2018 Building Performance Analysis Conference and
SimBuild co-organized by ASHRAE and IBPSA-USA
Chicago, IL
September 26-28, 2018

INTEGRATING CFD WITH BEM IN EARLY DESIGN STAGE TO OPTIMIZE DESIGN SOLUTIONS

Sedighehsadat Mirianhosseinabadi[1], Mohit Mehta[1], and Jamy Bacchus[1]
[1]ME Engineers, Golden, CO, USA

ABSTRACT

Integrating Building Energy Modeling (BEM) and Computational Fluid Dynamic (CFD) early in the design process provides comprehensive information to design teams to select the optimum HVAC system in terms of thermal comfort. This paper is a review of previous CFD studies and also evaluates the thermal comfort inside a multipurpose arena bowl by using CFD analysis of the HVAC air-side system with different configurations using the IES Virtual Environment (IES VE) software. The challenges of this process and the software are addressed. Lastly, the integrated BEM is modified based on the optimized system solution and energy saving results over the baseline are presented.

INTRODUCTION

The integration of BEM and CFD has established new potential for design and research by introducing environments in which we can manipulate and observe (Kaijima et al. 2013). CFD technology involves fluid mechanics, computing methods, computer graphics and many other disciplines (Guo et al. 2015). Today, CFD simulations are widely used as a design assistance tool by architects and engineers because CFD analysis can generate detailed information about building thermal performance, such as space cooling and heating loads, distributions of indoor air velocity, flow, temperature, and contaminant concentrations through all design stages. Architects and engineers collaborate to improve the quality of buildings by evaluating thermal comfort, indoor air quality, and energy consumption of a building. Generally, after initial design by the architect, an engineer creates the BEM to assess the building performance and optimize the design through possible energy conservation measures. To evaluate the thermal comfort and air flow in and around the building, CFD analyses are used in order to optimize the building design and HVAC system (Kim 2014). However, CFD is a complicated and time-consuming process which requires users to have some knowledge of mathematical modeling and experience with numerical methods. Integrating BEM with CFD reduces the computational and development costs for the industry applications and enables the modeler to control the model accuracy specific for each design stage (Padovani et al. 2011).

CFD analysis tools

There exist over 200 CFD related software packages with different capabilities and levels of perfomance over one another. The Open-Source CFD tools such as OpenFOAM permit users to study, change and improve the software; however, it lacks a helpful user-support and user friendly environment. In order to make open-source more user friendly, developers have wrapped CFD codes into more user friendly GUI environments bundled with additional software such as pre- and post-processors (ex: VisualCFD, HELYX and simFlow). Among various CFD analysis tools in the market, three commercial CFD software are more popular in evaluating indoor thermal comfort: ANSYS-Fluent, IES VE-Microflo and Star-CCM+. ANSYS-Fluent and Star-CCM+ offer more precise computations of fluid dynamics while IES ModelIT permits quick geometry creation which feeds into IES VE-Microflo enabling the user to test concepts at the early design stages with acceptable accuracy level and low computational time and costs (Li 2015). In CFD analysis, accurately defining the boundary conditions is very crucial. According to a study conducted by Zhai and Chen (2006), BEM provides the dynamic boundary conditions such as supply airflow rate and surface temperatures that can be used in a coupled CFD simulation to effectively predict the dynamic indoor environment through the entire design day. The only disadvantage of the full dynamics coupling is a much longer computing time. In IES VE, the dynamic energy model is coupled with the CFD analysis tool (Microflo) to determine the boundary conditions like surface temperatures, estimating the various heat gains/losses, flow rates through natural ventilation openings and amount of heating or cooling required under different conditions. This allows modelers to take into account the variations in gains and external conditions for the CFD model. The CFD model is then populated using these values, which provides a deeper insight into the conditions across the domain being analyzed.

MicroFlo is based on 'Finite Volume Method' of discretization of the partial differential equations that describe the fluid flow and uses steady-state three-dimensional convection-conduction heat transfer and flow model. MicroFlo features a structured non-uniform Cartesian grid. MicroFlo can only read the internal gain data when the boundary conditions are manually exported from BEM results and imported into MicroFlo by the user. Any changes/additions made to the model in MicroFlo will not be reflected in the results of the BEM simulation (IES 2015).

Literature review-CFD modeling

Several studies show the benefits of using CFD models in different stages of design. CFD modeling can help architects in evaluation and performance quantification of the passive design in green building projects. In a study conducted by Guo et al. (2015), CFD simulation was used to optimize the building design in terms of room depth, elevation aesthetics, and function to facilitate natural ventilation. In a similar study, the influence of the roof pitch and geometry of a generic isolated lowrise building was evaluated using CFD methods and discovered that the building with a 45-degree roof inclination angle provides 22% higher volume flow rate (natural ventilation) than for the reference case (Perén et al. 2015).

CFD modeling also helps mechanical engineers design HVAC systems which meet the owner and architect's needs. In this regard, the majority of the CFD studies are focused on investigating and predicting the environmental conditions and thermal comfort of the occupants in various environments such as conference rooms (Hajdukiewicz et al. 2013), libraries (Aryal and Leephakpreeda 2015), theaters, and religious buildings (Aste et al. 2017). Nada et al. (2016) studied the performance of an Under Floor Air Distribution (UFAD) system in a high ceiling theater and found that properly selecting the supply air temperature and velocity (64°F (18°C) and 2.6 ft/s (0.8 m/s)) with a higher numbers of diffusers, energy savings of UFAD system increased as the theater height increased.

In the literature, the results of CFD simulation in large-scale sport facilities can also be seen, e.g. the multifunctional Galatsi Olympic Hall in Athens (Greece) (Stamou et al. 2008) , the Amsterdam ArenA football stadium in Amsterdam (Netherlands) (van Hooffand Blocken 2009), and the halls of the "Cittàdello Sport"in Rome (Italy) (Caruso et al. 2007).

Stamou et al. (2008) evaluated thermal comfort in the Galatsi Arena stadium with CFD simulations, considering heating, ventilating and air conditioning systems and assuming two possible inlet air temperatures: 57°F (14°C) and 60°F (16°C). The calculated values of Predicted Mean Vote (PMV) and Predicted Percentage of Dissatisfied (PPD) showed that the thermal conditions were satisfactory when the inlet air temperature was equal to 60°F (16°C).

Few examples of CFD application in the design or study of ventilation systems for indoor ice rinks and multipurpose arenas can be found in published journals. This might be due to the complexity of multipurpose arena buildings' heating, cooling and air conditioning and because many objectives have to be reached simultaneously, such as occupants thermal comfort and optimal internal climate suitable for the distinctive activities, with special operational profiles and requirements (Tsoka 2015).

Koper (2016) tested a large sports event, an exhibition and an indoor ice event using CFD. He observed that for the ice event and exhibition, the thermal comfort was acceptable, but during the large sports event with the full audience the air temperature in the occupied zone was much higher than the desired value.

Palmowska and Lipska (2017) investigated the influence of different factors such as dehumidification, air distribution, number of people; and the impact of a low-emissivity ceiling on improvement of indoor thermal and humidity conditions in an actual ventilated ice rink arena. They determined that it was necessary to increase the volume flow rate of drying air and to apply a recirculation system with the minimal hygienic share of outdoor air. Another way to prevent condensation on the inside roof surface was to increase its temperature, which was achieved by installing a low-emissivity ceiling below it which reduced the dehumidifier capacity.

In the above studies, the geometry is generally over simplified and the boundary conditions (surface temperature and internal heat gains are numerically calculated or measured on-site and then manually imported into the CFD model which is very laborious and not cost-effective for most real-world design applications.

This paper presents an industry case study of a multipurpose arena to demonstrate how integrating the CFD analysis with BEM facilitates and accelerates the modeling process and influences the HVAC design process to achieve optimum thermal comfort. This study evaluates the thermal comfort inside the arena bowl by comparing five different HVAC air-side configurations using IES VE's CFD module (MicroFlo).

BEM AND CFD SIMULATION MODEL

The case study building is a multipurpose arena located in Elmont, New York. This new facility is a modern 18,000-seat arena consisting of the main arena, seating areas, surrounding concourse areas, transition areas, corridors, surrounding retail & restaurant areas, gym, food preparation, mechancial rooms, support areas including offices, dining and changing facilities. The total building floor area is approx. 590,000 ft^2 (55,000 m^2). The arena is expected to host approximately 250 events plus 41 NHL hockey games. With the remaining days being building down time. The main arena is composed of a large bowl with high ceiling, which requires a high level of dependency on mechanical ventilation with conditioned air. Furthermore, a great amount of energy could be required by traditional air distribution to maintain the optimal indoor conditions for a comfortable environment. For facilities with high ceilings, a UFAD system would also be an appropriate option to enhance thermal comfort with energy savings and allow both individual controls of ventilation volume and distribution of air only to occupied zones.

Five different HVAC air-side configurations were considered in the early stages as listed in Table 1, in order to study the bowl's environmental conditions during events. An evening concert event was selected for this study because the main arena bowl operates for events such as concerts and shows more than hockey games (250 events compared to 41 games) and the design team was more concerned about the HVAC performance when the arena floor is densly occupied with people at a relatively high activity level. Moreover, the operable baffle ceiling designed by the architect to separate the hockey, full and half house modes from the roof structure was another concern for the design team. In hockey mode, the bowl has the maximum volume and the operable baffle ceiling plane is located close to the cat walk without interrupting the air flow into the bowl. However, in the full house mode (floor area, lower bowl area, suite, and upper bowl area) and the half house mode (floor area, and lower bowl area) the lowered operable baffle ceiling may affect the airflow into the bowl, which needed further investigation.

To develop the BEM, a custom building annual operating schedule was created based on the events calendar. During game nights the arena occupancy levels peak for four hours from 18:00 to 22:00. Occupancy levels during standard event evenings were also assumed to follow a similar occupancy profile (music concerts, etc.). Day time occupancy levels were assumed to be 1% of the evening occupancy levels. This value essentially covers arena ground staff and team practice sessions.

The design team is considering the following Energy Conservation Measures (ECMs):

- Air side economizer (bowl and selected Air Handling Units (AHU)) and water side economizer
- Air side energy recovery in selected AHUs
- Demand Control Ventilation (DCV) in bowl area
- High-efficiency Variable Speed Drive (VSD) water-cooled chillers
- High-efficiency Variable Refrigerant Volume (VRV) system for electrical rooms
- High-efficiency modular condensing boilers (94% efficiency)
- VSD Condenser Water (CW) and Hot Water (HW) pumps
- Ice plant condenser heat recovery
- High-performance building envelope
- High-efficiency LED general lighting and bowl sports lighting

All of the above listed ECMs were explicitly modeled using IES VE software. After creating the BEM, the MicroFlo CFD model is employed in order to investigate environmental conditions and thermal comfort within the arena bowl.

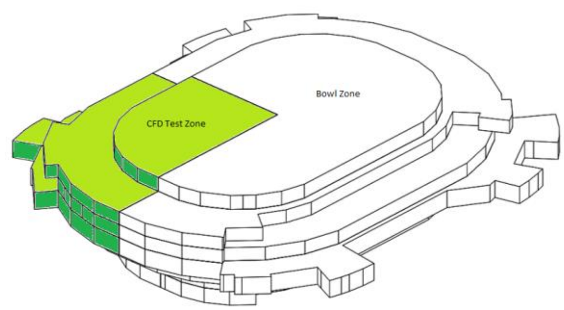


Figure 1 Bowl zone including CFD test zone

As shown in Figure 1, the bowl zone is a fairly large space and currently beyond the modeling capability of MicroFlo. After creating the CFD grid for the entire bowl zone, we realized that the aspect ratio was high. This is a critical parameter of the grid and the health of the solution is very much linked to this parameter. Cell aspect ratio is basically the ratio of the lengths of the longest and shortest edges of any of the cells within the domain (IES 2015). In order to achieve a lower aspect ratio (50:1), the CFD domain is limited to a quadrant of the bowl with the grid spacing of 4ft (1.2m). The turbulance model is set to standard k-e model which calculates turbulent viscosity for each grid cell throughout the calculation domain. The boundary condition was then assigned to the CFD model from the BEM for one instance of time (concert night). The imported boundary condition includes all surface (wall, window, door, and opening) temperatures and convective components of the internal gains specified in the BEM (overhead and sport lighting, scoreboard and large screen monitor heat gain). Spectator sensible heat gain per ASHRAE is 250 Btu/hr per person (73W per person). Assuming an attendance of 18,000, this load was evenly distributed to the arena seating bowl levels as shown in Figure 2.

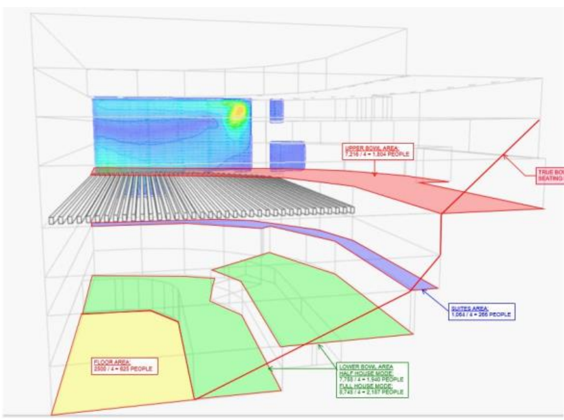


Figure 2 Spectators heat gain in the bowl

After adding internal gains, supply diffusers and return air extracts were added to the CFD model as shown in Figure 3.

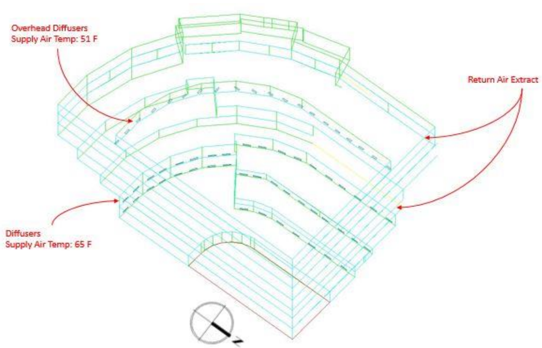


Figure 3 Supply diffusers and return air extracts

Conditioned air supplied to the arena was modeled at 51℉ (11℃) and 65℉ (18℃) for simulation of overhead and underfloor air distribution system respectively. Air flow rates, velocities and diffusers location, and trajectories are changed according to each option's specifications as listed in Table 1.

RESULTS AND DISCUSSION

The CFD simulation was performed during a concert event on the design day at 21:00 in order to evlautate the thermal comfort of the occupants with a higher activity level at the arena bowl's floor area. In this study, thermal comfort is assessed using the PMV and PPD approach. The PMV and PPD indicate overall thermal satisfation of the occupants within specific environmental conditions.

Table 1 CFD analysis scenarios

Air Delivery Location	**OHAD**		
Description	Upper ring duct (directed to back of dasher board)	Upper ring duct (directed at 45 degrees down angle to upper bowl)	Outboard upper ring duct
Option 1	80,000 cfm (37,755 L/s), 1100 fpm (5.6 m/s), 51°F (11°C)	20,000 cfm (9,438 L/s), (1200 fpm) (6 m/s), 51°F (11°C)	n/a
Option 4	60,000 cfm (28,316 L/s), (1100 fpm) (5.6 m/s), 51°F (11°C)	n/a	40,000 cfm (18,877 L/s), (1200 fpm, 6 m/s), 51°F (11°C)
Option 5	n/a	n/a	100,000 cfm (47,194 L/s), (1100 fpm, 5.6 m/s), 51°F (11°C)

Air Delivery Location	**UFAD**	
Description	UFAD (Upper bowl)	UFAD (Lower bowl)
Option 2	40,000 cfm (18,877 L/s),65°F (18°C)	60,000 cfm (28,316 L/s), 65°F (18°C)
Option 3	n/a	60,000 cfm (28,316 L/s), 65°F (18°C)

The PMV refers to a thermal scale that runs from Cold (-3) to Hot (+3), which was originally developed by Fanger and later adopted as ISO Standard 7730:2005. The PPD predicts the percentage of occupants that will be dissatisfied with the thermal conditions. It is a function of PMV, given that as PMV moves further from 0, or neutral, PPD increases. There are six factors taken into consideration when calculating thermal comfort: metabolic rate (met), clothing level (clo), air temperature, radiant temperature, air velocity, and humidity. Metabolic rate for the occupants and clothing levels were assumed at recommended values according to ASHRAE Standard 55-2017 (1.7-2 met, 1 clo) for standing and walking. Relative humidity was 50% as it is controlled by the HVAC system. Air temperature, air velocity and mean radiant temperature were the average values of air temperature, relative air velocity and mean radiant temperature inside the room simulated by the integrated CFD model.

Over-Head Air Distribution (OHAD)-system Options 1, 4 & 5

OHAD system does not inject the conditioned air directly into the occupied zone. Supply air coming from these systems is generally between 50°F (10°C) and 55°F (13°C) with a higher velocity than an UFAD system. Figures 4 through 6 show different OHAD configurations for conditioning the lower and upper bowl areas.

The CFD simulation model was used for evaluating the thermal field, the PMV and PPD profile within the occupied zone and to point out the effects of different air distribution configurations in the bowl during an event. Figure 7 compares three views of the simulated thermal field along the Y-axis and Figure 8 compares three views of the simulated thermal field along the X-axis for option 1, 4, and 5. The temperature range is between 55°F (13°C) to 72°F (22°C). The comparison shows that installing supply air diffusers above the operable baffles (as in option 1 and 4) does not let the cool air reach the center of the arena floor. It also shows that option 4 may be a better case in cooling the upper bowl spectators; however option 5 has the most uniform thermal field with less wasted conditioned air above the operable baffles.

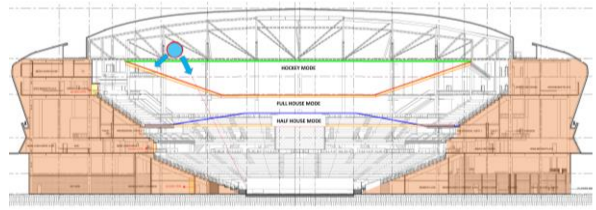


Figure 4 Option 1-OHAD from Upper ring duct directed at two different angles

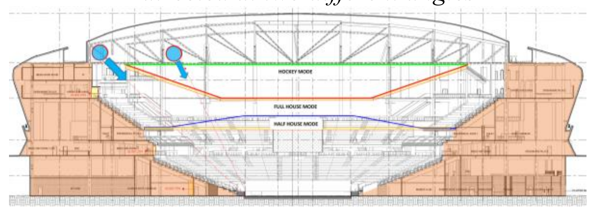


Figure 5 Option 4-Combination of OHAD from upper ring duct and outboard upper ring duct directed at one angle

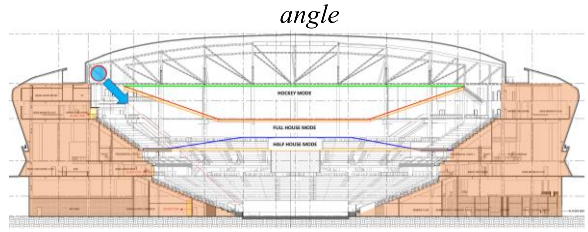
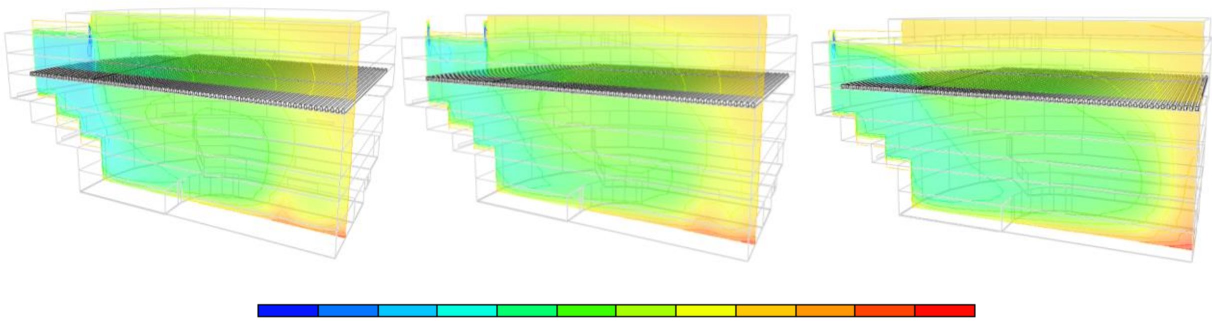
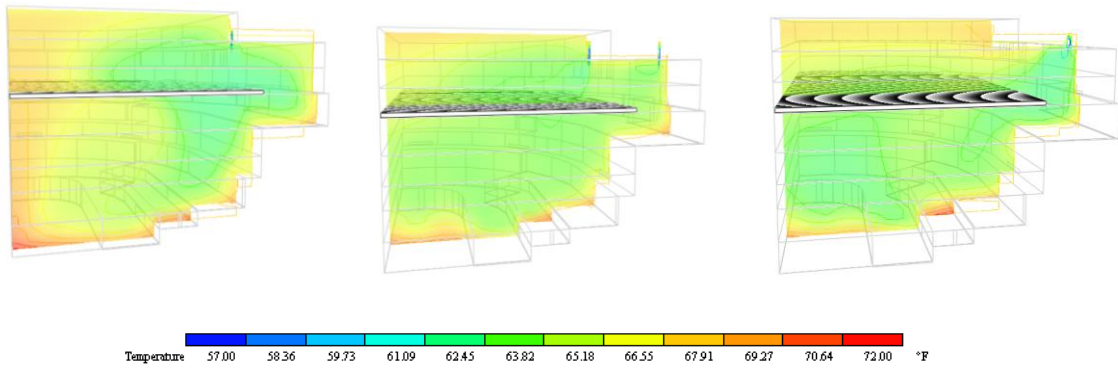


Figure 6 Option 5-OHAD from outboard upper ring duct directed at one angle



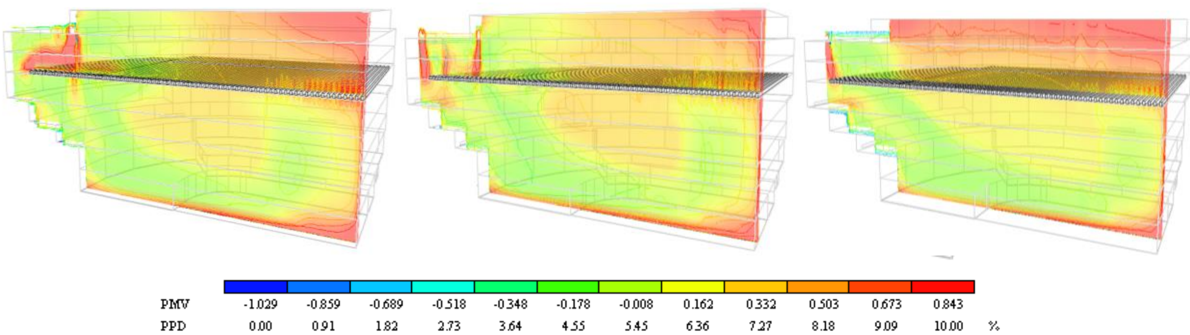
(a) *(b)* *(c)*

Figure 7 (a) Option 1 (b) Option 4 (c) Option 5 temperature profile (Y axis)



(a) *(b)* *(c)*

Figure 8 (a) Option 1 (b) Option 4 (c) Option 5 temperature profile (Y axis)



(a) *(b)* *(c)*

Figure 9 (a) Option 1 (b) Option 4 (c) Option 5 PPD profile

Figure 9 compares three views of the simulated PPD field along Y-axis for option 1, 4 and 5. The maximum possible number of people dissatisfied with their comfort conditions is 100%. Since you can never please all of the people all of the time, the recommended acceptable PPD range for thermal comfort from ASHRAE 55 is less than 10% persons dissatisfied for an interior space.

The comparison shows that option 5 has less discomfort toward the center of the floor and it may be a more efficient air distribution system in terms of energy use because it does not inject the conditioned air into the unoccupied area above the operable baffle ceiling.

Under-Floor Air Distribution System-Option 2, 3

UFAD systems supply conditioned air directly into the occupied zone of the building. This system allows higher thermostat setpoints compared to traditional overhead systems. UFAD has several potential advantages over traditional overhead systems; however, in an environment with high activity levels, the performance should be evaluated before decision making.

Figure 10 shows the configuration of UFAD system in the lower and upper bowl area and Figure 11 illustrates the combination of UFAD in the lower bowl and OHAD directed at 45 degrees down angle to the upper bowl.

Figure 12 compares two views of the simulated thermal field and Figure 13 illustrates two views of PPD profile along the Y-axis and X-axis for option 2 and 3.

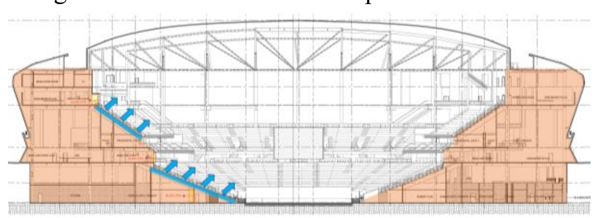


Figure 10 Option 2-UFAD in lower and upper bowl

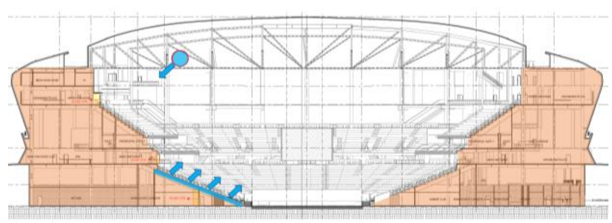
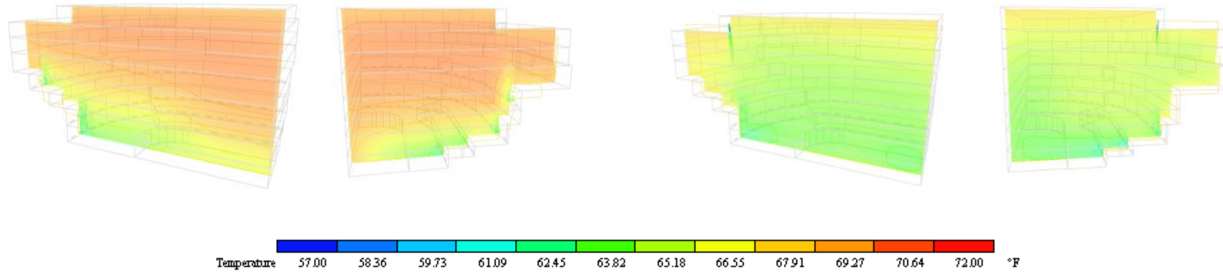
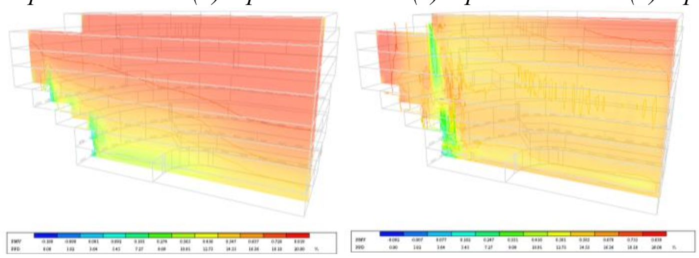


Figure 11 Option 3-UFAD and OHAD combination



(a) *(b)* *(c)* *(d)*

Figure 12 (a) Option 2- Y axis (b) Option 2- X axis (c) Option 3- Y axis (d) Option 3- X axis



(a) *(b)*

Figure 13 (a) Option 2 (b) Option 3 PPD profile

The figures show higher temperature in option 2 because of higher supply air temperature and lower air velocity. In addition, option 2 has higher PPD (12-14 percent) than other options at the arena floor, which sees high activity levels during a concert event; ideal/uniform thermal comfort is not achieved.

The designers tried to improve the thermal comfort by combining the OHAD with UFAD in option 3. Although the thermal field is now more uniform with lower temperature ranges than option 2, the PPD is still higher than 10 percent and the upper bowl spectators do not have acceptable thermal comfort.

Among all five options, option 5 appears to provide adequate and uniform thermal comfort while using less energy due to not injecting the conditioned air into the unoccupied area. UFAD system might have a better performance in spaces such as offices, theaters, churches, and spaces in which occupants are mostly seated. But for an environment such as multi purpose arena with a relatively higher activity level during events and limited solutions to deliver air to the event floor via UFAD, the thermal performance will not be completely achieved using UFAD in instances like this.

After selecting the best HVAC air distribution system in terms of thermal comfort, the integrated BEM is modified based on the optimized system solution and incorporating all building ECMs using IES VE software. The final model shows 20% energy savings over the ASHRAE 90.1-2010 baseline.

IES VE advantages and challenges

IES VE was used as the tool for this study. The advantages and challenges are listed below:

- ModelIT module is used to create the BEM that is also used for the CFD model. There was no need to create a separate CFD model, which would have been time-consuming within our fasttrack timeframe to provide the design team with feedback.
- IES VE MicroFlo has its own meshing tool which is called “CFD Grid”. This tool has the limitation of 3-4 million cells for the CFD model to run. In order to stay in the range for a large model, the model needs to be divided into smaller zones for the CFD analysis. Although the degree of precision in meshing is lower than other tools, IES VE MicroFlo uses the least time during generation of the mesh.
- The boundary condition (thermal envelope characteristics, weather data, and internal gains) is directly imported from the BEM, which was faster than other methods within the reduced timeframe.
- MicroFlo does not provide precise point value of any variable at any location within the simulated domain. The value of the selected point can approximately be measured on the distribution profiles which is not precise enough for some critical HVAC designs.

CONCLUSION

An integrated BEM with CFD model was developed using IES VE for an multipurpose arena to evaluate thermal comfort inside the bowl with different HVAC air-side configurations. PMV and PPD are calculated by the software to quantify the thermal comfort and facilitate the analysis process. The CFD analysis results show that an OHAD system located at the far end of the bowl is the most effective option in achieving thermal comfort for this case study building during a high activity event with a dense occupancy. BEM and CFD integration demonstrates the effective use of CFD analysis in the early design stage decision-making process of HVAC systems. In this study simplifications of the CFD model were inevitable to attain reliable results in a short timeframe, which is necessary from productivity standpoint for an industry case buidling.

Although IES VE-MicroFlo has some limitations in generating grids for large spaces and providing precise values within the simulated domain, this case study confirmed the ability to model systems and solutions despite software limitations to study conceptual solutions.

REFERENCES

Aryal, P., Leephakpreeda, T. 2015. CFD Analysis on thermal comfort and energy consumption effected by partitions in air-conditioned building. Energy Procedia, 79, 183-188.

ASHRAE Standard 2017. Standard 55-2017. Thermal environmental conditions for human occupancy, 9-11.

Aste, N., Della Torre, S., Adhikari, R. S., Buzzetti, M., Del Pero, C., Leonforte, F., & Cardenas, H. H. 2017. CFD Comfort Analysis of a Sustainable Solution for Church Heating. Energy Procedia, 105, 2797-2802.

Caruso, G., De Santoli, L., & Mariotti, M. 2007. Ventilation Design in Large Enclosures for Sports Events using CFD: the Halls of the "Città dello Sport" in Rome. Clima.

Guo, W., Liu, X., & Yuan, X. 2015. A Case Study on Optimization of Building Design Based on CFD Simulation Technology of Wind Environment. Procedia Engineering, 121, 225-231.

Hajdukiewicz, M., Geron, M., & Keane, M. M. 2013. Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. Building and environment, 70, 73-89.

International Organization for Standardization. 2005. Ergonomics of the Thermal Environment: Analytical Determination and Interpretation of Thermal Comfort Using Calculation of the PMV and PPD Indices and Local Thermal Comfort Criteria. International Organization for Standardization.

Kaijima, S., Bouffanais, R., Willcox, K., & Naidu, S. 2013. Computational fluid dynamics for architectural design. Architectural Design, 83(2), 118-123.

Kim, D. 2014. The Application of CFD to Building analysis and Design: A Combined Approach of An Immersive Case Study and Wind Tunnel Testing (Doctoral dissertation, Virginia Tech).

Koper, P. 2016. Performance assessment of air conditioning installation in multifunctional sports hall using CFD simulations. Architecture Civil Engineering Environment, 9(4), 123-134.

Li, N. 2015. Comparison between three different CFD software and numerical simulation of an ambulance hall.

Nada, S. A., El-Batsh, H. M., Elattar, H. F., & Ali, N. M. 2016. CFD investigation of airflow pattern, temperature distribution and thermal comfort of UFAD system for theater buildings applications. Journal of Building Engineering, 6, 274-300.

Padovani, R., Hes, D., MacLaughlin, F. 2011. Dynamic thermal modeling and CFD simulation techniques used to influence the design process in buildings. Proceedings of Building Simulation 2011, 12th Conference of International Building Performance Simulation Association, Sydney, 14-16 November.

Palmowska, A., & Lipska, B. 2017. Research on improving thermal and humidity conditions in a ventilated ice rink arena using a validated CFD model. International Journal of Refrigeration.

Perén, J. I., Van Hooff, T., Leite, B. C. C., & Blocken, B. 2015. CFD analysis of cross-ventilation of a generic isolated building with asymmetric opening positions: impact of roof angle and opening location. Building and Environment, 85, 263-276.

Stamou, A. I., Katsiris, I., Schaelin, A. 2008. Evaluation of thermal comfort in Galatsi Arena of the Olympics "Athens 2004" using a CFD model. Applied Thermal Engineering, 28(10), 1206-1215.

Tsoka, S. 2015. Optimizing indoor climate conditions in a sports building located in Continental Europe. Energy Procedia, 78, 2802-2807.

Van Hooff, T., & Blocken, B. 2009. CFD analysis of natural ventilation in large semi-enclosed buildings-Case study: Amsterdam Arena football stadium. In Eleventh International IBPSA Conference, Glasgow, Schotland (pp. 27-30).

Zhai, Z. J., & Chen, Q. Y. 2006. Sensitivity analysis and application guides for integrated building energy and CFD simulation. Energy and buildings, 38(9), 1060-1068.