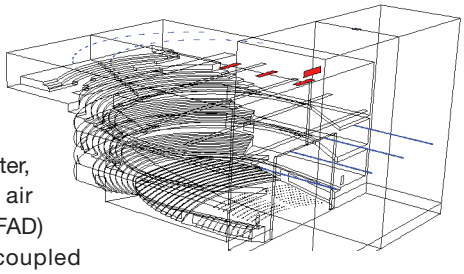
Meeting Green Building Design Goals with Engineering Simulation

Simulation is driving innovation in HVAC design for an assembly hall in Saudi Arabia.

By Sharbel Haber, Senior Mechanical Engineer, Balsam Nehme, Mechanical Engineering and Adnan Akhdar, Mechanical Engineer, Dar Al-Handasah, Beirut, Lebanon

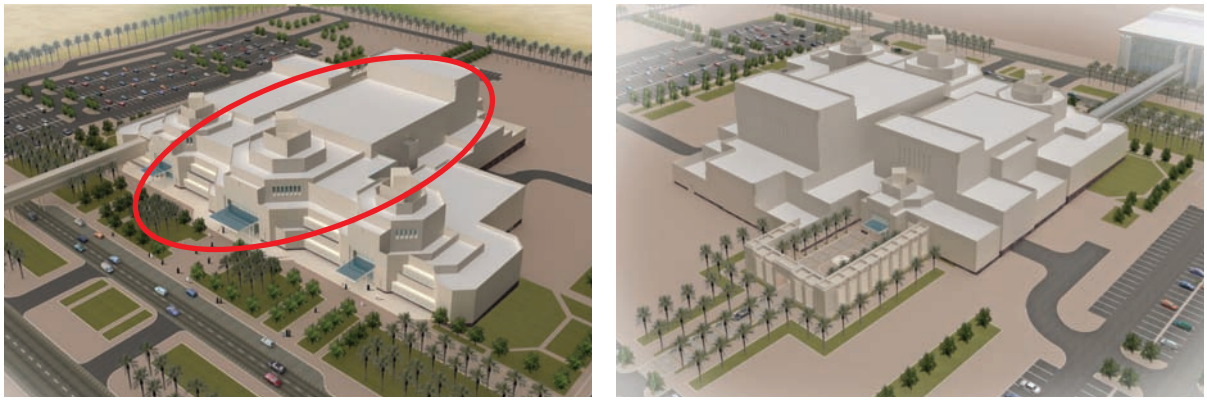
With the ever-increasing demands for sustainable buildings, engineers are developing more complex and diversified designs to reduce loads, boost efficiency and utilize renewable resources. Fluid dynamics simulations have proven to be a powerful and effective tool, providing flexible solutions in increasingly complex and demanding projects. At engineering firm Dar Al-Handasah, these simulations are extensively used as an optimization and validation tool at an early phase in the design process, since simulation supports implementation of innovative designs and energy-saving measures geared toward decreasing the overall facility's energy costs while maintaining or improving occupant comfort.

Dar Al-Handasah (Shair and Partners) has been a pioneering force in the planning, design and implementation of development projects in the Middle East, Africa and Asia since it was founded in 1956. Today, Dar Al-Handasah is one of the largest engineering and design firms in the world. A typical example of building for energy efficiency is the company's recent design for the 31,000-square-meter convention center at Princess Noura Bint AbdulRahman University for Women in Riyadh, Saudi Arabia.



Full-scale 3-D model of the convention center assembly hall

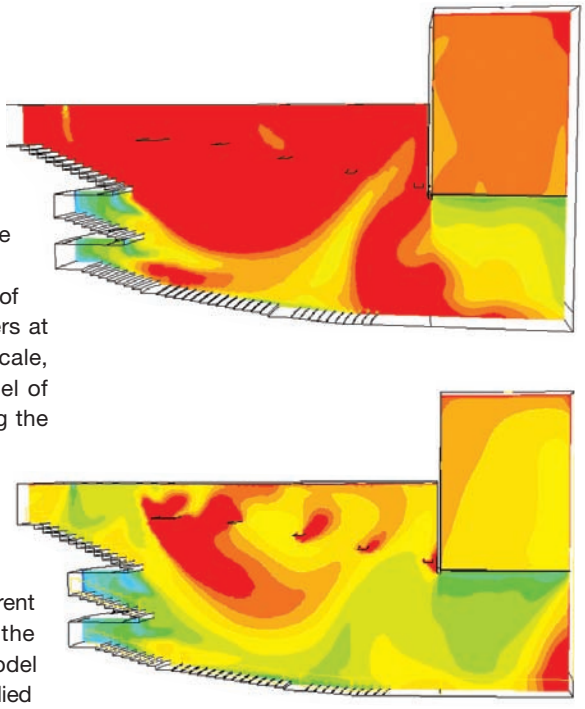
In the assembly hall of the convention center, an underfloor air distribution (UFAD) system was coupled with a conventional ceiling air supply. The assembly hall that includes the stage and seating area consists of four levels (basement, ground, mezzanine and first floor) interconnected via one air continuum, with all levels occupied. Dar Al-Handasah was charged with reducing the HVAC system energy consumption while ensuring audience comfort: The total airflow supplied by the hybrid cooling system should cool the occupied zones but only temper the upper regions where maintenance catwalks are located. The distribution of air supply outlets needed to be optimized to ensure proper air delivery in the 24-meter-high assembly hall while avoiding disturbance of the thermal stratification of the air, a key energy-saving measure. Fluid dynamics



Rendering of the convention center at Princess Noura Bint AbdulRahman University for Women in Riyadh. The center assembly hall was studied to reduce energy costs while maintaining comfort.

simulations with ANSYS FLUENT software were fundamental in optimizing the UFAD system to maximize energy savings and reduce energy costs.

To analyze the performance of the proposed system, engineers at Dar Al-Handasah built a full-scale, three-dimensional virtual model of the assembly hall, representing the complex geometry of the building. They accurately modeled 902 air diffusers beneath the audience seats in the hall and 62 ceiling flow bars distributed throughout the different levels. For greater accuracy, the team then meshed the 3-D model with local mesh refinements applied in the occupied zones and near the air outlets. Engineers conducted steady-state simulations using the ANSYS FLUENT solver to optimize the performance of the proposed hybrid air conditioning system to produce an environment that complies with the required comfort conditions. In particular, velocity and temperature distributions were generated and air distribution refined so that no disturbances occurred in the hot stratified region.

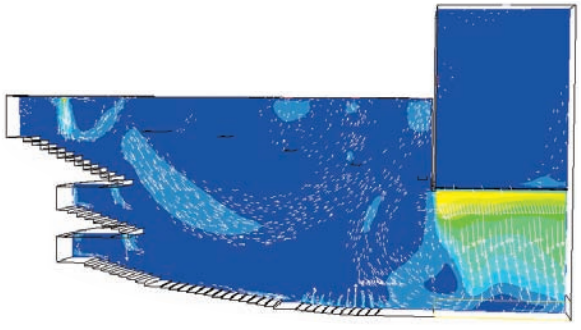


Original design scenario (top) and optimized design model (bottom) for temperature distribution across the convention center assembly hall

Initially, the flow simulation indicated a potential for improving the temperature distribution at various levels. In fact, high temperatures were observed at the first-floor-level seating area (around 31 degrees C) and at the stage area (around 25 degrees C), implying insufficient supply air flow to these zones. The team observed very high temperature (around 40 degrees C) in the core volume mainly due to high heat loads from the equipment on the catwalk. In addition, the mezzanine-level seating area was overcooled, with average temperature around 21 degrees C, indicating that the supply air flow delivered to that zone could be reduced.

Using the results from the initial fluid dynamics simulation, the engineers were able to visualize the airflow behavior inside the high-ceilinged hall and devise improved airflow delivery parameters, which were validated in a second simulation. Specifically, the supply air flow in the first-floor seating area was increased and directed at predefined angles, leading to better air delivery and accordingly lower temperature. Additionally, the team found the side flow bar diffusers at the balconies were inducing significant disturbances to the hot stratified air layer and, thus, engineers removed those diffusers from the design. Spot cooling was used to balance the high heat emitted by the catwalk's equipment so that disturbances would not be introduced into the upper hot stratified air layer. To capture the rising movement of buoyant hot air from the occupied zones, the team added air returns above the catwalks with an exhaust fan installed at the highest elevation of the stage. This fan also assisted in exhausting any contaminants.

Fluid flow simulations have become an instrumental tool in supporting the company's design process through accurate prediction of thermal comfort conditions, design validation enabling design optimization, and energy reduction for the HVAC building systems. Furthermore, the proven breadth and depth of advanced fluid dynamics modeling capabilities from ANSYS have allowed the company to tackle a wide range of complex problems ranging from HVAC and smoke simulations to dispersion modeling and pumping stations simulations. Dar Al-Handasah, with the support of ANSYS channel partner Fluid Codes Ltd for the Middle East, continues to explore opportunities to incorporate state-of-the-art tools in building design to continuously improve design quality while exceeding client expectations. Fluid dynamics software from ANSYS helps engineers to optimize HVAC designs and meet the ongoing challenge of developing models that are energy efficient, sustainable and compliant with standards.



Optimized design model showing flow pattern and velocity distribution across the assembly hall. Simulation assisted the designers in meeting energy efficiency goals.