A WHITE PAPER FROM ANSYS, INC.

Innovation in Building Design through Engineering Simulation

Walter Schwarz & Thierry Marchal, ANSYS, Inc.

INTRODUCTION

In today's world, engineers and architects increasingly employ engineering simulation to enable the design of atria, auditoriums, buildings, stadiums and sports arenas to meet energy-efficiency and sustainability goals within strict project timelines. Engineering simulation software is used to create a virtual prototype of a building or interior space on a computer, and calculate the heating, cooling and ventilation performance. This virtual building design approach leads to the rapid investigation of alternative designs and a better understanding of the design elements that can improve building performance, and allows the design team to more easily explore innovative solutions while reducing their exposure to risk.

Engineering simulation helps architects and engineers determine the best designs for heating and air conditioning systems, to analyze the performance of smoke management systems in the event of a fire and to ensure that the occupants are exposed to a desired level of thermal and moisture comfort. Building ventilation system designs that deliver indoor air quality, thermal comfort and energy efficiency can be better achieved through virtual building design.

In this paper, several projects are described to illustrate how engineering simulation software from ANSYS, Inc. has been used to achieve innovative green building and ventilation system designs.

VENTILATION DESIGN FOR MULTI-USE FACILITIES[1]

Several year-round indoor ice hockey arenas were recently designed in Russia. Engineering simulations of the heating, ventilation and air conditioning (HVAC) systems were carried out by Olof Granlund Oy, Finland's leading building services consulting firm. Hodynka Arena, in Moscow, Russia, was the main venue for the International Ice Hockey Federation (IIHF) World Championships in 2007 (Figure 1). This 62,000-square-meter arena had a capacity for 12,000 spectators during the championship finals. The indoor air conditions in the arena were maintained and delivered by a displacement ventilation system, which is well-suited to large, fully occupied stands. The Tsherepovets Arena, on the other hand, was designed for 6,000 spectators and the indoor air conditions were controlled by a mixing ventilation system. Because both arenas were also to be used for other events such as concerts, engineering simulation was used to better understand the interior conditions and airflows for a range of facility usage scenarios. The goal of these simulations was to determine how well the planned ventilation systems worked to deliver the desired indoor

Figure 1: The Hodynka Arena in Moscow, Russia

conditions, and to identify potential improvements such as the location and type of individual air supply diffusers.

Granlund frequently uses engineering simulation to investigate indoor air conditions in spaces where meeting the design requirements is a challenge Their typical methodology is to compare various HVAC system designs, air supply diffuser types and diffuser locations, all of which affect the indoor air quality of the facility. The optimal system design can then be chosen.

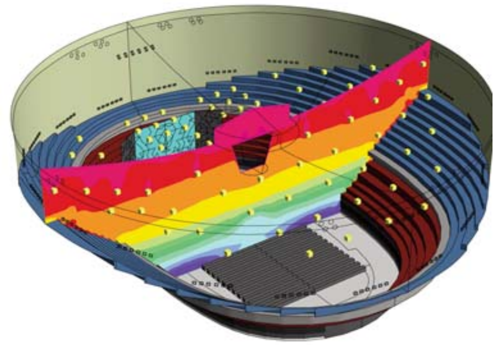

Figure 2. Temperature stratification during a concert event in the Hodynka Arena for summer conditions with displacement ventilation

For the ice hockey arena simulations shown in Figures 2 and 3, software from ANSYS was used to simulate and visualize the airflow in the sports arena. The first round of simulations typically performed for large-scale building projects such as these involve specifying the location and type of individual air supply diffusers and examining the resulting comfort conditions of the ventilated spaces. The performance of the diffusers in the simulations is ensured to match air jet theory and the velocity profile data and measurements supplied by the diffuser manufacturers. This step is important to realistically model the airflow behavior. These results give the design team insight to choose the best combination of diffuser types and locations.

The goals of the arena simulations were to incorporate the parameters that most affect the flow field and ensure that target thermal comfort conditions prevailed in zones that were heavily occupied by people at any given time. Given these requirements, the challenge was to design the supply air distribution so that adequate fresh airflows to fully occupied zones and improves the thermal conditions in the arena as a whole. Draft, humidity and temperature levels during different types of events in winter and summer conditions were considered.

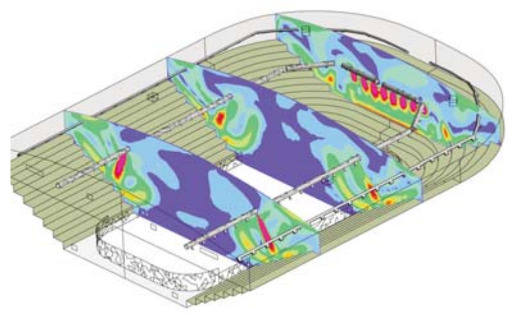

Figure 3. Velocity profile in the Tsherepovets Arena during an ice hockey game with mixing air flow from the roof zone

Supply air jets, forced and natural convection, and heat sources and sinks can cause complex three-dimensional airflow patterns. By using engineering simulation, building designers could quickly assess the performance of different airflow diffusers and supply air systems, such as mixing, displacement or a combination of both. First assumptions usually have to be revised several times before the target performance is reached.

SIMULATION OF SMOKE MANAGEMENT SYSTEMS IN ATRIUMS[2]

Atrium spaces are a popular means of creating a sense of openness and comfort for building occupants. The architecture of these spaces is becoming increasingly complicated as designers work to balance energy efficiency, aesthetics and visual impact. One of the challenges in designing such spaces, however, is engineering a smoke management system that can maintain tenable conditions in the space so that there is sufficient time for the occupants to escape in the event of a fire. The difficulties are a result of the interactions between the smoke, the architecture and the airflows. These interactions lead to disturbances in the rising smoke plume that cause excess mixing of the smoke with clean air, resulting in a larger volume of smoke to be exhausted. For example, overlapping levels or bridges across open spaces can lead to multiple balcony spill plumes, and architectural features can narrow the available flow area and cause local flow accelerations.

Authorities are now recognizing engineering simulations of fire scenarios as an important stage in the performance-based design cycle. The results are used to demonstrate that the smoke management system design preserves tenable conditions for occupants, as well as the structural integrity.

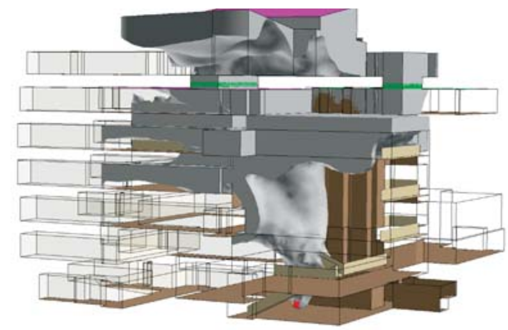

Figure 4. Smoke, represented by a gray iso-surface, penetrates into many occupied areas of a building with a complex atrium.

At RWDI, an internationally recognized engineering firm headquartered in Ontario, Canada, engineering simulation software from ANSYS has been used to better understand the air and smoke flows within complex atria in the presence of a fire (Figures 4 and 5). In one recent analysis, an atrium was studied that consisted of multiple levels and connected spaces. The space was outfitted with a smoke management system developed by following the local building code; the simulation results showed that with this system in place, smoke would penetrate into many of the occupied areas of the building. In RWDI's experience, providing a code-specified quantity of exhaust at the top of the atrium is not always sufficient for a safe atrium design. Other design strategies are necessary to help keep smoke out of the occupied zones, and RWDI uses a number of these to improve smoke management systems. For example, an atrium can be segregated into smaller and simpler atria when a fire erupts. Segregation in this particular atrium led to measurable reductions in undesired smoke propagation, used less than half the exhaust air and saved the owner both capital and operating costs.

By using virtual building models for the design of smoke management systems RWDI has developed a detailed understanding of smoke plume dynamics (including ceiling jets and thermal stratification), sprinklers, tenability (including visibility, toxicity and thermal exposure) and external wind effects. This has allowed RWDI to demonstrate successful ventilation designs, leading to safer and more cost-effective smoke management systems.

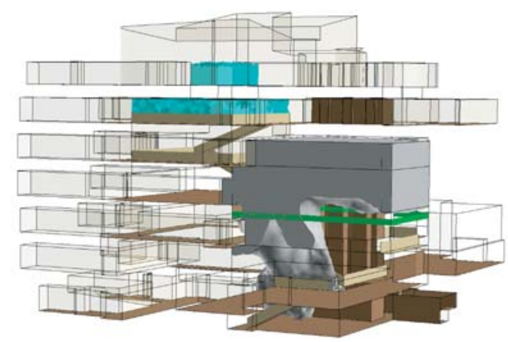

Figure 5. The same fire scenario as above, but with a smoke management system that segregates the atrium

Engineering simulation software from ANSYS provides a cost-effective and accurate means of designing efficient smoke management and detection systems. Simulations of fire scenarios and smoke propagation yield detailed information in an easily comprehensible form, which helps to optimize the position of detectors, fans, extractors, smoke screens, sprinklers and other firefighting systems and to plan emergency procedures.

SIMULATION IN SUPPORT OF ACHIEVING LEED RATING GOALS FOR AN OFFICE BUILDING[3]

In another interesting project, a multidisciplinary team of architects and engineers applied engineering simulation software from ANSYS to develop their award winning entry in the first annual U.S. Green Building Council (USGBC) Design Competition (Figure 6). The team from Chicago, Illinois-based OWP/P was the third-place winner in this competition for emerging green builders.

Entrants designed a theoretical addition to the existing facility for The Pittsburgh Project, an urban, neighborhood-based Christian community development organization serving the north side of Pittsburgh, Pennsylvania. Submissions were required to meet the stringent requirements needed to earn a Platinum rating under the USGBC's LEED® (Leadership in Energy and Environmental Design) green building rating system, while also meeting the aggressive target budget of $100 per square foot. To enhance occupant comfort, minimize the building's energy use and keep the project cost low, the team designed a naturally ventilated building with no mechanical ventilation or cooling.

The design concept relied on buoyancy-driven airflow developing within a solar-heated cavity created between the brick exterior of the existing building and a new, three-story plane of glass located at one end of the new building. This solar tower pulls air through the operable windows of each individual room, down the corridors and out of an exhaust stack at the top of the building. The thermal mass of the brick wall maintains warm temperatures in the solar tower through the night, driving cool night air through the building to remove heat built up during the day.

Figure 6. A schematic of the proposed addition to the existing facility for the Pittsburgh Project

The team used software from ANSYS to verify and develop the design concept. The model was developed from very early hand sketches and computer-based architectural models in order to confirm that sufficient ventilation would be provided by this design. Initial models showed that openings between the three levels outside of the solar tower allowed too much air to rise from floor to floor, bypassing the tower and creating uneven temperatures and air quality in the building. These openings were closed off from airflow with suspended pieces of glass, allowing light transmission and visual communication between levels. Analysis of the final design shows large amounts of air flowing in through all windows and out through the solar tower, in keeping with the design intent (Figure 7). Thus, engineering simulations during the early design stages allow concepts that might otherwise be considered unrealizable to be explored and tested quickly. Unviable concepts can be rejected, while promising concepts can be refined.

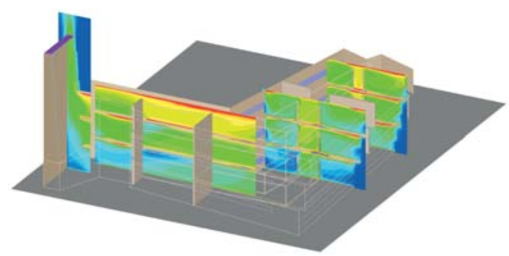

Figure 7. Temperature contours on selected planes in the proposed addition show the range of temperatures that can be expected to occur with natural ventilation.

SIMULATION IN SUPPORT OF ACHIEVING LEED RATING GOALS FOR A MUSEUM[4]

The William Jefferson Clinton Presidential Center is located on the south bank of the Arkansas River just east of downtown Little Rock, Arkansas, U.S.A. The main feature of the center is the Bridge Building. This building houses the Clinton Presidential Museum, which is a public exhibition gallery and museum space. Nearby is the Archive Building, which contains the National Archive and Records Administration facilities that include storage vaults and office space for researchers.

The museum space that is housed in the Bridge Building occupies a dramatic, double-height space that contains exhibits chronicling the tenure of the former president (Figure 8). The lower level consists of a series of permanent interactive exhibits. The upper level is open to the lower level in the middle with more exhibits around the perimeter of the space. The west wall of the space is a full-height glass wall, while the east wall is opaque.

Since many of the exhibits have large cooling loads due to specialty lighting and interactive display equipment, the design objective for the space was to create a stratified layer of air with conditions in the human thermal comfort range in the lower level. The heat would then rise through the middle of the upper level to a return at the ceiling. To accomplish this, several different methods of conditioning the space were utilized. On the lower level,

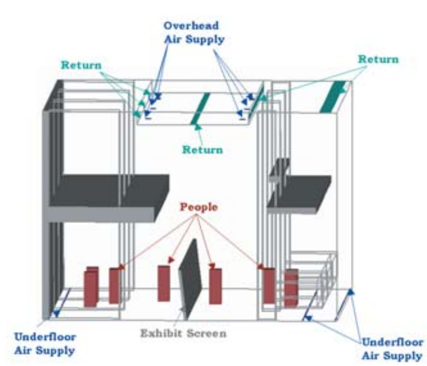

Figure 8. Geometry of the interior of the exhibit space housed in the Bridge Building at the William Jefferson Clinton Presidential Center

displacement air distribution was provided using linear slot diffusers along the west perimeter and in front of the exhibits on the east and west sides. This air was supplied in the occupied areas at design conditions while unoccupied areas along the perimeter were supplied at 55°F. On the upper level it was not possible to use underfloor distribution due to the structure of the second floor, so overhead air supplies were used. This air was supplied at 55°F from jet diffusers located over the open area. On both levels, a radiant floor was used to create a warm thermal mass in the winter and to help to absorb the space solar load in the summer.

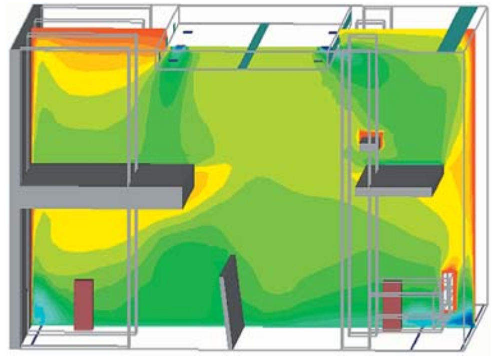
Figure 9. Temperature contours on a vertical plane

Since so many different systems were being used to condition the space, engineering simulations using software from ANSYS were performed by Flack & Kurtz Inc., New York, New York, U.S.A., to study the interaction of the various systems. Flack and Kurtz has successfully engineered and implemented many advanced green construction technologies in a variety of applications, using engineering tools that include computational fluid dynamics, advanced energy modeling and lighting analysis.

The simulation results showed that during the summer months, the space could be maintained at the desired thermal conditions (Figure 9). The displacement distribution created a layer of cooler air that fills the occupied area and forces the warmer air to rise up the middle. The upper-level overhead supply flow pushes across the catwalks and picks up heat given off by the exhibits before rising along the exhibit walls and circulating back to the return vents. Throughout the entire space, the airflow is assisted by the contributions of the radiant floor system. The engineering simulation was used to determine the optimum operating conditions for the airflow systems during regular occupancy, and to verify the ventilation effectiveness in support of a LEED credit.

SUMMARY

With more complex designs, stringent safety requirements and increasing energy concerns, designing or improving commercial buildings has become a critical engineering challenge. Comfort, safety and efficiency need to be guaranteed and able to stand up to intense scrutiny. Whatever the structure — office building, hotel, school, parking garage, mall, stadium, tunnel or concert hall — buildings must meet the rigorous aesthetic, social and environmental standards of the times. Offering a suite of powerful engineering simulation tools, ANSYS, Inc. empowers professionals to optimize building designs, especially in early stages when changes can be implemented efficiently and cost-effectively. Using these tools, architects and engineers can make informed decisions about the choice of building materials, system components and HVAC systems. Engineering simulation software from ANSYS assists in the process of developing and evaluating building ventilation system design concepts as well as contributing to the evidence that a proposed design meets various rating criteria.

REFERENCES

1. Scoring a HVAC Goal for Hockey Spectators by Sami Lestinen, Tuomas Laine and Tom L Sundman, Olof Granlund Oy, Finland, ANSYS Advantage, Vol. 1, Issue 1, 2007.
2. Containing Smoke in Complex Atria by Duncan Phillips and Ray Sinclair, Rowan Davies, & Irwin, Inc., Guelph, Ontario, CN, Fluent Newsletter, Spring 2003.
3. Shooting the Breeze in Pittsburgh by Erik Olsen, Andre Gilles, and Jessica Wojtas, OWP/P, Chicago, IL, Fluent Newsletter, Spring 2004.
4. Taking the Heat out of the Clinton Museum by Daniel Nall and Michael Eskra, Flack and Kurtz, New York, NY, Fluent Newsletter, Fall 2002.

About ANSYS, Inc.

ANSYS Inc., founded in 1970, develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries. The Company focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-effective product development, from design concept to final-stage testing, validation and production. The Company and its global network of channel partners provide sales, support and training for customers. Headquartered in Canonsburg, Pennsylvania, U.S.A., with more than 60 strategic sales locations throughout the world, ANSYS, Inc. and its subsidiaries employ approximately 1,600 people and distribute ANSYS products through a network of channel partners in over 40 countries. Visit http://www.ansys.com for more information.

www.ansys.com

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com

Toll Free U.S.A./Canada:
1.866.267.9724
Toll Free Mexico:
001.866.267.9724
Europe:
44.870.010.4456
eu.sales@ansys.com