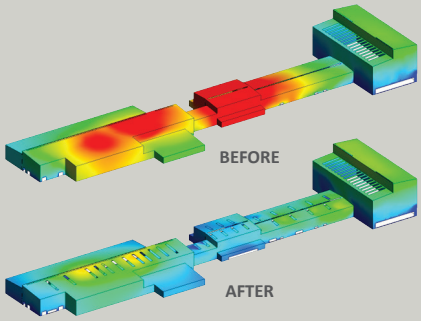
(800) 474-3267 | MoffittCorp.com | sales@moffittcorp.com

CFD MODELING PROCESS

1.	COLLECT DATA	Use customer provided drawings, satellite images, and (potential) site visit to collect data about the facility.
2.	MODEL BUILDING	Develop a model of the customer's building that simulates dimensions and spaces within the facility.
3.	DEFINE HEAT SOURCES	Approximate building temperatures using heat sources and existing ventilation equipment to match the internal heat load and ventilation rate.
4.	PRESENT MODEL	Present a summary of CFD results via e-mail and/or teleconference.
5.	REVISE	Additional model runs provided, (if needed, at cost).

PROJECT PROFILES





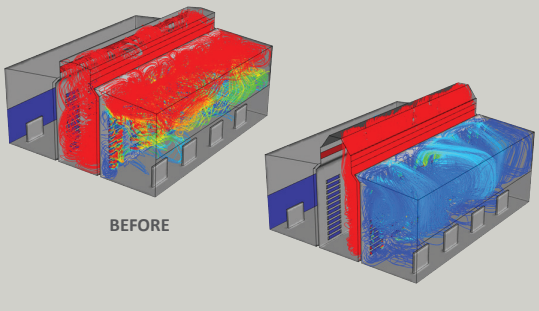
STEEL MILL

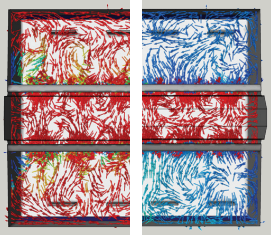
The temperature in the building with the old ventilation (BEFORE) is hot and stuffy (depicted in red). When the new solution is applied, (AFTER) the hot air vents and the building is cooler and more comfortable.

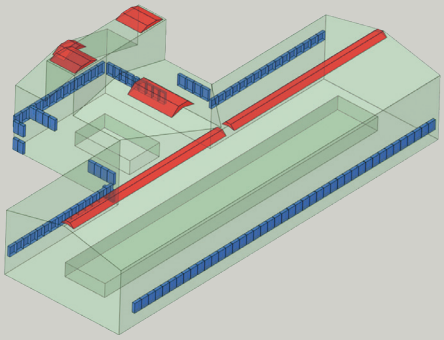
DATA CENTER

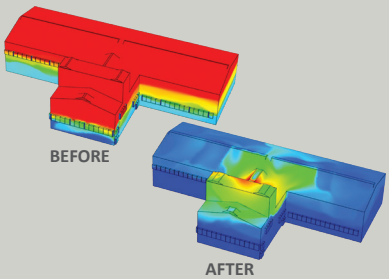
The left-hand models (BEFORE) depict warm air building-up around the computer servers, which can lead to overheating.

The right-hand models (AFTER) show the hot air exhausting away from the servers, allowing them to run at peak efficiency.









GLASS PLANT

The full building model, with the wall louvers (blue) and vents (red) is depicted on the left.

At right, the first model (AFTER) shows the built up of hot air at elevation. While the new solution (BEFORE) illustrates a cooler building, represented in blue.