Reputation Resources Results

CONSULTING ENGINEERS
& SCIENTISTS

TECHNOTES

ISSUE NO. 4

CFD ASSISTS DESIGNERS

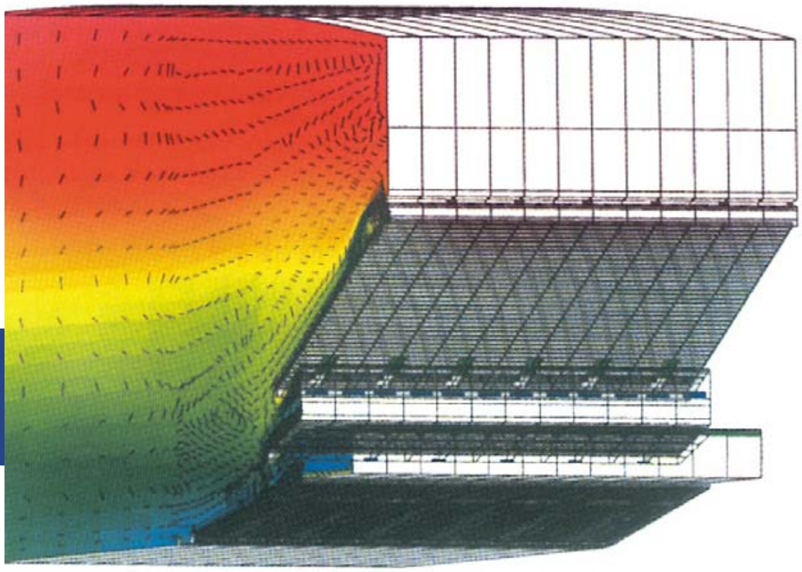
Computational Fluid Dynamics (CFD) is a computer modelling technique for simulating fluid flow. CFD can predict air speed and direction, temperature, humidity, turbulence levels, and concentrations of contaminants at thousands of points in the space to help design laboratories, cleanrooms, schools, theatres, hospitals, sports complexes, and industrial facilities. Designers can easily interpret the predicted flow patterns shown on colour images. Air quality and ventilation problems are readily identified and solutions quickly tested. Other uses of CFD include improving industrial processes, where transport of particles and chemical reactions can interact with the fluid flow.

In this issue of RWDI's newsletter, we feature typical examples of the use of CFD in design and problem solving.

VENTILATION REQUIREMENTS FOR NEW PHOENIX BALLPARK

By Scott Gamble, Principal

Bank One Ballpark, being designed for Phoenix as the home of the Arizona Diamondbacks, will be a state-of-the-art facility incorporating a number of recent technological developments in professional sports facilities. Most notable will be the use of a retractable roof system to provide shelter from sun and rain during events. The ballpark will also have a natural grass playing surface.



Vertical slice through seating areas and playing field of stadium showing air flow direction vectors and temperature gradients (blue is coolest, dark pink is warmest)

Major technical hurdles that had to be overcome when designing mechanical ventilation for this facility included providing the grass with sufficient sunlight for proper growth and damage repair between ball games, while ensuring that patrons will enjoy comfortable environmental conditions during events. Grass growth required that the roof be open during the day in a city where summer ambient temperatures often reach 110°F and can exceed 120°F. In contrast, comfort requirements dictated that the mechanical cooling system have a closed roof for proper climate control. The demanding daylight requirement of the grass severely limited the time available to cool the stadium with the roof closed prior to a game. Ellerbe Becket Architects retained RWDI to assist in the evaluation of the mechanical systems under the direction of the project mechanical engineers, M-E Engineers, Inc. Early in the design process, wind tunnel tests and CFD were used to test the effectiveness of augmenting the mechanical systems and natural ventilation through openings under the roof at two ends of the stadium.

(Continued on page 4)

RE-ENGINEERING OF A FURNACE SPRAY COOLING TOWER

By Ray Sinclair, Principal

At Falconbridge Limited, a mining company, hot gases laden with particles are discharged from a smelting furnace into a spray cooling tower before entering a cyclone separator. The existing spray cooling tower had a significant build-up of particles on the north inside wall that the company wanted to eliminate.

Four factors contributed to the problem:

- poor uniformity of the gas flow in the tower, caused by the geometry of the off-take duct connection to the tower;
- fine-mist spray nozzles possibly wetting the tower walls;
- variability of the off-gas flow rate, temperature, and particle loading; and
- chemistry of the particles causing them to be sticky under certain conditions.

RWDI worked with Falconbridge Limited to determine modifications to improve the gas flow distribution within the tower and then developed a CFD model to test the effectiveness of these ideas.

Figure 1 (a) is the before picture which highlights the relatively high speed flow that collides with the south wall. Note the blue coloured zone in the lower part of the tower on the north side, where the flow recirculates in a vortex-like fashion. These patterns agreed well with measured gas-speed profiles and photographs of the build-up on the walls of the existing tower.

Figure 1 (b) illustrates a sample of calculated trajectories of water droplets discharged from the spray nozzles. These spray predictions were useful in assessing the potential impact on the walls where wetting was known to cause build-up and also defined the extent of the cooling zones in the tower.

Figure 2 is the after picture which illustrates the redesigned geometry of the spray cooling tower that produced acceptable flow uniformity at an acceptable cost. The geometric modifications included angling the off-take duct so that the flow would be more aligned with the tower. One turning vane was installed in the angled elbow of the off-take duct to spread the flue gas flow more uniformly across its outlet. A second turning vane was located in the tower to help reduce or eliminate the flow separation bubble on the north side wall and produce a more uniform flow across the tower.

At the top of the tower, an extension with a new transition shape was added to provide enough time for the water spray droplets to completely evaporate within the tower at the future higher gas flow rates and not cause another build-up problem.

After modifications to the real tower, gas speed measurements again showed satisfactory agreement with the predicted flow patterns. This success was an important first step in upgrading the performance of the furnace off-gas system.

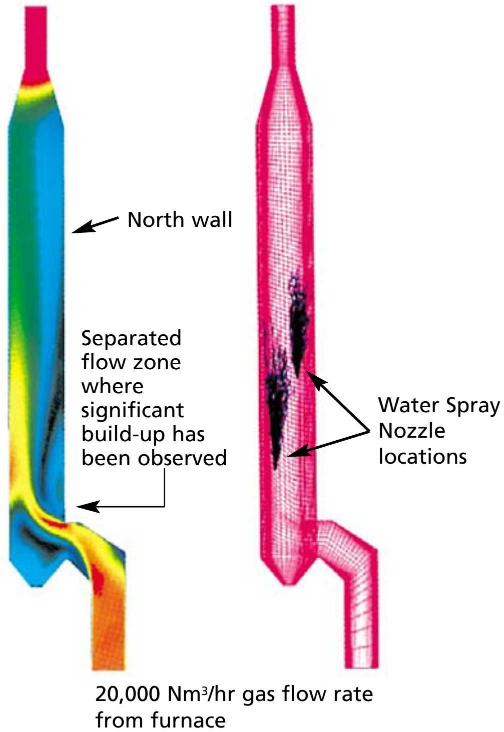


FIGURE 1: CFD model predictions for the existing spray cooling tower:
a) Gas speed distribution (blue is <3mls; dark pink is >17mls),
b) Water spray droplet trajectories.

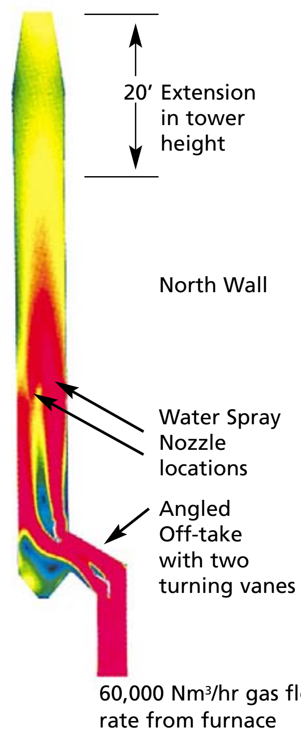


FIGURE 2: Predicted gas speed distribution in the modified spray cooling tower (blue is <3mls; dark pink is >27mls

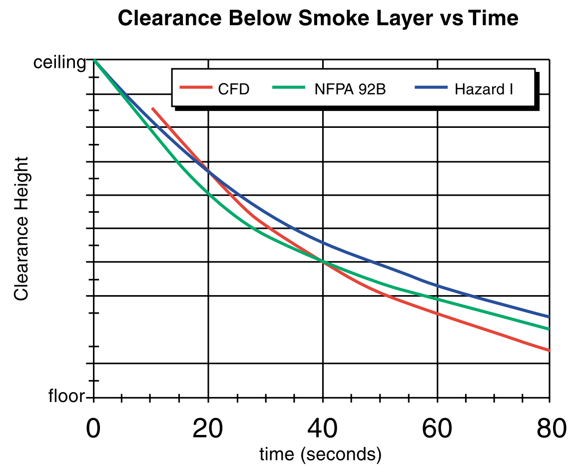
FIRE CODES ARE NOT ENOUGH

By Frank Kriksic, Principal

The design for smoke management systems in large indoor spaces such as atria, convention centres, airport terminals, sports complexes and industrial facilities has been governed primarily by prescriptive fire codes (e.g., a required number of air changes per hour). However, with an increasing emphasis on performance-based codes and the potential for litigation, there is a greater need for engineering analyses to demonstrate adequate performance (e.g., maintaining a smoke layer above a certain height). These analyses have traditionally involved calculation methods based on empirical equations to predict the migration of smoke during a fire.

Two types of computer models have evolved to assist designers with this process: these are zone models and CFD models. Zone models, such as Hazard I, divide the indoor space into two distinct zones with uniform characteristics to predict the smoke layer thickness and average temperature versus time. CFD models divide the space into a 3-dimensional grid with thousands of discrete volumes. With CFD, the predicted distributions of air speed and direction, temperature, pressure and smoke concentrations are available for the entire space.

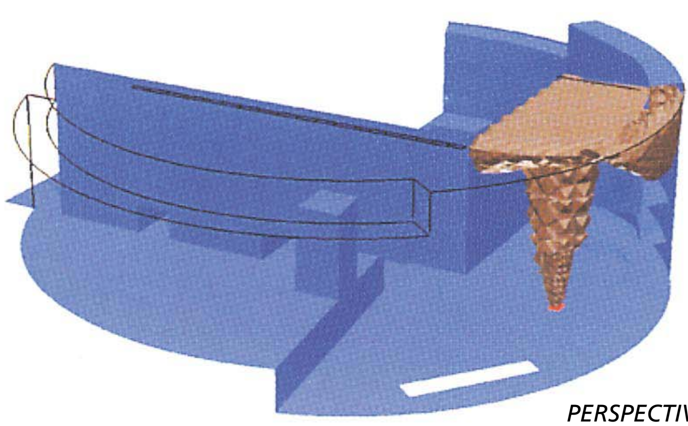
The adjacent figure compares the predicted growth of the smoke layer in a hypothetical atrium with no ventilation and a steady-state fire using the empirical calculation methods prescribed in NFPA 92B, the Hazard I zone model and a CFD model. As shown, there is good agreement among all three methods for this "idealized" case. However, in a more realistic scenario, complex building geometries and the effects of normal ventilation systems and smoke control systems can cause significant variation in the smoke layer depth. CFD modelling methods are well suited to predict smoke migration patterns in these more complex situations.

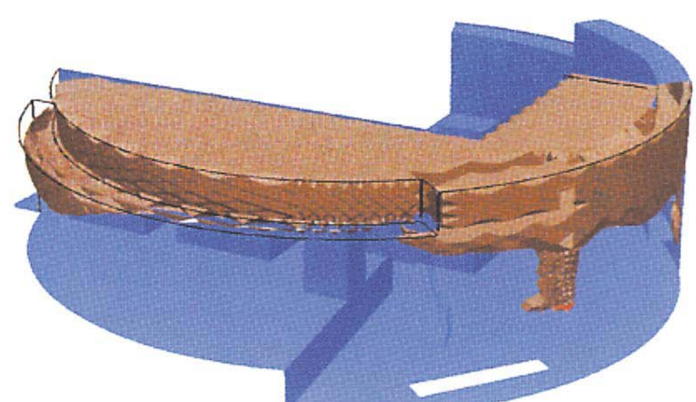


Comparison of CFD and traditional calculation methods

Shown below are two illustrations of an office building atrium that was modelled using CFD to simulate internal air flow and smoke migration patterns during a fire. The primary concern of the design team was smoke obscuration of emergency exit routes. The large size and complex geometry of the atrium warranted additional analysis beyond the simpler smoke modelling methods.

Results of the CFD simulation graphically illustrate that within 240 seconds of fire ignition, the upper portion of the atrium was engulfed in smoke obscuring many of the higher level perimeter walkways connected to the atrium. This indicated that the proposed smoke control system could not maintain safe egress passages for occupants, even though the tested design exceeded the required number of air changes prescribed by the fire code. A similar conclusion could not readily be reached with the use of traditional calculation methods described above. It was further demonstrated in this project that the distribution and location of the supply air proved to be critical in the performance of the system.





PERSPECTIVE VIEWS OF ATRIUM:

a) Smoke plume 80 seconds after ignition of fire

b) Smoke plume advance after 240 seconds

Results showed that wind-driven air currents in the stadium could adversely affect the distribution of chilled air. Therefore, to improve cooling performance and to reduce the cost of the air cooling system, the under-roof openings were eliminated.

A second factor in the design required evaluation of the solar energy stored by the concrete during the open-roof periods, since this would be a primary factor in cooling the facility prior to an event. To assess this effect, a numerical model was used to predict variations in concrete and air temperatures at several representative locations in the stadium through a typical midsummer day. Results of this analysis showed that time lags in the 4” to 5” thick concrete were such that only the thermal history over the previous six hours was significant. Additionally, hollow, lightweight seats provided significant shading for the concrete, which reduced the temperatures and improved comfort.

With the predicted heat load distribution of the concrete, as well as the estimated heat load of spectators and lighting, a CFD model was developed to predict airflow patterns and temperature distributions in the ballpark. This information was used to assess the effectiveness of the design of the conditioned air delivery system, in terms of providing uniform, comfortable conditions at game time. Results showed that negative buoyancy of the chilled air required re-aiming the supply air jets upward from their original direction. To provide the best balance between delivery of cool air out to the extremes of the seating decks and to avoid cold drafts near the jets, they were aimed horizontally instead of parallel to the seating deck.

The final configuration demonstrated the feasibility of the proposed operation. It also identified certain areas near openings to concourses where ventilation may be reduced at times in order to maintain comfort and reduce operating costs.

Rowan Williams Davies & Irwin Inc. (RWDI) is a leading wind engineering and microclimate consulting firm - the result of more than 30 years of growth and development. From offices in Canada, the United States and the United Kingdom, our consultants meet the world’s most complex structural and architectural challenges with experience, knowledge and superior service. In the early planning stages, careful attention to the effects of wind, snow, ventilation, vibration and related microclimate environmental issues on buildings and structures will save time, save money and reduce risk. Our capable and qualified staff uses advanced engineering tools and carefully defined consulting processes to deliver understandable and useful results.

RWDI

CONSULTING ENGINEERS
& SCIENTISTS

Rowan Williams Davies & Irwin Inc.
(519) 823-1311 www.rwdi.com

RWDI Anemos Ltd.
01582 470250 www.rwdi-anemos.com

Wind and Microclimate Services:

- Acoustics, Noise & Vibration
- Environmental Engineering
- Hazard & Risk
- Wind Engineering
- Microclimate
- Regulatory Permitting
- Industrial Processes