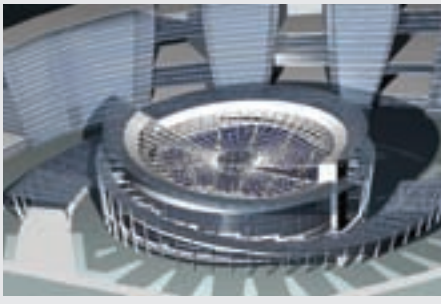
Scoring an HVAC Goal for **Hockey** Spectators

CFD is used to design ventilation systems for sports arenas.

By Sami Lestinen, Tuomas Laine and Tom L Sundman
Olof Granlund Oy, Finland

In today's world, the design of auditoriums, stadiums and sports arenas is not complete without one or more simulations of the interior air flow. Analyses of this type are used to determine the best heating and air conditioning systems, analyze smoke removal in the event of a fire and ensure that the occupants are exposed to a predefined thermal comfort range most or all of the time.

Several year-round indoor ice hockey arenas were recently designed in Russia. CFD simulations of the HVAC systems were carried out by Olof Granlund Oy, Finland's leading building services consulting firm. Hodynka Arena in Moscow is the main venue for the International Ice Hockey Federation (IIHF) World Championships, to be played on April 27 to May 13, 2007. This 62,000 square-meter arena has a capacity for 12,000 spectators during the championship finals. The indoor air conditions in the arena are based on a displacement ventilation system, which is well-suited to large, fully occupied stands.



The Hodynka Arena in Moscow; CFD was used for designing the indoor air conditions.

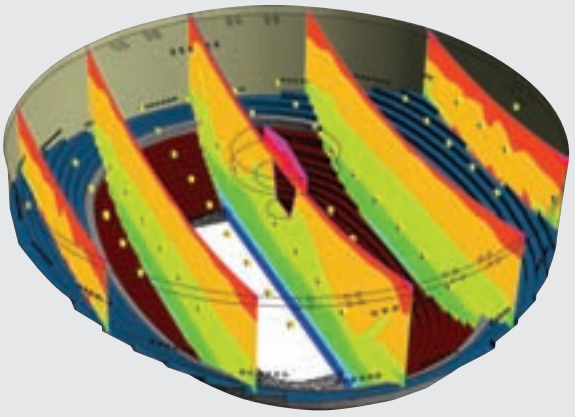
The Tsherepovets Arena, on the other hand, was designed for 6,000 spectators and the indoor air conditions are controlled by a mixing ventilation system. Because both arenas also will be used for other events, such as concerts, CFD was used to better understand the interior conditions and flow behavior for a range of usage scenarios. The goal of these simulations was to determine how well the planned ventilation systems work to meet the desired indoor conditions.

Granlund uses CFD to research indoor air conditions in spaces where design requirements are high and detailed flow field information is important. Their typical focus is to compare a number of HVAC systems, air flow outlets, construction methods and other sources, all of which affect the indoor air quality of the finished structure.

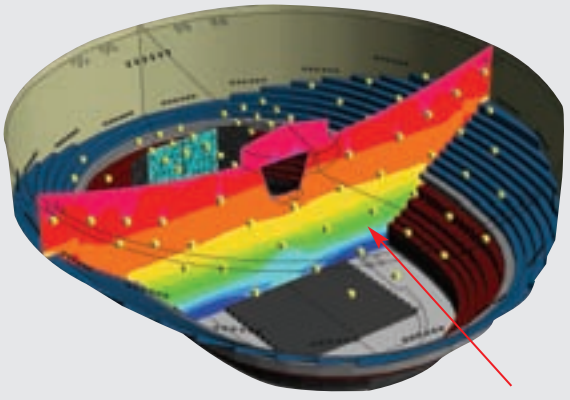
For the ice hockey arena simulations, ANSYS ICEM CFD software was used to build the model, and ANSYS CFX tools were used to simulate and visualize the flow. The first round of simulations that is typically performed for large scale building projects such as these involves individual air supply devices to test and compare operating conditions. The results allow the design team to choose the appropriate devices for each specific location.

The device simulation results are compared to air jet theory and to the manufacturer's profile data and measurements. This step is important if the air flow behavior is to be estimated realistically in models of the building as a whole, in which the simulated supply air jet profiles can be used as boundary conditions. This technique requires fewer calculation nodes in the large model, which saves simulation time. The device models and simulation results are saved to an object library for future projects.

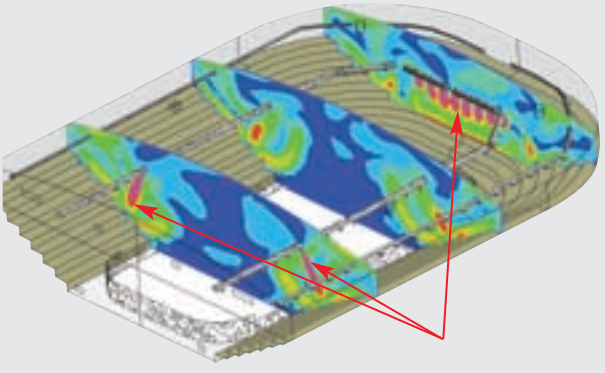
The goals of the arena simulations were to incorporate the parameters that most affect the flow field and ensure that target thermal conditions prevail in zones that are heavily occupied by people at any given time. Given these needs, the challenge was to design the supply air distribution so that fresh air flows 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. Supply air jets, forced and natural convection, and heat sources and sinks cause very complicated 3-D flow fields, so the simulations needed to be performed with care. This need was constrained by the accuracy of the initial data, the approximations used, the level of convergence and restrictions on the allowable simulation time. The benefits of the simulations are that they provide the possibility to try out different air flow device types or supply air systems, such as mixing, displacement or a combination of both. First assumptions usually have to be corrected one or more times before the target is reached. In the end, however, a correctly performed CFD simulation is the only calculation method that can capture the indoor air flow field with the accuracy necessary for design purposes. ■



Temperature contours in the Hodynka Arena, showing stratification on several planar slices, during an ice hockey game in summer conditions with displacement ventilation



Temperature stratification during a concert event in summer conditions with displacement ventilation; the arrow points to the area where cool supply air flows to the occupied zone on the field.



Velocity profile in the Tsherepovets Arena during an ice hockey game with mixing air flow from the roof zone; the arrows point to the areas where supply air jets flow to the occupied zones of the stands.