Zhai, Z., Chen, Q., Scanlon, P.W. 2002. “Design of ventilation system for an indoor auto racing complex,” *ASHRAE Transactions*, 108(1), 989-998.

Design of a Ventilation System for an Indoor Auto Racing Complex

Zhiqiang Zhai, Student Member ASHRAE
Qingyan Chen, Ph.D., Member ASHRAE
Paul W. Scanlon, P.E., Member ASHRAE

ABSTRACT

Computational fluid dynamics (CFD) can be a very useful tool to help designers evaluate the indoor air quality and thermal comfort provided by HVAC systems. This paper describes how a CFD program is being used to guide the design process and optimize the ventilation system design for the world’s first indoor auto-racing facility. The facility is primarily a single space building with a floor area of over $2x10^6$ ft^2 ($0.2x10^6$ m^2) and a ceiling height of 150 ft (46 m). The facility is being designed to accommodate a variety of future possible occupancy conditions for a wide variety of events - 60,000 spectators in the grandstands and/or 60,000 spectators in the infield, as well as lesser occupancies within various areas of the infield. Of all the possible events anticipated for this unique facility, the design team selected a “worst case” condition based on the maximum possible occupant density during a racing event in which 45 race cars are running simultaneously on the track at speeds of up to 155 mph (250 km/h). The CFD results are being used to improve the base case – conventional ventilation system design, step by step, to an optimal design – combination of displacement ventilation and overhead duct system and partial air curtain system. The CFD results are much more informative and accurate than those that could be obtained manually or with other methods.

Zhiqiang Zhai is a Research Assistant and Qingyan (Yan) Chen is an Associate Professor at the Building Technology Program, Department of Architecture, Massachusetts Institute of Technology, MA, and Paul W. Scanlon is the Director of Engineering, Baker and Associates, Beaver, PA.

INTRODUCTION

Ventilation systems are critical to maintaining a comfortable and healthy indoor environment in a building. Conventional design of indoor air distribution typically relies on the use of charts provided by diffuser manufacturers and jet formulae that were developed from laboratory data. The use of such data can result in great uncertainties when it is applied to large spaces (such as atria, concert halls, and sports facilities) or applications that are dissimilar from those upon which the laboratory data was developed.

Alternative design aids, including the use of instrumented, full-scale mock-ups and/or smaller scale physical models, may also be of limited practical use to the design team when applied to the design of large facilities.

One approach used in the past to verify the design of indoor air distribution is to build a full-size mockup with which the distributions of airflow, air temperature, and contaminant concentrations can be physically measured. However, such a mockup study is generally time-consuming and expensive. In fact, the mockup is normally used to evaluate small spaces, such as an office space. It is not practical to build a full-scale mockup for large spaces, such as sport facilities.

A similar approach is to construct a scale model rather than a full-scale mock-up. Complicating this approach is the fact that small-scale models may require the use of a different fluid than air, such as water (Heiss 1987) and refrigerant R-114 (dichlorotetrafluoroethane) (Olson and Glicksman 1991), in order to ensure the similarity of the inertial forces and buoyancy forces between the scale model and a full-scale facility. Even when such steps have been taken in the past, however, Heiss (1987) and Olson and Glicksman (1991) found discrepancies in the results between the models and prototypes due to scaling problems. Moreover, due to the difficulties in changing such models, they are rarely used to achieve an optimal design by comparing different alternatives. For these reasons, scale models are generally used only for the evaluation of final designs.

Due to the costs and problems in the mockup and scaling model approaches, numerical simulations by computational fluid dynamics (CFD) have proven to be an increasingly attractive approach in ventilation system design. Recent advances in computing power and friendly data input/output interfaces enable a designer to use CFD to predict the air distribution, heat transfer, and chemical species transport in an indoor environment with different ventilation systems. Some examples of CFD applications for flow prediction in large spaces include the work from Guthrie et al. (1992), Yuan et al. (1992), Moser et al. (1995), Nielsen (1995), and Yang et al. (2000).

Compared with mockup and scale modeling, CFD is inexpensive and can quickly provide designers detailed information concerning airflow, heat transfer, and chemical species transport. The CFD is extremely useful for the optimization of ventilation system design in the early design process. Changing a design parameter in CFD simulations, such as airflow rate, is as simple as a click of the mouse. However, because the CFD uses turbulence models, numerical techniques, and approximations in modeling airflow in a building, there may still be uncertainties in the CFD results. Furthermore, the simulation of air distribution in large-scale spaces impose additional difficulties because the flows are more complex, as evidenced by the high Reynolds and Grashof numbers in the flows due to the large characteristic length and temperature gradient. A designer's experience in using the CFD approach is therefore quite crucial in resolving any uncertainties.

This paper demonstrates a design process using a CFD program to guide the design of the ventilation system for a large space, an indoor auto-racing complex to be built in Pittsburgh, PA. The design is especially challenging because of the high speed of the racing cars, a huge difference between the size of the building and the heated objects (such as spectators), and the enormous amount of heat generated from the cars. To accomplish the goals of this project, the CFD consultant had to work very closely - from the outset of the project - with the facility's design engineers.

CFD TECHNIQUE

The present investigation used the CFD technique to solve a set of partial differential equations for the conservation of mass, momentum, energy, and species concentrations. These equations govern the transport phenomena in the indoor auto-racing complex. Since airflow in the building complex is turbulent, the CFD technique used a turbulence model (the renormalized-group k-ε model from Yakhot et al. (1992)) to reduce the computing costs. With the turbulence model, the airflow, temperature, and species concentration transport can be described by the following unsteady time-averaged Navier-Stokes equations:

$$\frac{\partial(\rho\Phi)}{\partial t} + \mathrm{div}(\rho V\Phi - \Gamma_{\Phi,\mathrm{eff}} \cdot \mathrm{grad}\Phi) = S_\Phi \qquad (1)$$

Where

ρ = air density

Φ = 1 for mass continuity

= V_j (j = 1, 2, 3) for three components of momentum

= k for turbulent energy

= ε for the dissipation rate of k

= T for energy transport

= C_i for contaminant concentration i

V = velocity vector

$\Gamma_{\Phi,\mathrm{eff}}$ = effective diffusion coefficient

S_Φ = source term

Many textbooks have detailed the CFD theory, such as Wilcox (1993) and Versteeg and Malalasekera (1995). The governing equations can be closed with appropriate thermo-fluid boundary conditions at all the boundaries such as air inlets, outlets and wall surfaces. The values of velocity, temperature, kinetic energy, the dissipation rate of kinetic energy, and species concentration should be set at the boundaries.

A commercial CFD program (CHAM 1999) was used in the present investigation to solve the time-dependent conservation equations together with the corresponding boundary conditions. The program discretized the indoor space into non-uniform computational cells (length3width3height=1003100355), and the discrete equations were solved with the SIMPLE algorithm (Patankar 1980).

BASE CASE SETUP

The indoor auto-racing complex studied is a large space as shown in Figure 1. The geometry of the space is irregular, and is approximately 2300 ft (700 m) long, 1200 ft (366 m) wide, and 150 ft (46 m) high. The actual floor area is over 2×10^6 ft^2 (0.2×10^6 m^2). The space is being designed to accommodate auto-racing events as well as other activities, such as conventions, trade shows, consumer expos, entertainment, and other recreational events. The central structure will house a one-mile banked oval speedway with up to 60,000 permanent seats in the grandstands and 60,000 temporary seats in the infield area. In addition, it will provide sixty to two hundred luxury skyboxes for lease. The facility will have special lighting and large screen displays for televised events, food and retail concessions stands, and so on. The track facility is designed for a maximum of 45 racing cars running simultaneously on the track at a maximum speed of 150 mph (242 km/h) and an average speed of 135 mph (217 km/h). Such a large-scale and complicated building with a variety of indoor components undoubtedly challenges the experience and capability of ventilation system designers, even with the aid of CFD modeling as a design tool.

It is not an easy task to design a ventilation system for such a large space that can provide an acceptable thermal comfort and indoor air quality level in the occupied zones.

This is because an enormous amount of heat and chemical components generated from the fuel used by the cars will have a strong impact on the thermal comfort and air quality experienced by the occupants. Combining the conventional approach to sports facility HVAC design with some new design concepts, a "base case" ventilation system design for this space was established as illustrated in Figure 2. This initial concept included supplying fresh air to the grandstands (occupied zone #1) and the infield (occupied zone #2); in addition, air-curtains between the track and the occupied zones were envisioned to help isolate the occupied zones from the hot plumes generated by the cars. The rising plumes of hot, polluted air were then to be mechanically exhausted from a series of large exhaust fans located along two clerestories at the roof level. The grandstand area of the base case design assumed a traditional overhead duct system to supply fresh air, while the infield area was assumed to be ventilated by a displacement ventilation system that supplies fresh air underneath the seats. Rather than attempting to provide full air-conditioning of the entire volume of the facility during a major racing event, the design goal was to use the required ventilation air to provide partial "spot-cooling" of occupied areas, which would provide comfort levels similar to that experienced by racing fans in a conventional outdoor race track.

With a minimum ventilation rate of 15 cfm (7 L/s) per spectator, the total fresh air supply for 120,000 spectators is 1.8×10^6 cfm (850 m^3/s). Therefore, the ventilation system is designed to supply 1.0×10^6 cfm (472 m^3/s) of fresh air to the grandstands (including skyboxes) and the same amount to the infield. The air supply temperature was assumed to be 50°F (10°C) at a humidity ratio 0.008 kg_v/kg_a. The ventilation system also included another 1.0×10^6 cfm (472 m^3/s) of unconditioned fresh air to create an air curtain between the occupied zones and the track. The air velocity from the air curtain was initially set at 600 fpm (3 m/s). Table 1 gives a summary of the ventilation system and the air supply parameters for the base case.

In order to use CFD modeling to successfully simulate the air distribution in the complex, a CFD model must first be created which translates the real world into a description of the flow physics suitable for numerical processing. Simplifications and numerical techniques were needed to describe special thermo-fluid components in the complex, such as moving cars. The following paragraphs describe the simplifications and techniques used.

Building Enclosure

The CFD model represents the auto-racing facility in a similar but abstract manner, as shown in Figure 3. Figures 1 and 3 look alike, but there are differences. For example, the curved racetrack is simulated in the CFD model using square blocks.

The ventilation systems for the complex must be designed to work under a worst-case scenario. For this facility, the design team established the worst-case scenario as a major auto-racing event, with a maximum number of racing cars on the track and the maximum number of spectators inside the complex, under summer design conditions. According to the summer design conditions for Pittsburgh, PA, the dry-bulb air temperature is 86°F (30°C) and the corresponding wet-bulb temperature is 70°F (21°C). With a roof insulation value of R-13 and R-4 insulation for the walls, the interior surface temperature of the roof and walls were assumed to be 99°F (37°C) and 86°F (30°C), respectively, for the computations. The CFD model also assumed an adiabatic floor.

Cars

Simulating a moving car using the moving boundary technique in the CFD model for such a space is almost impractical. However, the impact of moving cars on the indoor environment can be reasonably approximated by considering their velocity momentum and their affects as heat and contaminant sources. Therefore, the CFD model simulated these 45 cars as "still" objects with momentum, heat, and contaminant sources characteristics. This approach has been proved to be acceptable and practical in one of our earlier studies (Yang et al. 2000), which simulated a moving ice resurfacer in an ice rink. In that investigation, the CFD predictions of indoor airflow, temperature, and contaminant concentration in an ice rink agreed well with the measured data obtained in the rink.

A simplifying technique used in this study was to group the 45 racing cars into 15 groups of three cars each, in order to reduce the data inputs. The 15 groups of cars were uniformly distributed on the track and assumed to be traveling at the same average speed of 135 mph (217 km/h). The heat generated by each car is 750 horsepower (559 kW). There are contaminants, such as about 3 kg/hour lead, from the gasoline used by 45 cars.

A typical racing event lasts for three to five hours. At the end of the event, the indoor conditions reach a steady state. Therefore, the worst-case scenario should use the steady-state conditions.

Spectators

The total heat generated by each person in such an event is 510 Btuh (150 W), and the moisture generation rate is 0.121 lb/h (0.055 kg/h) per person. However, due to the scale-difference and input-quantity limitations, it is impossible to simulate so many spectators individually in the CFD model. Therefore, focusing on the macro influence of the spectators on the indoor environment, all of the spectators are simplified into several solid blocks of resistance, heat and moisture sources. The CFD model uses an average occupied area of 5.5 ft^2 (0.5 m^2) for each person. The same technique is used for the spectators in the skyboxes.

Diffusers

Since the size of diffusers is much smaller than those of other components in the building and also the types of diffusers have not been specified at this early design stage, the investigation employs the uniform air-supply assumption for all the diffusers with the usage of the momentum method (Chen and Moser 1991) assuming 50% real supply area of the gross diffuser opening area.

DESIGN OPTIMIZATION PROCESS WITH CFD ANALYSIS

The CFD model described above serves as a base case (*Case 1*). This section demonstrates how the CFD model is used to improve the design of the ventilation system for the auto-racing facility. The CFD simulations predict the distributions of pressure, airflow, temperature, humidity ratio, relative humidity, lead concentration, etc. The predicted results are then used to incrementally improve the design in order to achieve an optimal design of the ventilation system.

Since almost all auto races are currently outdoor events, this facility is considered to be an "outdoor" space with roof coverage to reduce the discomfort level of the spectators from direct sunshine and rain. Therefore, the design is not intended to provide a comfort level comparable to the ASHRAE comfort standard (ASHRAE 1992). Due to the enormous amount of internal heat gains and reasonably dry outdoor design conditions, relative humidity in the complex is low. Additionally, spectators at such a sport event actually prefer some air movement to experience the sensation of the car movement. Therefore, air temperature is actually the most important comfort parameter. For indoor air quality, lead concentration is used as a representative pollutant, but it is simplified to be a general type of gas-phase pollutant in the CFD, instead of particles which involve the more complicated two-phase flow simulation.

Figure 4(a) shows the air temperature in the mid-section of the facility under the steady state condition, which means the cars are running on the track continuously and the outdoor air temperature is always at the design condition. Under these conditions, the mean air temperature in the grandstands is 95^{o}F (35^{o}C) and the infield is around 97^{o}F (36^{o}C). Since the cars run counterclockwise around the track shown in Figure 1(a), the car-induced winds coming out of the turn at the northwest corner causes the highly polluted and hot air from the track to penetrate into the grandstand area at that location. Table 2 summarizes the air temperature in the mid-section of the grandstands, the infield, and the northwest corner. Note that the air temperature is generally too high in all the occupied zones, and the air temperature at the northwest corner of the grandstands is the highest at 107^{o}F (42^{o}C) due to the car-induced air movement. The corresponding relative humidity is less than 40% for the base case. If we imagine the condition in the occupied zones as a shaded outdoor space with 96^{o}F (35.5^{o}C) air temperature and 40% relative humidity, it would be considered unseasonably warm even with the benefit of good air motion. Some modifications to the design were therefore considered necessary to lower the air temperature in the occupied zones.

Figure 5(a) shows the lead concentration distribution normalized to a source strength in the mid-section under the steady-state condition. Figure 6(a) shows the airflow pattern at 16 ft (5 m) above the track and Figure 6(b) illustrates the airflow in the mid-section close to the grandstands. The results shown in Figure 6 demonstrate that the air curtain to is not very effective in isolating the polluted, hot plumes from the occupied zones. In fact, the high-speed cars generate a strong backwind on the track that induces the contaminated air into the occupied zones at the northwest corner of the grandstands. The air curtain appears too weak to block the wind generated by the cars, although it does help to cool the air from the track and further dilute the pollutants associated with the car exhausts. Note that the car speed is 20 time higher than the velocity from the air curtain. In addition, the curtain is more than a mile long (2 km) long and the total air supply is only 1.0x10^{6} cfm (472 m^{3}/s). The air curtain is therefore fairly weak compared to the air induced by the cars.

In order to more closely evaluate the impact of the air curtain, the air curtain was removed from *Case 2*. Otherwise, Case 2 is the same as the base case. Without the air curtain, the lead concentration in the occupied zone became slightly higher. The increase of the lead concentration is far less than 33%, although the air supply rate without the air curtain was reduced by 1/3. This is because the overhead ventilation system in the grandstands and the displacement ventilation system in the infield were found to have a greater impact on the air quality in the occupied zones. However, the air temperature in the grandstands, the infield, and the northwest corner in Case 2 increased to 100, 106, and 111^{o}F (38, 41, and 44 ^{o}C), respectively. Thus it was determined that the air curtain can cool down the indoor air substantially, even though it uses unconditioned outdoor air that is only 86^{o}F (30^{o}C) - still much lower than the indoor air.

Case 3 replaced the base case overhead duct system in the grandstands with a displacement ventilation system that supplies air underneath the bleachers. With the same ventilation rate as that in the base case, the air quality in the grandstands for Case 3 is dramatically improved. The lead concentration is reduced by at least 20%, compared with that in the base case. The air temperature in Case 3 is also 2 to 8^{o}F (1 to 5^{o}C) lower than that in the base case, as detailed in Table 2. However, the displacement ventilation system was modeled to supply an air temperature of 50^{o}F (10^{o}F) - the same as that in the overhead duct system in the base case, which may lead to a cool draft sensation by the spectators. The draft problem cannot be confirmed by the CFD results, because the CFD model does not simulate the details of the air distribution around the spectators in the grandstands.

The results in the previous three cases show that the air temperature and lead concentration in the northwest corner of the complex are the highest. Hence, *Case 4* uses a partial air curtain in the northwest area between the auto tracks and the grandstands to isolate and dilute the polluted air. The total flow rate from the air curtain was only 3x10^{5} cfm (142 m^{3}/s). The supply air velocity from the air curtain was increased to 1,300 fpm (6.5 m/s). In addition, Case 4 uses also the displacement ventilation for the grandstands as in Case 3. All other thermo-fluid conditions are the same between Case 4 and the base case.

With these incremental design changes, the air temperature in the grandstands, the infield, and the northwest corner becomes 95, 99, and 100^{o}F (35, 37, and 38^{o}C), respectively. The air temperatures are higher than those in Case 3 because of the reduction of the total air supply from the air curtain. The air temperatures in the occupied zones are more uniform than those in both the base case and Case 3. Case 4 does not have any hot spots in the occupied areas. In addition, the lead concentration in the northwest corner of the complex is reduced by 10%, compared with that in the base case.

The results of the four cases studied find that the displacement ventilation seems more appropriate for the auto-racing facility. According to the investigation of displacement ventilation for offices, classrooms, and workshops by Yuan et al. (1999), the supply air temperature from a displacement diffuser should be substantially higher than that from an overhead air supply system to avoid creating an uncomfortably cool draft at the point of contact with the occupants. Although the auto-racing complex is regarded as an "outdoor" space with roof coverage, the supply air temperature of 50^{o}F (10^{o}C) should be increased.

Case 5 assumed displacement ventilation for both the grandstands and the infield. The supply air temperature was also increased to 65^{o}F (18^{o}C) to reduce the thermal draft risk at foot level. In this case, the total flow rate was 3x10^{6} cfm (1,416 m^{3}/s), 1.35x10^{6} cfm (637 m^{3}/s) to the grand stands, 1.35x10^{6} cfm (637 m^{3}/s) to the infield, and 3x10^{5} cfm (142 m^{3}/s) to the partial air curtain. The total flow rate in this case was the same as that in the base case and Case 3, but higher than that in Case 4 (2.3x10^{6} cfm or 1,085m^{3}/s), as shown in Table 1. Case 5 also uses a narrower air supply diffuser for the air curtain to increase the supply air velocity to 2,400 fpm (12 m/s). This increase was necessary because the air supply velocity from the curtain at 1,300 fpm (6.5 m/s) in Case 4 was not strong enough to effectively block the wind generated by the cars. With all these efforts, the air quality improved greatly in Case 5 compared to that in the previous cases. However, the air temperature is still high (around 100^{o}F or 38^{o}C), as indicated in Table 2.

The results in Cases 3, 4, and 5 do not show strong temperature stratification in the auto-racing facility with the displacement ventilation. The conclusion is contradicted to that found from other displacement ventilation studies, such as Yuan et al. (1999). Therefore, *Case 6* was designed to find out why the displacement ventilation in the facility does not perform as well as that in other indoor spaces. The major difference between this facility and other indoor spaces is the moving cars. To isolate the impact of the moving cars, Case 6 was modeled to assume that all the cars stop in their tracks while their engines still run at full capacity. All other conditions between Cases 5 and 6 remained the same. Under these conditions, the results show fully the flow characteristics of displacement ventilation. The air is much cleaner, and the air temperature in the occupied areas is lower than in the previous cases. For example, the air temperature in Case 6 is 4 to 14^{o}F (2 to 8^{o}C) lower than that in Case 5. Therefore, the wind generated by the moving cars was shown to stir the air in the facility, turning the stratified flow of a typical displacement ventilation system into a near mixing condition.

The above results indicate a trade-off between the overhead duct system and the displacement ventilation system. The overhead duct system can use a lower supply air temperature so that the total amount of air supply may be smaller than that in the displacement ventilation system. The reduction in air supply implies a smaller duct size or lower initial costs. The displacement ventilation system can improve the indoor air quality, but with a higher thermal draft risk. Therefore, the CFD analysis suggests that a careful design of the displacement ventilation system to supply lower temperature air would be very beneficial. This study would require either a mockup or a detailed CFD simulation of the airflow for a few seats in the bleacher. In both the mockup study and CFD simulation, the flow domain studied will be limited to a small area in order to obtain detailed information. Then, the difficulty is how to simulate accurately the wind from the cars. A possible solution in the mockup study of the partial flow in a few seats is to use a fan to simulate the wind. The CFD simulation could use the airflow predicted in the full-scale CFD computation as the boundary conditions for the partial flow boundary. The detailed study by either mockup or CFD has not yet been conducted for the project.

The current available information from the literature suggests an air supply temperature of 65^{o}F (18^{o}C) for a displacement ventilation system and 50^{o}F (10^{o}C) for an overhead duct system. The higher supply air temperature of this displacement ventilation system would be much less likely to lead to an uncomfortably cool thermal draft effect. Our preference is to develop further from Case 5 to decrease the overall average supply air temperature, while increasing the temperature at the displacement ventilation system air outlets. For example, the air supplied to the air curtain can be conditioned to 50^{o}F (10^{o}C) to carry a greater portion of the cooling load. The total air supply may also need to be increased to more than 3x10^{6} cfm (1,416 m^{3}/s).

Since initial cost is a major factor in the determination of the ventilation system type, increasing the overall ventilation rate may not be the optimal solution from an economic standpoint. Through a number of CFD runs, the present investigation identified *Case 7* as a more practical design. In Case 7, the grandstand uses the overhead duct system with a supply air temperature of 50 ^{o}F (10^{o}C) and a supply flow rate of 1.5x106 cfm (707 m^{3}/s). The infield uses the displacement ventilation system with a supply air temperature of 65^{o}F (18^{o}C) and a supply flow rate of 1.2x10^{6} cfm (566m^{3}/s). A partial air curtain with a flow rate of 3x10^{5} cfm (142 m^{3}/s) and a temperature of 50^{o}F (10^{o}C) is used in the northwest corner of the facility. Table 1 compares the ventilation systems and supply air parameters of Case 7 versus those in the other cases.

Figures 4(b) and 5(b) show the distributions of the air temperature and lead concentration, respectively. Case 7 provides a relatively low lead level. Most importantly, the air temperature in the facility is moderate. The temperature in the grandstands, the infield, and the northwest corner is 88, 90, and 95°F (31, 32, and 35°C), respectively. Although the temperature in the northwest corner is the highest, the air velocity in that area is also the highest. The overall thermal comfort level is rather uniform in the occupied zones. The temperatures are a little higher than normal comfort standards. However, considering the temperature is for an "outdoor" shaded space and the case is for the worst summer scenario, the thermal comfort should be accepted by spectators. As the planning process for the project evolved concurrently with these early CFD studies, it became apparent that major racing events would not likely be scheduled for an indoor facility during peak summer conditions, when outdoor racing events are very popular. This will allow the design team to significantly reduce the size and capital cost of the central refrigeration plant, while refining the HVAC system to further improve the comfort conditions within the facility during off-peak months.

The efforts described above were aimed at removing the excessive heat and contaminants generated by an auto-racing event in summer, a situation that presents the worst-case scenario for cooling. On the other hand, the facility will also be used year-round for many other events, such as conventions, trade shows, consumer expos, entertainment, and other recreational events. Therefore, there is also a worst-case scenario for winter heating, where the facility has little internal heat gains but has a high heat loss through the building envelope due to the severe winter weather in Pittsburgh, PA.

The worst-case cooling scenario assumed all 45 cars running on the tracks in summer design conditions under a steady state. However, in the worst-case heating scenario, it is important to know how long it takes to heat up the facility from a cold, unoccupied condition, and how comfortable the condition is after occupants enter the facility under the winter design conditions. Therefore, two cases must be investigated for the worst-case heating scenario: an unoccupied warm-up period (Case 8) and an occupied scenario with little heat gains (Case 9).

This design assumed an event with minimum occupants and lighting levels as the worst-case heating scenario: a concert with only 31,800 people gathered in the central part of the infield. Each person occupies 6 ft^2 ($0.6m^2$) floor area. General lighting is provided only to the central part of the infield. The desired preheat temperature in the occupied zone (prior to the actual event occupancy) is 55°F (13°C), and the winter outdoor design temperature is 7°F (-14°C).

Case 8, the warm-up mode for preheating the space prior to a winter event, assumes zero heat gains from the occupants, lighting, etc. The initial wall, roof, and indoor air temperature are assumed to be the same as the outdoor design temperature (7°F or -14°C). Case 8 assumes only the overhead duct system from Case 7 is used to heat the grandstands. The supply airflow rate from the ventilation system is $2x10^6$ cfm (944 m^3/s) at a temperature of 86°F (30°C). These values were determined manually based on heat transfer principles. With a perfect mixing and static assumption, the manual calculation estimates that it takes about two hours to preheat the indoor air to 55°F (13°C). The CFD simulation was used to test the accuracy of the manual calculation. The CFD results show that the warm-up period can actually be more than three hours, as shown in Figure 7. This is because the warm air will heat up the indoor air in the upper zone first, due to thermal buoyancy effect. The warmer air in the upper part of the space will increase the heat loss through the roof. The average air temperature was found to be much higher than that in the occupied zone due to the thermal stratification of the indoor air.

For the occupied winter scenario (Case 9), the CFD analysis shows that it is better to use the displacement ventilation in the infield. With the supply airflow rate for each person of 15 cfm (7 L/s) and the supply air temperature of 65^{o}F (18^{o}C), the ventilation system can maintain an air temperature of 68^{o}F (20^{o}C) in the occupied zone, which is quite acceptable for the occupied condition.

With the design improvements suggested through the use of CFD modeling of the worst-case cooling and heating scenarios, the ventilation systems should be able to provide a comfortable indoor environment for all other events. Nevertheless, the design of the ventilation systems has not been completed, because the architectural design has not been finalized. As with many other projects, the design may continue to evolve until construction begins. The team is continuing to improve the design.

CONCLUSIONS

This paper demonstrates how to use a CFD program to assist in the design of a ventilation system for an indoor auto-racing complex. The large-scale indoor space and the high speed of the racing cars impose special challenges to the ventilation system design and the CFD simulations. The CFD results are very useful in making a number of design decisions. Some of the major conclusions from the CFD analysis are as follows:

For the worst-case summer cooling scenario for an auto-racing event, the CFD analysis found that the use of a full-length air curtain is not totally effective in isolating the spectators from the fumes generated by the cars, due to the strong wind induced by the cars. It is more economical and effective to use a partial air curtain, of lower supply air temperature and higher velocity, in the northwest corner of the complex. The displacement ventilation system can provide a better indoor air quality and a lower air temperature in the occupied zones than the overhead duct ventilation system. However, the low air supply temperature from the displacement ventilation system may impose thermal draft risk, which needs to be further studied. The study found that the flow rate and air temperature to different parts of the indoor space should vary to satisfy the thermal comfort requirements at different zones, according to the ventilation systems used.

For the worst-case winter heating scenario for a concert event, it would take more than three hours to heat up the space with a supply air temperature of 86^{o}F (30^{o}C). The heat up period is longer than that calculated with mixing assumptions. This is because the indoor air is highly stratified. It is difficult to heat up the air in the occupied zone, due to the thermal buoyancy effect. For the concert event, the displacement ventilation performs better than the overhead duct ventilation system.

REFERENCES

ASHRAE. 1992. *ASHRAE Standard 55-1992: Thermal Environmental Conditions on Human Occupancy*, Atlanta: ASHRAE.

CHAM, 1999. PHOENICS Version 3.1, London: CHAM Ltd.

Chen, Q., and Moser, A. 1991. "Simulation of a multiple-nozzle diffuser," *Proc. of 12th AIVC Conference*, Vol. 2, pp. 1-14.

Guthrie, A., Ikezawa, H., Otaka, K., and Yau, R.M.H. 1992. “Airflow studies in large spaces - a case study of an airport passenger terminal building,” *International Symposium on Room Air Convection and Ventilation Effectiveness*, Tokyo, Japan, pp. 517-523.

Heiss, A. 1987. "Numerische und Experimentelle Untersuchungen der Laminaren und Turbulenten Konvektion in Einem Geschlossenen Behatter," *Ph.D. Dissertation*, Technische Universitat Muenchen.

Moser, A., Off, F., Schalin, A. and Yuan, X. 1995. “Numerical modelling of heat transfer by radiation and convection in an atrium with thermal inertia,” *ASHRAE Transaction*, 101(2): 1136-1143.

Nielsen, P.V. 1995. “Airflow in a world exposition pavilion studied by scale-model experiments and computational fluid dynamics,” *ASHRAE Transactions*, 101(2): 1118-1126.

Olson, D.A. and Glicksman, L.R. 1991. “Transient natural convection in enclosures at high Rayleigh number,” *Journal of Heat Transfer*, 113: 635-642.

Patankar, S.V. 1980. *Numerical Heat Transfer and Fluid Flow*, New York: Hemisphere Publishing Corp.

Versteeg, H.K. and Malalasekera, W. 1995. *An Introduction to Computational Fluid Dynamics*, London: Longman.

Wilcox, D.C. 1993. *Turbulence Modeling for CFD*, La Canada: DCW Industries, Inc.

Yakhot, V., Orzag, S.A., Thangam, S., Gatski, T.B., and Speziak, C.G. 1992. “Development of Turbulence Models for Shear Flows by a Double Expansion Technique,” *Physics Fluids A*, 4: 1510-1520.

Yang, C., Demokritou, P., Chen, Q., Spengler, J.D., and Parsons, A. 2000. “Ventilation and air quality in indoor ice skating arenas,” *ASHRAE Transactions*, 106(2).

Yuan, X., Chen, Q., Moser, A. and Suter, P. 1992. “Numerical simulation of air flow in gymnasia,” *Indoor Environment*, 1(4): 224-233.

Yuan, X., Chen, Q., and Glicksman, L.R. 1999. “Performance evaluation and design guidelines for displacement ventilation,” *ASHRAE Transactions*, 105(1): 298-309.

Table 1. The ventilation system and the air supply parameters used

Case		Grandstands			Infield			Air Curtain		
		Flow (mcfm /m^3/s)	Supply Method	Supply Temp. (^{o}F/^{o}C)	Supply Method	Flow (mcfm /m^3/s)	Supply Temp. (^{o}F/^{o}C)	Curtain length	Flow (mcfm /m^3/s)	Temp (^{o}F/^{o}C)
1	BaseCase	1.0/472	Duct	50/10	Disp	1.0/472	50/10	Full	1.0/472	86/30
2	NoCurtain	1.0/472	Duct	50/10	Disp	1.0/472	50/10	No	N/A	N/A
3	Displace	1.0/472	Disp	50/10	Disp	1.0/472	50/10	Full	1.0/472	86/30
4	PartCurtain	1.0/472	Disp	50/10	Disp	1.0/472	50/10	Partial	0.3/142	86/30
5	Disp18C	1.35/637	Disp	65/18	Disp	1.35/637	65/18	Partial	0.3/142	86/30
6	CarStop	1.35/637	Disp	65/18	Disp	1.35/637	65/18	Partial	0.3/142	86/30
7	Mixed	1.5/708	Duct	50/10	Disp	1.2/566	65/18	Partial	0.3/142	50/10
8	WinHeat	2.0	Duct	86/30	No	N/A	N/A	No	N/A	N/A
9	Concert	No	N/A	N/A	Disp	0.477/225	65/18	No	N/A	N/A

Table 2. The air temperature computed by CFD for different occupied zones ($^oF/^oC$)

Case		Grandstands	Infield	Northwest Corner
1	BaseCase	95/35	97/36	107/42
2	NoCurtain	100/38	106/41	111/44
3	Displace	91/33	95/35	99/37
4	PartCurtain	95/35	99/37	100/38
5	Disp18C	95/35	100/38	100/38
6	CarStop	91/33	90/32	86/30
7	Mixed	88/31	90/32	95/35

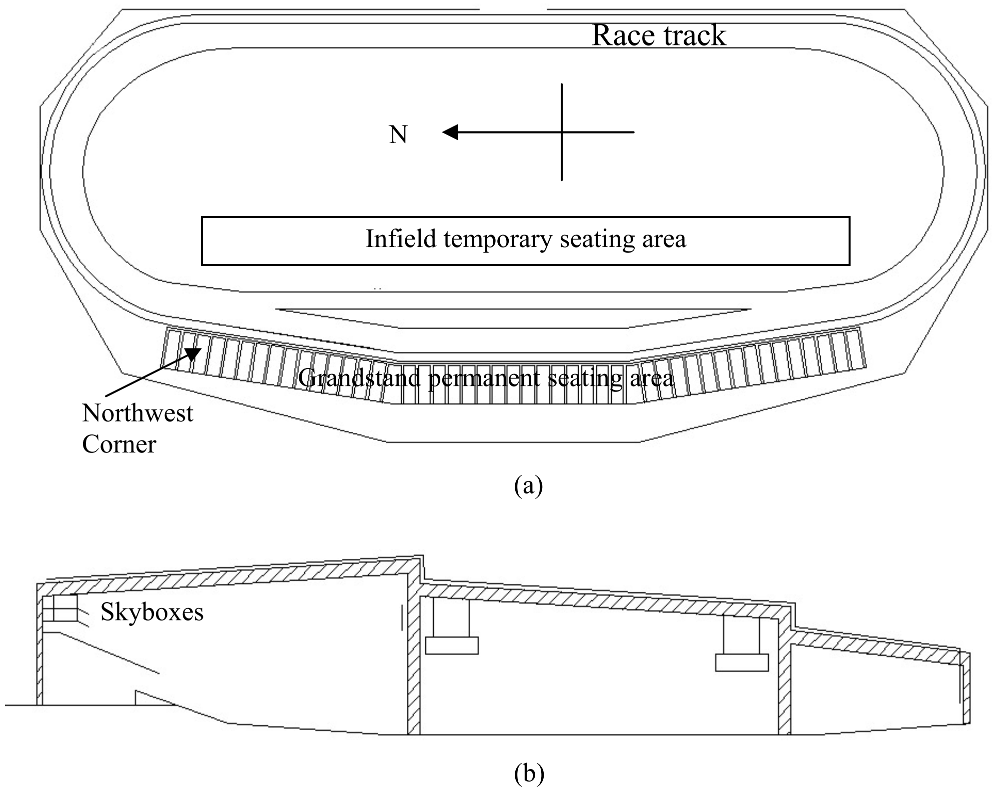


Figure 1. The architectural blueprint for the auto-racing complex: (a) plane and (b) middle section.

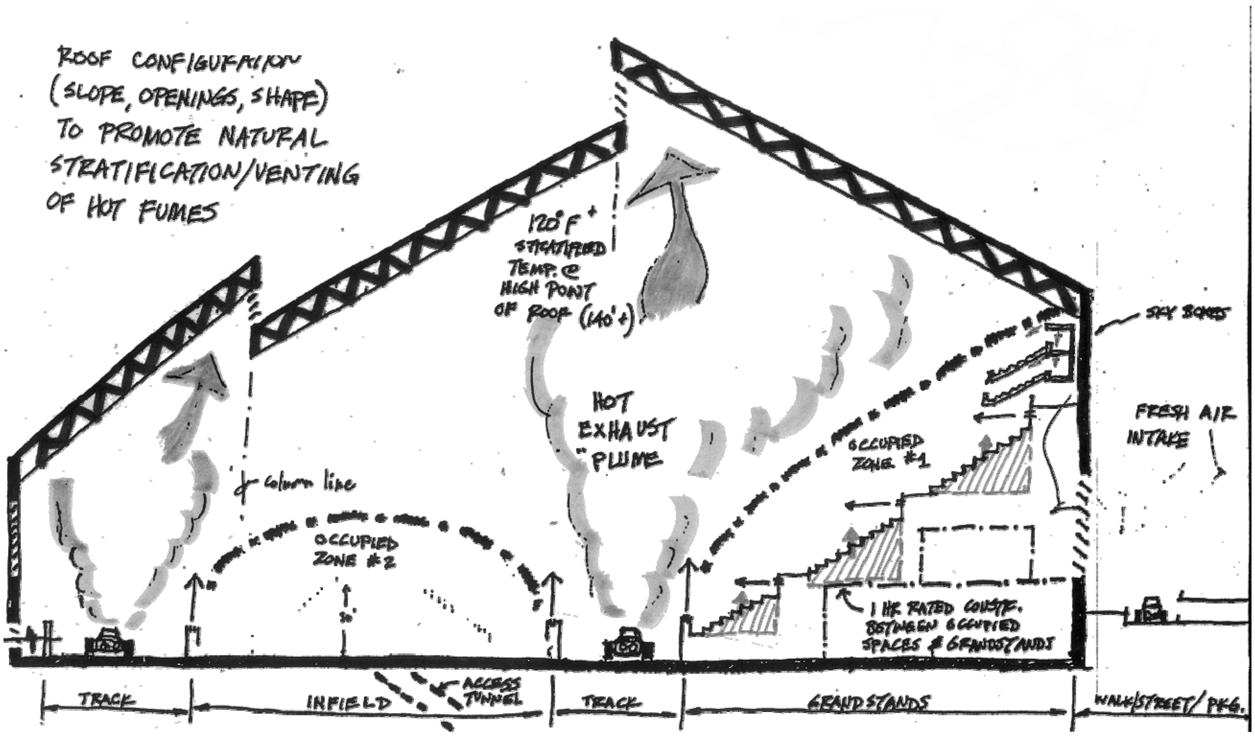
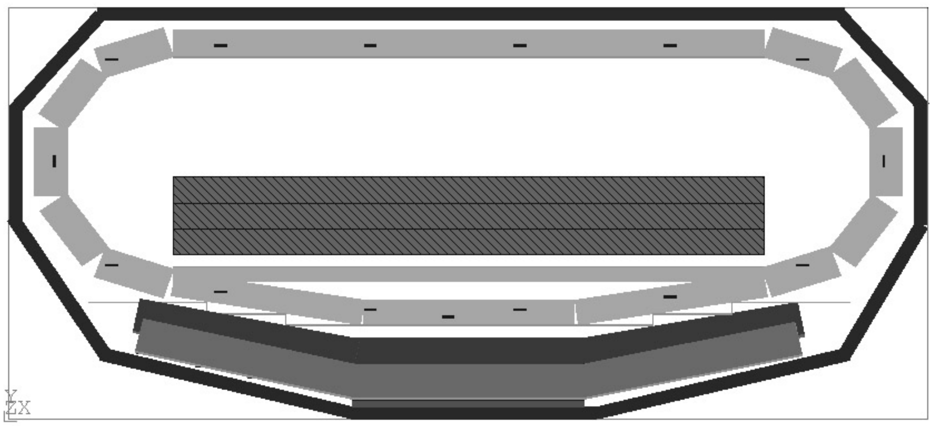
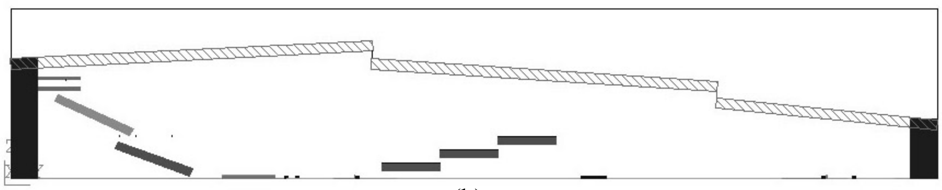


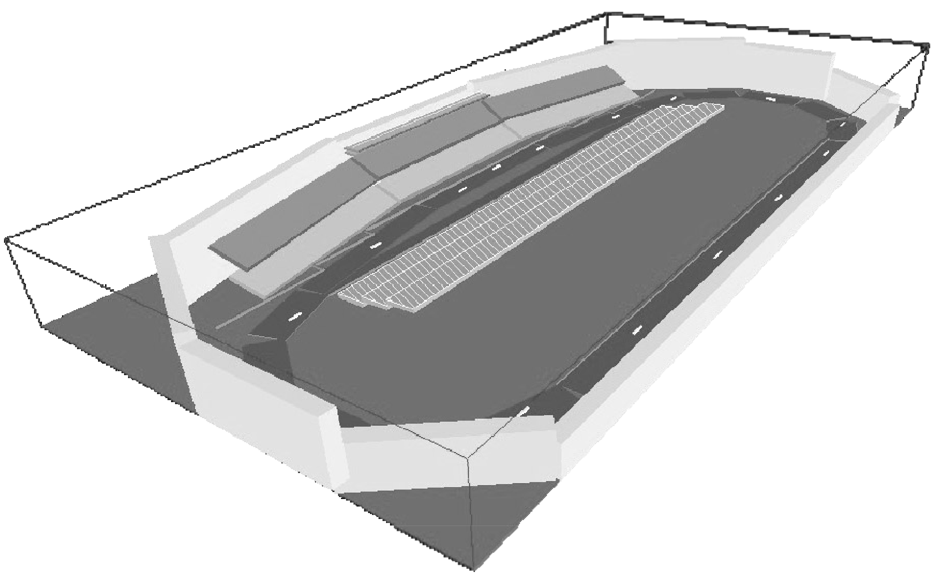
Figure 2. Illustration of the ventilation strategy



(a)

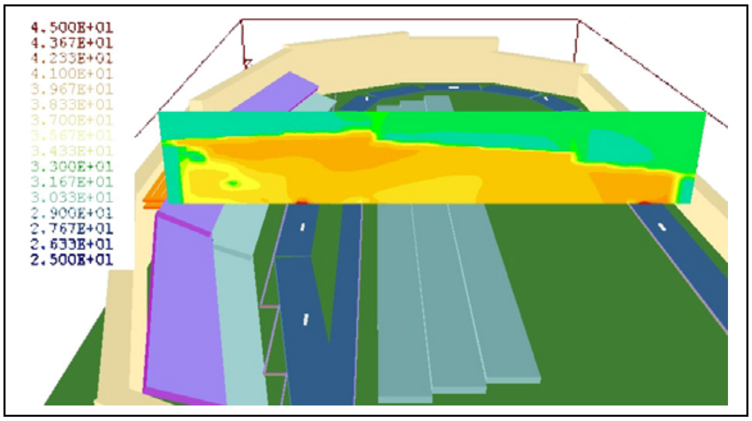


(b)

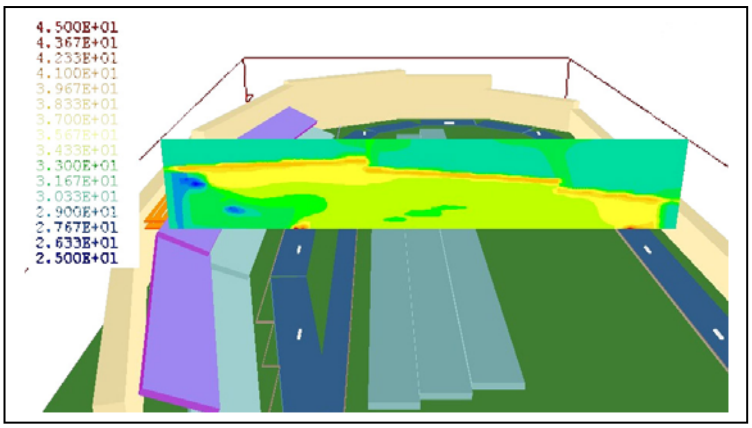


(c)

Figure 3 The CFD model for the auto-racing complex

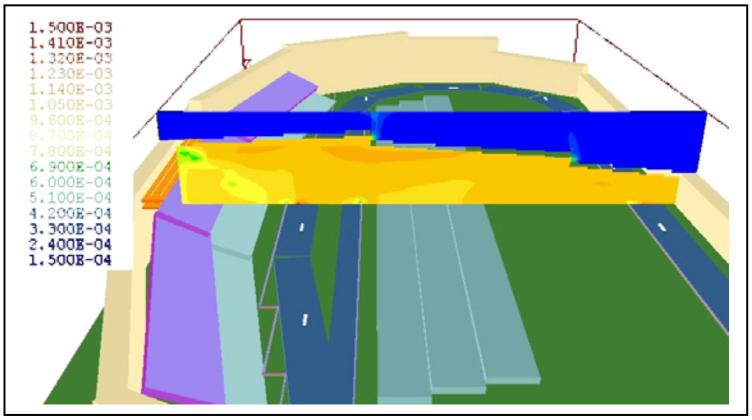


(a)

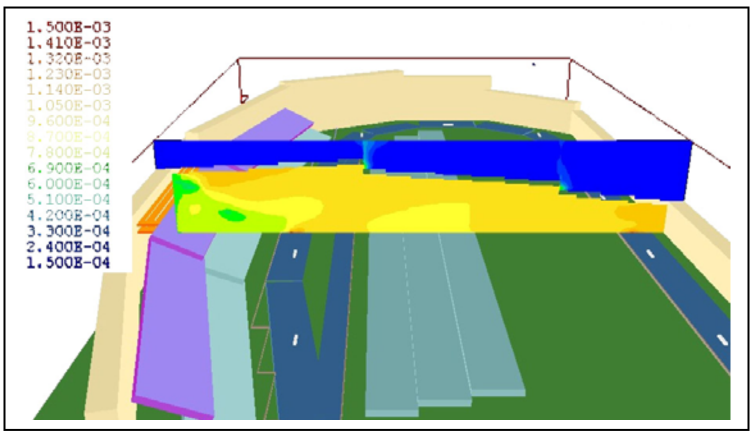


(b)

Figure 4. The air temperature distribution in the middle section: (a) Case 1 and (b) Case 7 (unit: 8C)

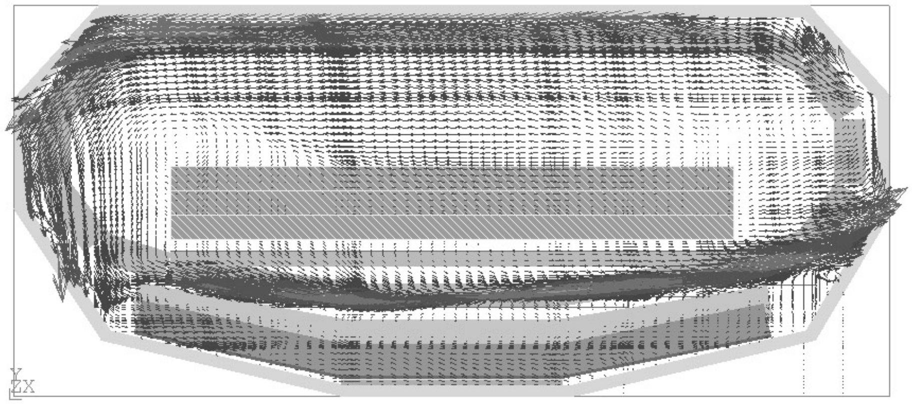


(a)

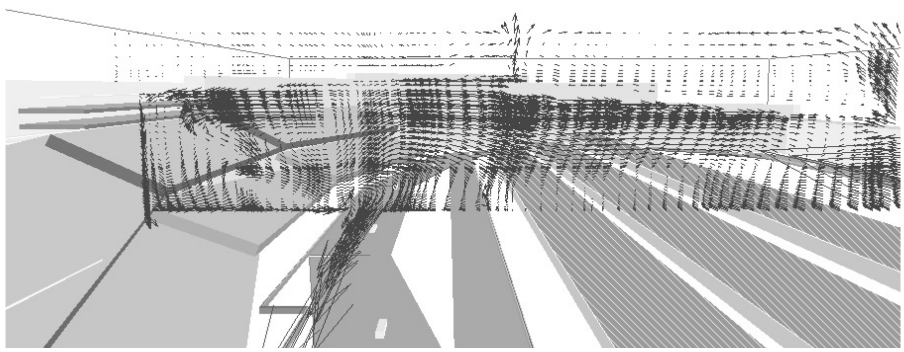


(b)

Figure 5. The lead concentration distribution in the middle section: (a) Case 1 and (b) Case 7 (unit: gLead/kgAir)



(a)



(b)

Figure 6. The air velocity distribution for Case 1 (a) at 16 ft (5 m) above the tracks and (b) at the middle section near the grandstands

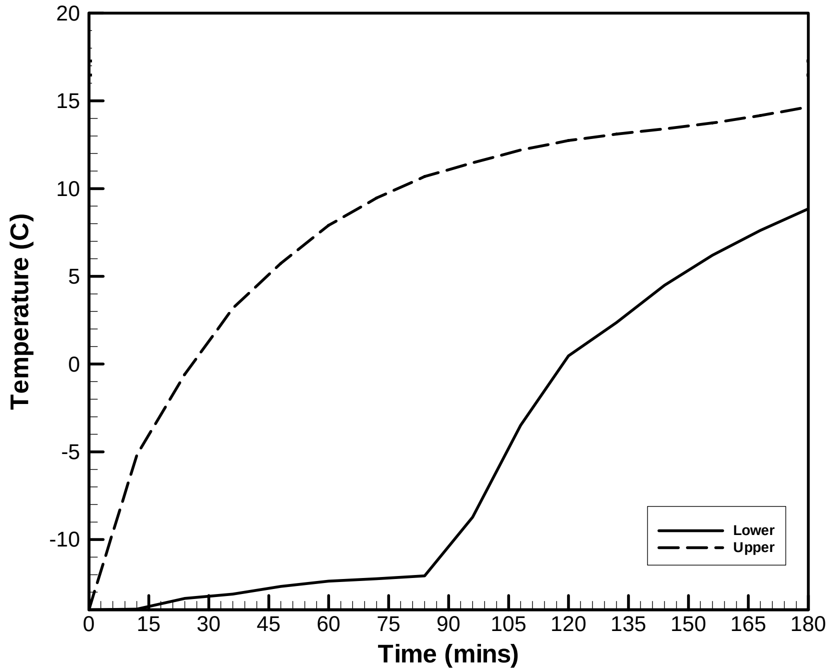


Figure 7. The air temperature in different parts of the indoor space for the winter heat up case (Case 8).