The 6th IBPSA Canada Conference
La 6e conférence d'IBPSA – Canada
Winnipeg, May 19 – 20

USE OF CFD AS A PREDICTIVE TOOL IN AIR FLOW AND THERMAL COMFORT

Warren Brooke, MSc, EIT, LEED AP
Stantec Consulting Ltd.
Calgary Alberta, Canada

ABSTRACT

Computational Fluid Dynamics (CFD) has become an effective tool in analyzing airflows related to building operation. However, in terms of non-academic use the field is still in its infancy and there is currently a lack of real-world validation of computational results within the literature.

In this study, CFD was used to evaluate the effectiveness of different ventilation schemes for SaskPower's B-plant thermal power station at Boundary Dam in Estevan, SK. This coal-fired power station is a very large space with extremes of heat gains, high local air velocities, and complex geometry. This represents a challenging test of the capabilities of CFD to produce realistic results.

The Boundary Dam B-plant powerhouse contains two 150-MW lignite-fired boiler units (Unit #3 and #4) and the turbine hall houses the associated turbines and generator sets. In summer conditions, workers in the plant are exposed to extreme temperatures which radically reduces their allowable work/rest schedule. The reduced work schedule (15 minutes of work/45 minutes of rest in an air-conditioned space) precludes the possibility of summer maintenance on one boiler while the other is operating. SaskPower requested ventilation options that would result in conditions on the 12th floor that would yield a work/rest regime of 45-minutes work to 15-minutes of rest when one boiler is operating and the other is off, and given the 0.4% annual ASHRAE outdoor design air temperature which is 32.1^{o}C in Estevan, Saskatchewan.

Using CFD, several ventilation options have been evaluated, including adding additional exhaust fans along the walls at the highest level, ducting cool outdoor air directly to the work zones, and installing passive gravity ventilators on the roof. Using CFD as a quantitative evaluation tool, the different ventilation schemes were compared to determine their respective effectiveness at reducing the air temperature in the plant to a level that would greatly enhance worker comfort and productivity.

Stantec's team was able to validate the CFD method by accurately reproducing historical temperature data taken within the plant by operators. With the validated model, Stantec was able to definitively demonstrate that the gravity ventilator option was the most effective solution, with the lowest capital cost and by far the lowest operating cost.

INTRODUCTION

The field of fluid dynamics is governed by the Navier-Stokes equations which describe the transport of momentum for an incompressible flow of a Newtonian fluid. In Cartesian coordinates these equations, in the x, y, and z directions respectively, are shown in equations (1) where u, v, and w are the components of velocity, ρ_f is the fluid density, t is time, g_x, g_y, and g_z are the components of gravity, P is the pressure and μ is the viscosity of the fluid. This set of equations is closed with the continuity equation.

$$\left.\begin{aligned}
\rho_f\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}\right)&=\rho_f g_x-\frac{\partial P}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)\\
\rho_f\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)&=\rho_f g_y-\frac{\partial P}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)\\
\rho_f\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)&=\rho_f g_z-\frac{\partial P}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)
\end{aligned}\right\}\quad(1)$$

These equations represent a balance between convection of momentum on the left, and gravity, pressure and viscous forces on the right. It is interesting to see that the entire field of fluid dynamics can be stated so succinctly, but their neat form veils a frustrating fact: this set of equations has remained un-solved since their publication in 1822. The Navier-Stokes equations exist as one of the so-called "Millennium Problems" for which the Clay Institute of Mathematics has sponsored a one-million dollar prize for a solution. As Delvin (2003) puts it, "No one has been able to find a formula that solves the Navier-Stokes equations. In fact, no one has been able to show in principle whether a solution even exists! The most significant lesson we have learned to date is that the mathematics of fluid flow seems to be extremely hard."

As an alternative to finding closed-form solutions to the Navier-Stokes equations, Computational Fluid Dynamics aims to find approximate solutions to the set of NS equations by separating the solution domain into small discrete volumes, and then reforming the set of continuous partial differential equations into a matrix of simultaneous algebraic equations.

Buoyancy-driven flows, such as those presented in this paper, require a special treatment of the density terms in the Navier-Stokes equations, since it is the differences in fluid density that drives flows of this type. In buoyancy-driven flow the Boussinesq approximation is employed, in which the change of density is ignored on the left-hand side of equations (1), and is evaluated only in cases where the density is multiplied by gravity. This approximation is valid since the difference in inertia (left hand side of NS equations) will not be great due to density variation, whereas the differences in body forces are non-negligible. The change in density is typically driven by changes in temperature and evaluated using the coefficient of thermal expansion.

The CFD technique did not take off until the late 1980 and early 1990's when computer processing speeds and memory capacity made it possible to solve such problems.

In a search of the literature, Martin (1999) states that "CFD [is] a tool that has been reserved mainly for researchers and manufacturers with research and development budgets, [but now] is ready for use as an HVAC&R consulting engineering tool."

Martin (1999) also offers some insight into why CFD results from industry are scarce within the literature: "A serious commitment is required for a consulting firm to begin offering CFD analysis. To obtain meaningful CFD solutions, an engineer must be familiar with the use of complex software, must have knowledge of fluid mechanics and the ability to relate the results to real world situations. Most consulting engineering projects do not have research and development budgets that permit physical verification or calibration of a CFD model. Instead, an experienced engineer, knowledgeable about the physical process and software, will analyze the software outputs for meaningful results and errors."

One of his conclusions promotes the power of CFD to provide insight when developing design solutions to unusual situation.

Within the last decade the cost of CFD software continues to decline, while the speed of computer processors and memory capacity continues to climb. Now there are many commercially available CFD solver codes as well as pre-processor mesh generators. Also these programs have become more user-friendly as Zhigiang Zhai (2006) comments: "buildings and systems modelled through CFD have become more and more sophisticated, while less knowledge of fluid mechanics and building science is required to conduct CFD simulations due to the smart graphic user interfaces of commercial CFD programs". This, however, may also lead users with little knowledge of fluid mechanics to provide erroneous results to their clients, since some sophistication on the part of the user is required to interpret the validity of CFD results.

Wang and Zhu (2003) presented a practical case study of methods for improving airflow from un-shrouded air impellers within a large industrial malt processing facility. Wang and Zhu state, "in this case, the simulation test initially provided the evaluation with fruitful information on the significance of retrofitting, and the owner was therefore persuaded to invest in the retrofitting project... The cost of computation tests decreases continuously while labour and material costs [for physical tests] continue to rise…In the actual retrofitting design process, the simulation allows a large number of tests to be conducted, which is costly and impractical in terms of time, and the best retrofitting design can be achieved within a very short time."

One of Wang and Zhu's conclusions about using CFD for analysis is, "the simulation outputs can give fruitful and convincing numerical details of the effects and benefits of retrofitting. This "numerical evidence" given by the CFD simulation tests can provide great support in convincing the managers to fund the retrofitting project."

Pitarma, Ramos, Ferreira, and Carvalho (2004) used CFD to illustrate the effects of indoor air quality with respect to the relative location of smokers to the air delivery to, and exhaust from, an indoor space. They state that, "the CFD technique permits prediction and visualization of the indoor environment [and] it represents a powerful new tool to teach and sensitize about indoor-environment problems."

Zhigiang Zhai (2006) gives a recap of several trends in CFD applied to building design. Zhai first makes the case for using CFD for site planning, which will influence flow around neighbouring buildings and topology, which in turn affects pedestrian comfort, energy loss from the building and the provision of natural ventilation. He then provides examples of CFD used in internal flows where CFD informs the design of HVAC systems for thermal comfort and enhanced indoor air quality. Zhai states that CFD is particularly useful in HVAC design for innovative systems and for large spaces such as atria, concert halls and sports facilities where empirical formulae for unconfined jets and charts from diffuser manufacturers may not be applicable.

Zhigiang Zhai and Qingyan Chen have presented several papers (2003, 2005, 2006) about the integration of CFD into other building energy simulation software. Although such a coupling of these two approaches is computationally expensive and time consuming, there is a natural synergy between the two since CFD can provide to the energy simulation software more accurate estimations of the convective heat transfer at exterior surfaces as well as the dynamic indoor conditions that affect thermal comfort and indoor air quality, whereas the energy simulation software can provide the boundary conditions necessary for accurate CFD predictions. However, even in it's currently advanced state and using parallel computer processors, a CFD model can take hours to days to converge on a solution, so full integration on each time step of a sub-hourly energy model is currently not possible. They report that a CFD simulation performed once every 2 simulated hours of building energy modelling time can produce reasonably good results, whereas a CFD frequency of 3 hours or more may give "obviously non-smooth solutions". A conclusion they draw is that indoor air temperature stratification has only a weak influence on the simulated heating and cooling load, while the same effect can influence the required supply air flow rate by as much as 40%. This can have a large impact on the simulated fan energy consumed.

METHODOLOGY

The general approach to building the CFD model of SaskPower's Boundary Dam B-plant was to include components that introduce or exhaust air from the space, solid components that impede the flow, and components that add heat to the space. For heat-generating components, the heat flux was assumed to be uniform over the surface of the component.

The heat gains to the space are mainly attributable to the two 14-storey coal-fired furnaces, with their related economizers, air heaters, primary air ductwork, forced-draft air ductwork, flue-gas ductwork, and steam piping. Other gains to the space included infiltration from the turbine hall, infiltration from A and C plants, heat gains from electric motors, and building envelope losses.

Five different methods were employed by Stantec's Fredericton office to estimate the heat loss from the boilers. These methods are listed below:

Method 1: Heat loss from a hot surface to the surroundings following methods published in the ASHRAE Fundamentals Handbook.

Method 2: Using heat loss per square foot values that Babcock and Wilcox have published in the Steam Generation handbook, 41st Edition.

Method 3: The idealized heat transfer based upon first principles. This method does not include consideration of practical installation conditions such as insulation gaps, thermal bridges or degradation of insulation.

Method 4: Heat gain based on the incoming air inflow and outflow conditions on the days when physical measurements were made in the plant. This method treats the plant as a "black box" in that the heat gain is inferred from the exhausted air temperatures rather than by summing the heat gains from individual pieces of equipment within the plant. This method directly accounts for all of the heat gains inside the building, not just the boilers.

Method 5: This method follows ASME Performance Test Code 4 (ASME PTC-4 1998) which deals with heat loss through the walls of a boiler to the surroundings.

A summary of the heat gain calculations are given in Table 1.

Table 1: Summary of Calculation Methods for Heat Gain to the Boiler House

Method	Heat Gain Method	One Boiler Operating BTU/hr (kW)	Two Boilers Operating BTU/hr (kW)
1	ASHRAE	18,659,692 (5,464)	33,972,856 (9,948)
2	Babcock and Wilcox	18,345,458 (5,372)	33,344,389 (9,764)
3	First Principles Heat Transfer	11,061,684 (3,239)	18,776,841 (5,498)
4	Airflow Heat Balance	21,814,244 (6,387)	31,689,674 (9,279)
5	ASME PTC-4 1998	19,300,474 (5,651)	35,254,420 (10,333)

The heat gain calculations shown in Table 1 generally agree with one another except for method 3, which is based on first principles heat transfer from an idealized boiler wall. This outlier was removed and the four remaining values averaged to determine the heat gain to the space. When only one boiler is operating there is approximately 21,000,000 BTU/hr heat gain to the powerhouse, while there is 35,000,000 BTU/hr heat gain when two boilers are operating. For the CFD model, these heat gains were distributed to the equipment within the plant based on their surface areas. The main limitation of this assumption is that it assumes each component has the same heat loss per unit area, whereas in reality some components may have higher or lower heat loss than others.

The floor grating throughout the plant was not included in the CFD model since it has a high percentage of free area and will offer little resistance to flow, especially at the low velocities encountered in a large building. Concrete floors were included since these will impede the flow of air.

Air is actively exhausted from the space by a series of tube-axial fans located on the north and south walls at the 14th storey (highest level). On the north wall four fans exhaust 48,000 cfm (22.65 m^3/s) each, while on the south wall, six fans each exhaust 32,000 cfm (15.10 m^3/s). The roof has two passive "mushroom hoods" which are estimated to exhaust 50,000 cfm (23.60 m^3/s). Air is also removed from the plant through four primary air fans, and two forced-draft fans that feed the two furnaces (2 PA fans and 1 FD fan for each furnace). Each PA fan removes 55,500 cfm (26.28 m^3/s) from the 9th floor level, while each FD fan removes approximately 235,000 cfm (111 m^3/s) from the 7th floor level.

Air enters the plant through a series of operable windows on the north wall of the turbine hall, and a set of eight 12-ft x 12-ft roll-top doors on the south wall. A number of smaller openings on the south wall were also included in the model.

The model geometry is shown in Figure 1.

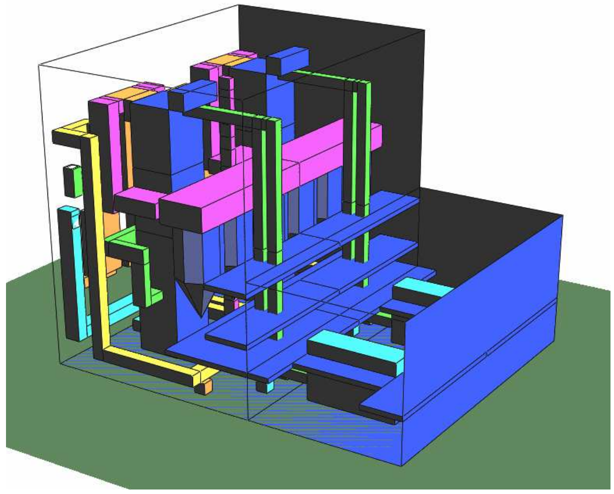

Figure 1: CFD model of SaskPower Boundary Dam B-plant

The alternatives examined for mitigating heat stress include (a) adding more exhaust fans along the walls at the same level as the existing exhaust fans, and (b) keeping the current exhaust fans and adding passive gravity roof ventilators to allow hot air to escape using its own buoyancy.

SIMULATION

For this project, a building analysis software called IES Virtual Environment was used, which includes a CFD module called MicroFlo. This package includes a modeling environment, an orthogonal grid generator, a solver and a post-processing viewer.

In the CFD simulation, the building is subdivided into approximately 2.5-million rectangular cells, with a default dimension of 0.5m per side. A hybrid discretisation scheme was employed, as opposed to the less computationally expensive (but also less accurate) Upwind scheme. Since grid lines are automatically attached to the boundary of each component, smaller cells are generated at numerous locations. At each cell, the air velocity, temperature, density, and turbulent kinetic energy is calculated. The turbulence model employed in this study was the well known k-ε model. In this model, the turbulent kinetic energy, k, and the dissipation rate, ε, are used to calculate a turbulent viscosity which is added to the dynamic (laminar) viscosity to obtain the effective viscosity.

The initial condition was set to 35 C, and the boundaries (walls) were also set to 35 C. This value was chosen since it is the temperature on the ground floor following a linear trend from the temperature data taken throughout the plant on April 09, 2008.

RESULTS AND ANALYSIS

CFD simulation results were generated for the existing ventilation conditions at Boundary Dam and compared to the historical data measured within B-plant by SaskPower personnel during the summer of 2008. The data taken on August 09, 2008 was used for the case when both Unit #3 and Unit #4 are operating. This data series includes measurements taken at a location between the boilers on each floor between floors 2 and 14. The outdoor drybulb temperature at 2:00pm on August 09, 2008 was taken as 26.3 °C from the weather data recorded at Estevan Airport. The assumptions of this simulation were that all of the fans on the upper level were exhausting air at their full rated capacity, and that all of the openings in the building envelope were fully open. The model validation chart for August 09, 2008 when two boilers were operating is given in Figure 2. The model 113 Heat Stroke Checker instrument that SaskPower used to measure the temperature in the plant has a stated uncertainty from the manufacturer of ±2°C. This uncertainty interval is shown on each measurement point.

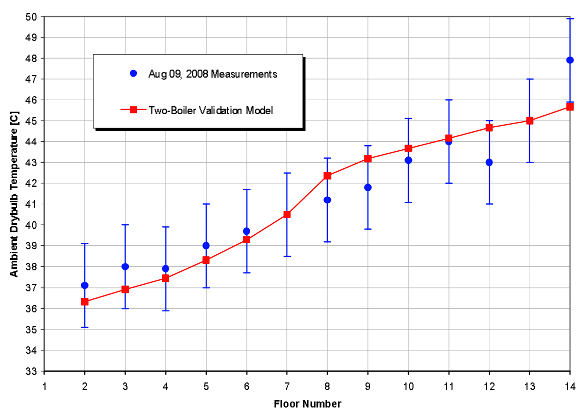

Figure 2: CFD model validation curve using August 09, 2008 temperature measurement data when two boilers were operating.

Temperature data was also taken at various points in B-plant on August 26, 2008, during a period when Unit #4 was offline. This data series was used to validate the CFD model for the case when only one boiler is operating. The historical weather data shows that on August 26, 2008, the outdoor drybulb temperature was 20.6°C. The validation data points for the one-boiler case are shown in Figure 3. In this case the measurement locations are not the same for each floor, and also are not in the same locations as in Figure 2 in the two-boiler case. Due to their different locations, no trend line was plotted in this case. The location of the measurements is noted on the chart.

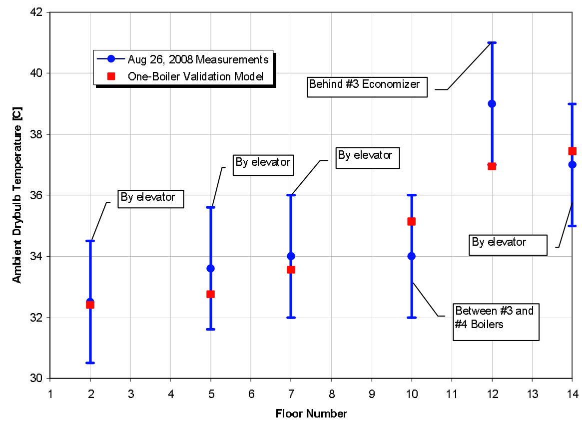

Figure 3: CFD model validation curve using August 26, 2008 temperature measurement data when one boiler was operating.

Figures 2 and 3 demonstrate that the CFD model is able to predict the temperature within the plant when given the corresponding outdoor drybulb temperature as a boundary condition. The only predicted temperatures that are outside of the error bands of the measurement device are on floor 12 in the one-boiler case, and floor 14 on the two-boiler case. In the one-boiler case, this result may be due to the uncertainty in the location that the measurement was made within the plant. In the two-boiler case it is interesting to note that the measurements made on August 09, 2008 tend to follow a different trend for floors 12, 13 and 14 than floors 2 through 11. This may reflect different processes going on within the boiler at the upper floors which increases the heat loss to the plant at these elevations. In general, the assumption of constant heat loss over the surfaces of each component seems to be supported by the results shown in Figures 2 and 3, but perhaps the heat flux on the upper three floors of the furnaces is higher than on floors 2 through 11.

Given the validation shown in Figures 2 and 3, it is felt that the CFD model adequately represents the powerhouse heat gain and the performance of the ventilation systems. These results indicate that the CFD analysis method is suitable for problems of this type, and the heat gain calculation methods listed in Table 1 are valid.

The CFD model has been validated for the existing ventilation case in the B-plant powerhouse, both for conditions when one and two boilers are operating and for differing inlet air temperature boundary conditions. This validation lends credibility when applying the model to the purposes of evaluating the effectiveness of different ventilation improvement options.

SaskPower's preferred ventilation option was to simply add more exhaust fans at the 14th floor walls where the existing exhaust fans are. This option would allow easy access to the fans and motors for maintenance, and mounting locations were readily available. This option makes intuitive sense, since the fans would actively exhaust the hot air that would naturally pool at the ceiling. Figures 5 and 6 show the results of adding an extra 1,100,000 cfm (519.14 m^3/s) of active exhaust to the highest level of the north and south walls.

To facilitate comparison, the CFD results have been arranged and presented all together on the next page.

The scale in Figure 4 has been used to indicate local dry-bulb temperature for each of the CFD results shown in Figures 5, 6, 7, and 8.

Figures 5 and 6 demonstrate that simply adding more exhaust fan capacity does not mitigate the high temperatures within the core of the building. A hot plume rises up the operating boiler and then spreads out across the width of the plant at the ceiling. A high-temperature "cone" of air extends half-way down the height of the building, while cooler outdoor air is simply drawn up the walls and exhausted without penetrating into the core of the building where maintenance crews would be working. Through several iterations of different exhaust flow rates, it was determined that with the fans arranged at the walls at the high level, no amount of additional exhaust fan capacity would address the high temperatures within the core of the building. This insight would not be possible through any analysis method other than CFD.

As an alternative, Stantec recommended the installation of passive gravity roof ventilators with throat areas expected to exhaust approximately 1,100,000 cfm (519.14 m^3/s) given the heat gain within the space and the height of the building. The results of this simulation are shown in Figures 7 and 8.

Figures 7 and 8 show that the gravity roof vents are very effective at mitigating the heat build-up within the plant. The "pool" of hot air that collects at the ceiling is allowed to escape through the roof vents driven by its own buoyancy. Thus, this option does not consume power in operation, nor does it require the installation of motor-control centres or the associated power cabling. Also the gravity vents do not require the maintenance that would be associated with powered fans and motors.

CONCLUSIONS

Although CFD has been used in academic settings since the 1980's, validation of this technique within real buildings has been rarely reported within the literature.

This paper gave the results of an analysis of air-flows and temperatures within a real industrial building using computational fluid dynamics as an analysis tool. It has been shown that despite the complex geometries and high heat gains within the building (a 150-MW coal-fired power station), the CFD results were able to closely reproduce physical temperature measurements taken within the plant by SaskPower operators.

With respect to buoyant flow within tall buildings, it has been shown that exhaust fans placed along the walls at a high level do not necessarily mitigate the temperature rise within the core of the building. It has been shown that the cool outdoor air introduced through wall openings at lower levels is simply drawn up along the outer walls and is exhausted at the high level exhaust fans. The CFD results in this study have illustrated flow characteristics that would be impossible to determine through other analysis techniques other than physical models which are time-consuming and costly. Further study is required to determine if there is a critical building width-to-height ratio that would make the wall-mounted fans more effective.

In this building, the most effective solution for mitigating the high temperatures experienced within the plant is to put passive gravity ventilators on the roof. This allows the hot, buoyant air to escape the building rather than pooling at the ceiling.

This project has demonstrated that CFD is very versatile and once a model is built, many design options can be evaluated in a short time and for far less cost than building physical test models.

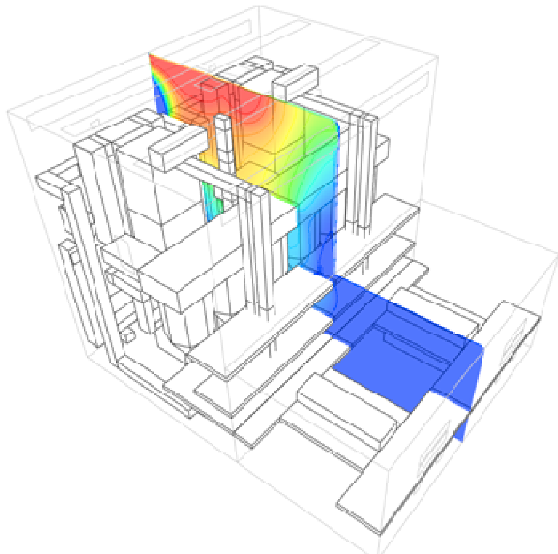

Figure 5: CFD results of adding 1,100,000 cfm of additional exhaust capacity along the north and south walls at the 14th floor, with one boiler operating. Slice between boilers.

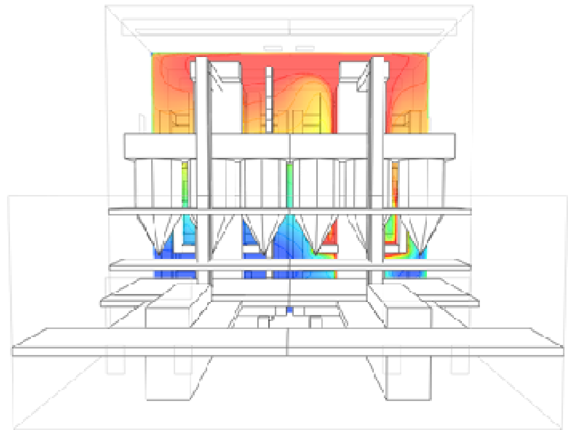

Figure 6: CFD results of adding 1,100,000 cfm of additional exhaust capacity along the north and south walls at the 14th floor, with one boiler operating. Slice across boilers at centerline.

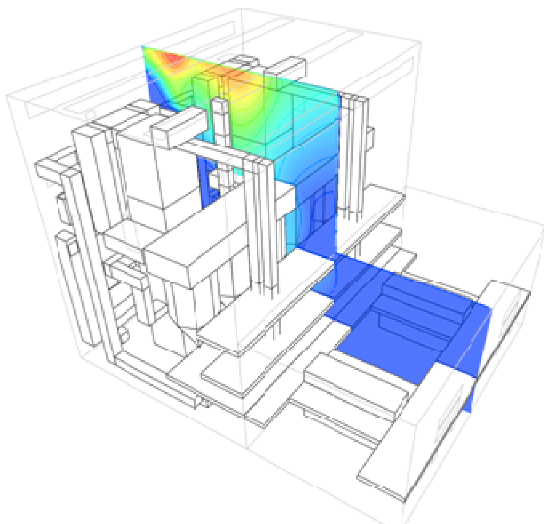

Figure 7: CFD results of adding 1,100,000 cfm of passive gravity roof ventilator capacity in two strips along the roof. One boiler is operating. Slice between boilers.

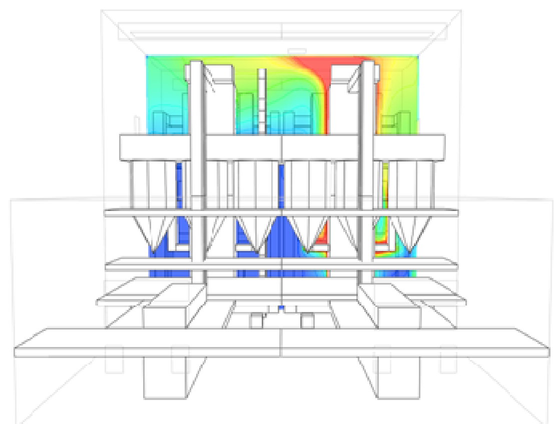

Figure 8: CFD results of adding 1,100,000 cfm of passive gravity roof ventilator capacity in two strips along the roof. One boiler is operating. Slice across boilers at centerline.

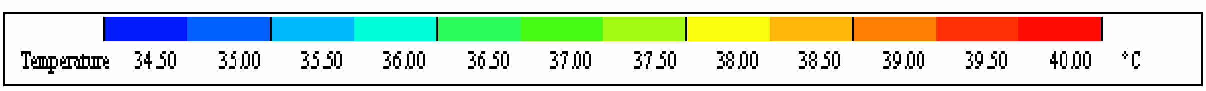

Figure 4: Colour scale of local dry-bulb temperatures applied to Figures 5, 6, 7, and 8.

ACKNOWLEDGEMENTS

The author is very grateful to the contributions made by Kevin Curtis and Chris Reid of the Stantec Fredericton N.B. office who performed the heat gain calculations necessary for this work, and who recommended the gravity roof ventilator concept. This investigation was sponsored by SaskPower Corporation who have generously allowed the results to be presented to the technical community. While SaskPower funded this study, it neither endorses nor rejects the findings of this paper. The methods, results and conclusions of this paper have been presented in the interest of furthering the field of CFD within the buildings research community and inviting comments and subsequent studies from other researchers.

REFERENCES

Delvin, K., The Millenium Problems, 2003, Basic Books, New York.

Incropera F.P. and DeWitt D.P. (1985), *Fundamentals of Heat and Mass Transfer*, John-Wiley and Sons.

Martin, P.; "CFD in the real world"; *ASHRAE Journal;* Jan 1999; 41, 1; ProQuest Education Journals, p.20.

Wang, S., and Zhu, D.; "Application of CFD in retrofitting air-conditioning systems in industrial buildings"; *Energy and Buildings*, vol. 35, p. 893-902, 2003.

Pitarma, R.A., Ramos, J.E., Ferreira, M.E., Carvalho, M.G.; "Computational fluid dynamics – An advanced active tool in environmental management and education"; *Management of Environmental Quality: An International Journal*; Vol. 15, No. 2, 2004, pp.102-110.

Zhiqiang Zhai, Qingyan Chen; "Solution characters of iterative coupling between energy simulation and CFD programs"; *Energy and Buildings*; vol. 35, 2003, pp.493-505.

Zhiqiang Zhai, Qingyan Chen; "Sensitivity analysis and application guides for integrated building energy and CFD simulation"; *Energy and Buildings*; vol. 37, 2005, pp.333-344.

Zhiqiang Zhai, Qingyan Chen; "Performance of coupled building energy and CFD simulations"; *Energy and Buildings*; vol. 38, 2006, pp.1060-1068.

Zhiqiang Zhai; "Application of Computational Fluid Dynamics in Building Design: Aspects and Trends"; *Indoor and Built Environment*; 2006, v.15, no.4, pp 305-313.

Anon.; ASHRAE Handbook Fundamentals; American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc.; Atlanta, Georgia, 1997.

Anon.; Steam, Its Generation and Use, 41st Edition; Babcock and Wilcox Company; Bartlett Orr Press, New York.

ASME Performance Test Code (ASME PTC-4), 1998.