

Computational Engineering Solutions LLP



CFD Simulations for Building & HVAC Industry

Singapore

Overview

Computational Engineering Solutions LLP, a leading Computational Fluid Dynamics solution provider, offers a niche expertise in the areas of Fire Simulation, Pollution Modeling, Natural Ventilation Simulation, Thermal Comfort Predictions and Water Treatment Solutions. Other enterprise offerings and solutions include Evacuation Modeling.

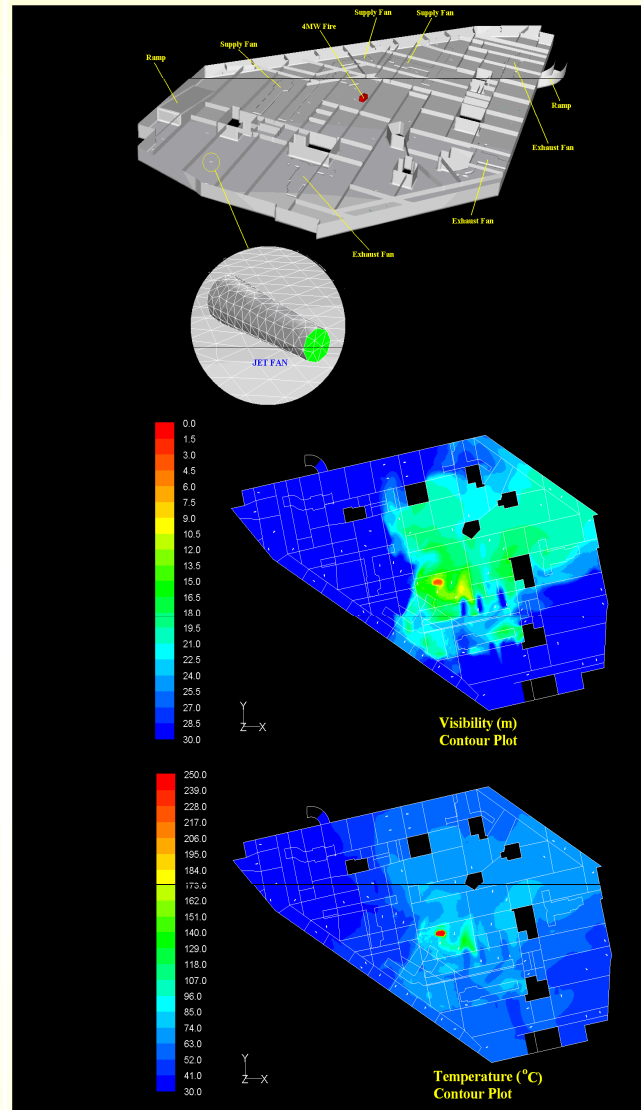
Incepted in 2007 with a team of competent CFD Engineers and complemented with an In-House Professional Engineer and Fire Safety Engineer, our extensive experience ensures first-rate engineering solutions for our clients, On Time Every Time. CES has a well established network of partners, clients and business relationships in Singapore and our clients are among the premier Architectural, HVAC and Water Treatment companies in Singapore. CES has set up a strong local presence in Singapore equipped with immense local knowledge.

A premium Computer Aided Engineering solution provider with strong domain knowledge, CES's expertise in the Fire Safety and Pollution Control is unmatched.

Testimonial

ANSYS FLUENT CFD package is an important tool in the optimization of ductless jet fan MV system design for car parks and other non-naturally ventilated enclosures. The use of Ductless Jet Fan MV system must be accompanied by rigorous CFD to substantiate that the prescribed level of safety and function provided by ducted designs are met. The accuracy and speed of the CFD simulations provided by ANSYS FLUENT makes it possible to exceed our clients' expectations.

Shyam Dayanandan
Operations Manager
Computational Engineering Solutions LLP



Challenge

The Fire and Engineering Codes prescribe many requirements to ensure the safety of occupants and facilitate fire fighting. Often times cost and structural limitations cause civil and M&E designs to deviate from these prescriptions. Assurance must be given to Fire Safety and Shelters Department that although there is deviation from the prescription in these specific instances, the alternative solutions proposed adequately meet safety requirements

Solution

The proper placement and orientation of Ductless Jet Fans and Main Fans are critical in ensuring the adequate management of smoke in the enclosure. The use of ANSYS FLUENT CFD package to simulate the fire, helps to analyze the spread of smoke and temperature within the enclosure. The outputs from the simulation is critical in optimizing the MV designs and ensuring the safety of occupants in the event of an emergency.

Benefits

The pre-processor offers a versatile platform to construct a detailed model of the carpark. The accuracy and speed of the simulation enable many design configurations to be analyzed. The clear and concise post processing offered makes it possible to predict areas of weak flow and stagnation, and further optimize the MV design.