AIRFLOW AND TEMPERATURE FIELD CALCULATIONS FOR WINTER SPORTS FACILITIES

Andrea Frisque*

Rowan, Williams, Davies & Irwin (RWDI), Vancouver, BC, V5Z 1K5, Canada**

Abstract

In the design of indoor winter sports facilities Computational Fluid Dynamics (CFD) simulations are used to calculate the velocity and temperature distribution throughout the space, in order to complement traditional mechanical design and increase confidence into the proposed design. This process is described here using the example of a competitive curling rink. In the introduction the capabilities and limitations of CFD simulations are briefly lined out. The physics of the model of the curling venue are described. Simulation results are presented for the velocity distribution and the temperature distribution for two different heat gain scenarios. The first scenario includes only heat gain from lights and represents the maximum potential to pre-cool the space before major events. The second scenario includes heat gains from lighting and people at maximum occupancy. The simulation results confirm that the design intent of uniformely low air speeds and temperatures can be expected to be achieved.

Keywords

Computational Fluid Dynamics, Sports Venue, Stratification

Introduction

In the present study Computational Fluid Dynamics (CFD) simulations are used to support the design of a competitive curling venue. The performance of the ventilation system and the resulting airflows and temperatures strongly influence the sporting conditions in indoor winter sports facilities such as curling rinks, speed-skating ovals, hockey rinks, etc. Parameters that are affected include the quality of the ice surface, comfortable temperatures and airflow conditions. Temperatures and airflows above the ice surface should be evenly distributed in order to support the ice maker in providing a homogeneous, high-quality ice surface. Temperatures in the occupied areas above the ice should also be within the comfort range for the particular sport. Airflows should not be noticeable, and clearly airflow conditions that could be perceived as "head winds" or similar conditions need to be avoided. At the same time sufficient amounts of fresh air need to be provided to the space above the ice surfaces to maintain satisfactory air quality, remove humidity and avoid fogging. Besides providing sporting conditions that support the competitors, comfortable thermal conditions for the spectators need to be provided.

CFD has been used before to design competitive indoor winter sport facilities, for example for the ice hockey stadium in Turin [Chown J. 2003.]. The contributions of a CFD study to the design process consists of resolving the velocity and temperature field within a space as opposed to the assumption of homogeneous, or "perfectly stirred" HVAC zones. On many occasions ventilation systems are designed based on the assumption of homogeneous zones, potentially complemented with heuristic relations for airflow pattern, air velocities, stratification an such like. For a wide range of ventilation design situations this approach is appropriate and sufficient, particularly for space configurations for which a large basis of experience exists. For uniquely shaped, large open spaces with high performance requirements, such as indoor winter sport facilities, more detailed information is desirable. CFD calculations are able to provide more detail by numerically solving the field equations for velocity (or more precisely momentum) and temperature given by the Navier-Stokes equations describing conservation of momentum and the energy conservation equation. For more information refer to a fluid dynamics text book such as [Smits AJ. 2000.]. CFD simulations are not uniquely used in building simulations but generally describe the behaviour of fluids. To list a few examples, they describe aerodynamics of cars and airplanes as well as the gas flow through gas turbines and internal combustion engines. However, a large challenge to CFD calculations arises from turbulent flows. Airflows in buildings are virtually always turbulent due to the large dimensions. To describe turbulent flows turbulent models need to be used, which severely limit accuracy of the solution. For an overview of turbulence models see [Pope. 2000.]. Using turbulence models, the velocity and temperature field are typically expressed in statistical quantities, such as average velocities and temperatures and fluctuations. These limitations need to be considered when interpreting and relying on CFD simulation results for the prediction of airflows and temperatures in building spaces. Further limitations result from the fact that the simulations describe a simpli-

* andrea.frisque@rwdi.com

** The author performed the studies described here as an employee of Stantec, Vancouver, BC, V6B 6A3, Canada, and gratefully acknowledges Stantec's support.

fied model situation. It is important to realize that there is a wide range of additional parameters that will affect the airflow in the real sports facility, such as for example open doors.
In the present paper the use of CFD in support of the design of the ventilation of a indoor winter sports venue is shown using the example of a large, competitive curling venue. The particular requirements relating to the sporting competition investigated by the present CFD study are:

- Uniform air velocities and temperatures over the ice surface.
- Low air velocities: people usually do not feel air moving at 0.4 m/s or less. For comparison, walking speed is around 4 – 6 km/h corresponding to 1.1 – 1.7 m/s (= 2.5 – 3.7 m/h).
- Low temperatures (15°C).

A further question relates to the implementation of the design intent. It is intended to supply the air overhead to the spectators, draw the air through the spectator ranges removing the heat and moisture produced by the spectators before they enter the space and extract the air underneath the seating. The question arises, whether it would be necessary to distribute the extraction points underneath the seating using ducts, or whether it is sufficient to provide air extraction through louvers in the walls of the space in order to produce the desired airflow pattern.
In the following sections the model of the curling venue used in the CFD simulations is described, the simulation results for velocity and temperature distribution are presented and discussed, and finally conclusions are drawn.

Model

A description of an internal airflow model consists of the geometry of the space, the boundary conditions for velocity and temperature, and the physics models for turbulence, buoyancy etc. In the case of the present example the geometry of the space was modelled in StarDesign software based on specifications from the architect and the mechanical engineer. The model for the example of the curling venue is shown in Figure 1. The overall geometry of the space including subdivisions and the major ducts, rows of spectators, and boards around the ice-surface are included in the model. Generally, the diffusers, through which the air is supplied to the space, are located at the sides of the ducts facing the spectators and below the ducts facing down. On the duct supplying air to the smallest of the spectator sections the diffusers are angled at 45° toward the spectators, with blades arranges in such a way that one third of the air is directed horizontally, on third is at 45° and one third is directed downward. The locations of air extraction are below the spectator seating. In order to reduce the level of effort and the potential for complications when building a mesh and converging a solution, the level of detail needs to be kept to a minimum.

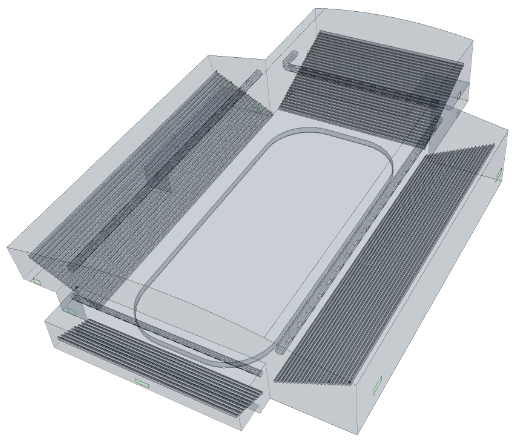


Figure 1: Model of the interior space.

The geometry model is the base for a computational mesh subdividing the air volume inside the sports venue. This mesh was build using Star-CCM+ software. A range of different kind of mesh types with different cell geometries exist, for example tetrahedral, hexahedral, polyhedral. For the present analysis a trimmed hexahedral mesh is used, which was found to converge better for the present example than more complex cell shapes. The mesh was refined at the boundaries and over the ice, where a large temperature gradient exists. The mesh subdivides the large volume of air in the sports venue into many small “finite volume” cells that can be viewed as a refinement over the common zonal space models. In mathematical terms the mesh provides a spatial discretization for the numerical solution of the differential equations describing the continuous development of the velocity and temperature field inside the sports venue. The physics of the airflow and temperature distribution in the sports venue are represented through the realizable k-ϵ turbulence model [Shih TH et al. 1994.] for an isovolumetric (constant density) flow. Buoyancy is represented through the Boussinesq approximation with a thermal expansion coefficient of the air of $1/T$, with T being the temperature, approximating the air as an ideal gas.
In the present example, air is distributed to the space through diffusers. The angle of the diffuser blades is represented in the model through the direction of the air entering the space. The spacing

between the blades is considered through the turbulent length scale at the air inlet locations. The boundary for air extraction is a simple outflow condition, where the amount of air entering the space is at all times balanced by the amount that leaves the space. The relative ratios of how much air is extracted through which location is chosen in such a way that the individual spectator areas are balanced.

For the simulations in which people are considered to be present, heat is released into the space from the surface of the geometrical elements representing the spectators. Heat representing lighting above the ice is introduced through the ceiling. The ice surface is kept at a constant temperature. All other surfaces are adiabatic.

Simulation Results and Discussion

Three questions are addressed with the following simulations:

- What level of air velocities should be expected in the space, particularly within the 2 m directly above the ice?
- In order to evaluate the pre-cooling potential of the space, what are the lowest temperatures that can be achieved within the space while the lighting is on, without relying on heat loss through the envelope?
- What is the warmest condition that should be expected in the space when fully occupied?

Steady state CFD simulations are used to examine these issues. In the following sections first the results for the velocity magnitudes are presented, which are only negligibly affected by the different temperature scenarios. Secondly, the results for the temperature distribution in the pre-cooling condition are shown, and finally the results for the temperature distribution in the warmest condition.

When interpreting turbulent CFD results it is important to keep in mind the implications of the particular turbulence model used. The results that are presented below are the results of a CFD calculation using a k-ϵ turbulence model. This means that the results for velocities and temperatures represent average values, not instantaneous values of the turbulent flow. As defined by the Taylor decomposition [Pope. 2000.] the instantaneous value is the sum of the average value and the turbulent fluctuation.

Air Velocities

The results for the air velocity distribution within the space are found to not be influenced by the different temperature scenarios that were considered. In the case of winter sports facilities, with ice on the floor level and lights at the ceiling, no instabilities, as they would be expected in the reverse case with a hot plate at the bottom, are present.

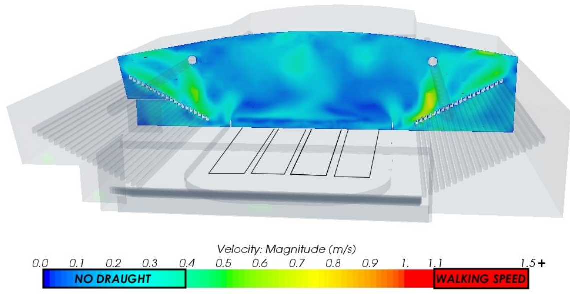


Figure 2: Velocity magnitude on an east-west section.

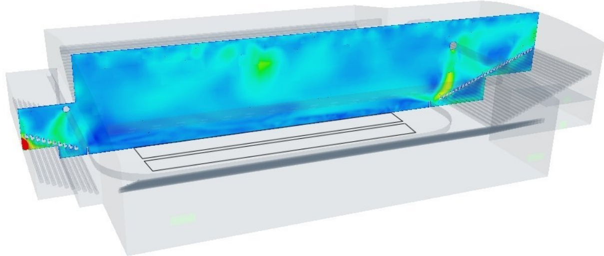
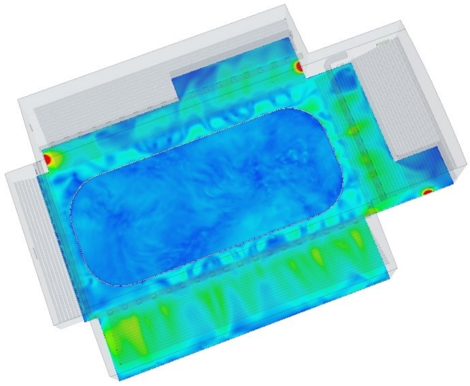
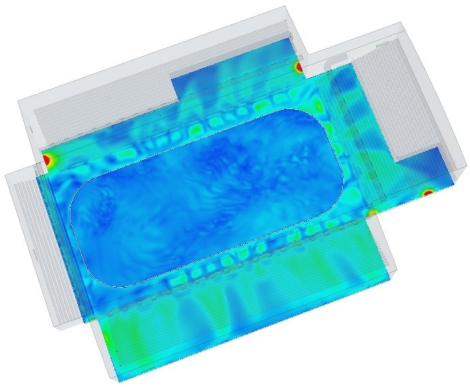


Figure 3: Velocity magnitude on a north-south section. The colour coding is identical to Figure 2.

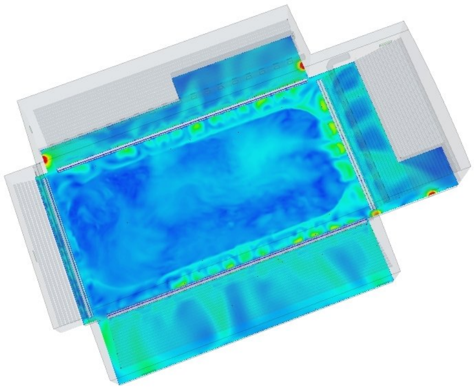
In Figure 2 the velocity magnitudes on an east-west section through the middle of the space is shown, and in Figure 3 for a north-south section through the middle of the space. In those images it can be seen how the air enters the spaces from the diffusers at higher speed and quickly slows down as it spreads out and mixes with the air in the space. The air is drawn through the spectator rows and extracted below the seating. The confinement of the walls of the space force the air to move in large eddies; the supply air induces recirculation zones. It can be seen that the design is successful in ensuring low velocities in the space. The majority of the sections shown in Figures 2 and 3 show velocities below 0.4 m/s. The high velocity that can be seen indicated in red in Figure 3 is a result of the air accelerating close to an extraction location.



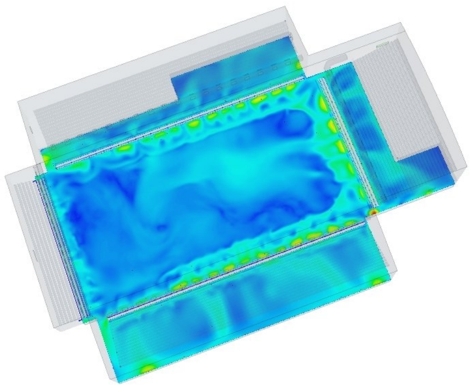
(a) 0.5 m



(b) 1.0 m



(c) 1.5 m



(d) 2.0 m

Figure 4: Velocity magnitudes on horizontal planes at different elevations. The colour coding is identical to Figure 2.

In order to provide satisfying conditions for sport competitions, the air movement in the occupied zone over the ice is particularly important and needs to be kept to a minimum. In Figure 4 four planes at 0.5 m, 1.0 m, 1.5 m and 2.0 m heights above the ice are shown. It is clearly visible that the velocity magnitudes are evenly distributed over the surface of the ice. The ice surfaces is surrounded by boards 1 m in height. The influence of the boards on the air velocities can clearly be seen. With increasing elevation the air velocities slowly increase, however, they remain very low. The velocities exceed 0.4 m/s only close to the diffusers or to the extraction locations. The areas of high velocities indicated in red are again located at the air extraction points.

Temperature Distribution

The supply air does not perfectly stir the air in the rather large space, but instead there are different temperatures in different areas of the space. Particularly, colder air pools above the cold ice surface and warm air rises to the top of the space. The temperature distribution within the space is a result of the supply of cold air to the space, producing the airflow pattern and velocities as shown above, combined with heat from lighting and people, as well as the cold ice surface.

In Figures 5 and 6 the vertical temperature distribution in the space is shown for the case where only the lighting is adding heat to the space, no people are present. The sections shown in the figures are the same sections as presented in the velocity results above. There is an area of cold air above the ice and an area of warm air at the ceiling, where in the present model the heat of the lights enters the space. As this is a steady state simulation, the results represent the situation that one can achieve after the cold air has been supplied for such a long time that changes to the state inside the space no longer occur and the result becomes independent from the initial condition. In this sense the results represent the maximum pre-cooling that can be achieved for the curling venue. It can be seen in the figures that the temperatures are fairly low, so this condition may not be desirable in actual operations of the space. However, the results show that the design provides significant potential for pre-cooling.

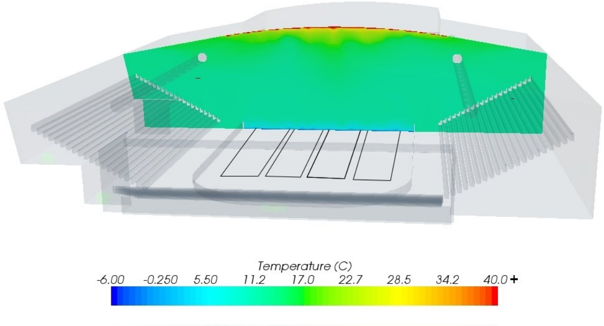


Figure 5: Start-of-Game Condition: temperature on an east-west section.

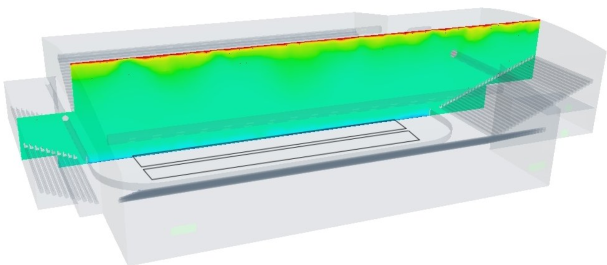
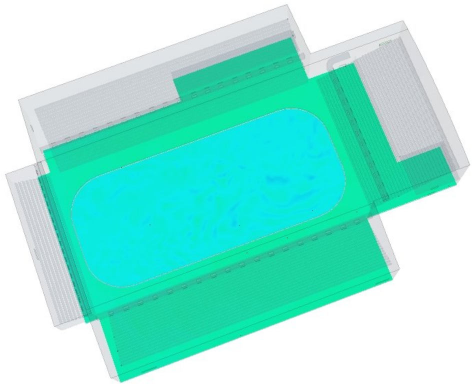


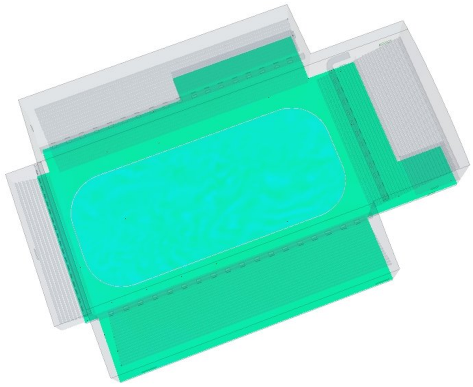
Figure 6: Start-of-Game Condition: temperature on a north-south section. The colour coding is identical to Figure 5.

Figure 7 shows horizontal temperature distributions on planes above the ice at 0.5 m, 1 m, 1.5 m and 2 m height. On all levels the temperature distribution over the area of the ice is homogeneous. It can be seen how the boards around the ice field contain a pool of cold air above the ice. As elevation increases, the temperature over the ice rises as well and eventually there is no longer an influence of the boards.

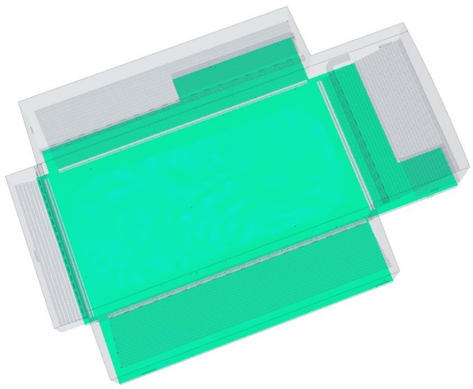
The same simulations as above were repeated for a fully occupied venue. The resulting vertical temperature distributions are shown in Figures 8 and 9. As expected, it can be seen that in general temperatures within the space are higher then they were in the pre-cooling condition described above. Again the results presented in the figures are results of steady state simulations. Therefore they represent the warmest condition that can be expected in the space, after all pre-cooling that might have been present is used up. Compared to the pre-cooling condition, the temperature stratification in the space is more pronounced. However, the temperatures within one meter above the ice remain unchanged. It can be seen even more clearly that the boards around the ice surface contain a pool of cold air. As well the results show that the location and orientation of the air supply as well as the air extraction points is successful at containing the heat from the spectators. The air is drawn through the spectator ranges carrying the heat with it directly to the extraction points.



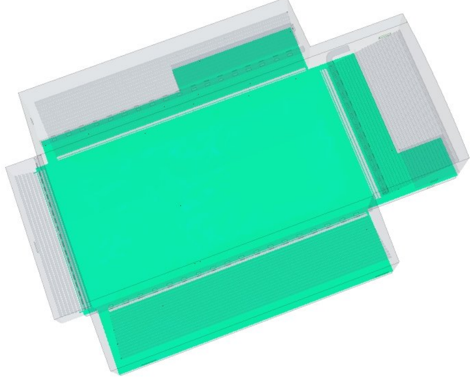
(a) 0.5 m



(b) 1.0 m



(c) 1.5 m



(d) 2.0 m

Figure 7: Start-of-Game Condition: temperature on horizontal planes at different elevations. The colour coding is identical to Figure 5.

In general, the heat does not enter the centre areas over the ice. The vertical air supply to the front rows of the spectator ranges effectively provides an air curtain separating the air volume in the centre of the space from the spectator areas. In additional simulations not presented here it could be seen that, if the vertical air distribution is moved too far inside the spectator ranges, hot air can escape from the spectator ranges and be entrained into the centre of the venue. In Figure 10 the horizontal temperature distribution over the ice is shown on planes above the ice at 0.5 m, 1 m, 1.5 m and 2 m height. The pool of cold air above the ice within 1 m above the ice is clearly visible. As the elevation above the ice increases, the cold air increasingly mixes with the surrounding air. As in the previous results, it can again be seen that the temperature distribution above the ice is homogeneous.

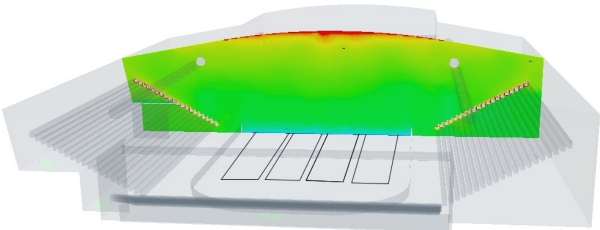


Figure 8: Warmest Condition: temperature on an east-west section. The colour coding is identical to Figure 5.

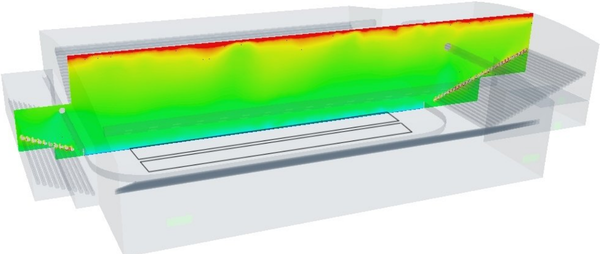
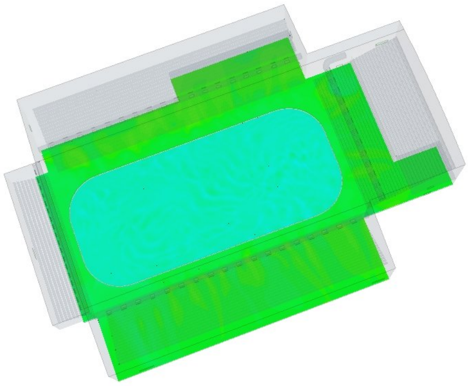


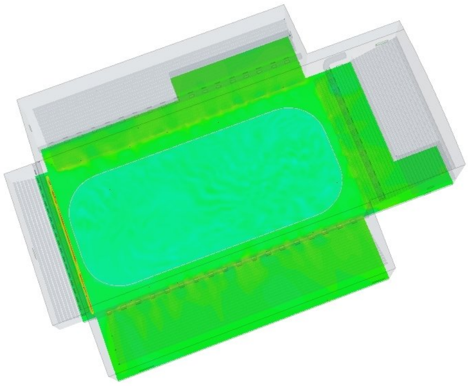
Figure 9: Warmest Condition: temperature on a north-south section. The colour coding is identical to Figure 5.

Conclusions

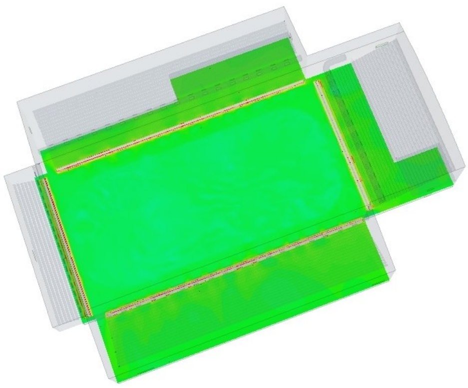
The CFD simulations presented here show how advanced airflow and temperature distribution calculations can improve confidence in the design of indoor winter sports facility. The process is described using the example of a competitive curling venue. The competition requirements that are investigated are the uniformity of air velocities and temperatures above the ice, the velocity magnitudes of the airflow, and the temperature levels.



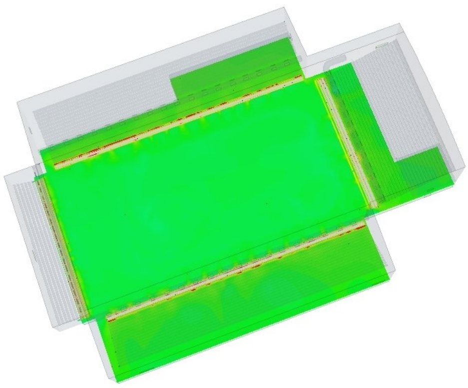
(a) 0.5 m



(b) 1.0 m



(c) 1.5 m



(d) 2.0 m

Figure 10: Warmest Condition: temperature on horizontal planes at different elevations. The colour coding is identical to Figure 5.

In order to address these issues, two different temperature scenarios have been simulated. In one case no heat contributions from people are considered, only lights are on. This represents the maximum potential for pre-cooling the space. The second scenario considers maximum occupancy and the corresponding heat contribution from people. The airflow pattern and velocities are not affected by the change in temperature scenarios.
The simulation results indicate that

- Air velocities and temperatures above the ice are uniform.
- The majority of air velocities is below 0.4 m/s.
- It is possible to pre-cool the space to approximately 12°C in the relevant regions.
- In the warmest situation, once all precooling has been used up, the temperature right above the ice remains below 10°C. It increases to about 15°C at the top of the boards surrounding the ice. Lower spectator ranges are at a temperature of approximately 17°C, rising to approximately 19°C for the upper ranges. In reality these temperatures can be expected to be lower because of ice phase changes (sublimation), heat radiated to the ice, and heat loss through the roof, which are not modelled in the present simulations.
- Air extraction ducts underneath the seating are not necessary.

As discussed in the introduction, the simulations describe a simplified model situation. Additional parameters will affect the airflow in the real curling venue in ways that are not represented in the present simulations, such as for example open doors. However, the simulation results presented here, in combination with careful traditional mechanical design, create confidence in the performance of the proposed ventilation system as intended by the design.

References

[Chown J. 2003.] Building Simulations as an Aid to Design. Eighth International IBPSA Conference, Eindhoven, Netherlands, August 11-14, 2003.

[Pope. 2000.] Turbulent Flows. New York: Cambridge University Press.

[Shih TH et al. 1994.] A New k-ϵ Eddy Viscosity Model for High Reynolds Number Turbulent Flows – Model Development and Validation. NASA TM 106721.

[Smits AJ. 2000.] Fluid Mechanics. New York: John Wiley & Sons.