DETERMINATION OF COOLING STRATEGY ON A 200M HIGH GLAZED LIFT SHAFT: A CFD APPROACH

Guillaume Jolly[1,2], Dr Julien De Charentenay[1], and Brenton King[1]
[1]Floth Sustainable Building Consultants, Brisbane, Australia
[2]Corresponding author Tel: +61419565870, Fax: +61732522499, Email: gjolly@floth.com.au

ABSTRACT

CFD modelling was identified as the only tool able to predict accurately the performance of a cooling system for a 200m high glazed building lift shaft located in Brisbane, Australia.

A conjugate heat transfer, multiband radiation and moving mesh transient CFD modelling strategy was used to assess and design the most energy efficient environmental control possible for this design.

The lift movement within the shaft increased the convective heat transfer by a factor of 5, top-down air distribution and transient CFD modelling analysis reduced by 60% the cooling load compared to a steady state bottom up ventilation model.

INTRODUCTION

Lift shafts are typically part of a building that has no requirement for environmental control systems and as such are not a typical part of a mechanical engineers design routine.

The One One One Eagle building situated in Brisbane, Australia consists of 62,500m^2 of office space spread over 52 floors and located within the Brisbane city CBD. The architect had a different approach to lift shafts than is typically seen in Australia by designing a fully glazed lift feature located on the western facade and oriented toward the city. The lift shafts were designed as a major feature of the building by providing lift users with views of the city of Brisbane as they travel up the building.

The western orientation of the façade meant that there was an operational need to control the internal environment of the lift shafts due to the large solar heat loads present. This requirement and the proposed facade design made a passive ventilation design untenable with respect to local climatic conditions. The building was targeting a 6 star Green Star environmental rating (Green Building Council of Australia, 2011) hence needed to employ an energy efficient design for the lift shaft environmental control system in order to achieve the desired environmental rating.

A number of unusual design challenges outside the scope of traditional design became apparent:

- Conventional heat load design calculation methods could not take into account the extreme temperature stratification expected in a 200 metre high atrium.
- The black coloured load bearing structural steel frame supporting the facade would absorb and transfer to the air large amounts of solar heat that would have otherwise been absorbed by the concrete.
- Air movement and convection currents generated by the high-speed lifts would significantly affect the heat balance within the lift shaft environment.

Conventional industry heat load calculation methods are based on the assumption of temperature homogeneity in the space. This assumption is not valid in this case due to:

- A highly stratified temperature profile
- The presence of the frame

There is no information in literature that can be used for quantifying the effect that the lift movements have on the convective heat transfer coefficient in the shaft. A moving mesh strategy was thus required in order to calculate the convective heat transfer coefficients (HTC_{conv}) due to the lift movements.

Due to the strong thermal coupling between all the elements of the shaft and the outdoor/solar conditions, using simplified calculation of boundary conditions was unfeasible. This complexity of heat transfer within the shaft required the use of a conjugate heat transfer method with a multiband radiation model.

The CFD software used for all models was STAR-CCM+ v4.06 by CD-Adapco apart from the moving mesh models for which STAR-CCM+ v5.02 was used.

DESIGN PROCESS

A number of deisgn options were considered and discarded:

- Natural Ventilation was not an option due to the requirements of the lift manufacturer. The air had to be filtered to remove the saline spray, which was considered as a potential threat to the lifts working order.

Natural ventilation also led to regulatory compliance issues.

- Colour of the shaft walls and steel frame were investigated early in the design, but concerns about cleaning requirements of light colours in a lift shaft led to black steel with grey walls to be adopted as the final solution.
- State of the art thermo-chromic glass or Double Facade solution was also discarded based on cost analysis and maintenance concerns.
- Ultra low SHGC glass solution was discarded based on the need for high Visible Light Transmittance required to make the moving lifts an architectural feature.
- A PCM based solution was discarded as being too difficult/lengthy to model. The model was already very complex and the team had no experience with PCM modelling at the time of design.

The following operational parameters were used during the study:

- Maximum Shaft Air Velocities
  The lift manufacturer has limited the lift shaft air velocity to a maximum of 5.0 m/s to ensure optimal lift car ride (i.e. Minimise buffeting).
- Maximum Lift Car Speed
  The modelling was to be based on a maximum lift car speed of 8.0 m/s.

MODEL GEOMETRY

The arrangement of the lift shaft is uniquely different when compared to a typical air conditioning zone. The shaft is less than 3 metres deep (including the concrete wall), nearly 200 metres high and as such will cause significant convective currents and temperature stratification.

The shaft has a high thermally absorbent metal frame which acts as a radiator within the shaft. The lifts travel at speeds up to 8 m/sec, thereby causing significant heat transfer to occur between the different solid elements, the outside and the air within the shaft.

The shaft dimensions used in this CFD model are 11m x 2.45m x 192.7 m high. The total glazed height to the lift motor room is 189.4 m (level 6 to level 55). The passenger's enter the lift at level 9 which then runs express to level 40. The lift can then stop at every floor between level 40 and 54. There are 3 lifts running in the shaft. The lift shaft is oriented west with a project North that is 14 degrees East of True North.

A simplified geometry for the lift shaft (shown in figure 1 below) was used in order to complete the CFD analysis within a timely fashion while maintaining practical level of accuracy.

Specifically, the following elements were simplified:

- The openings for ventilation through the concrete wall of the lift shaft were not modelled.
- The air inlet and outlet were modelled the whole top or bottom section of the shaft.
- The openings between the lift shaft and the lift motor room were not modelled as the current specification for sound attenuation around the lifts cables would make the airflow of little significance to the thermal conditions in the lift shaft.

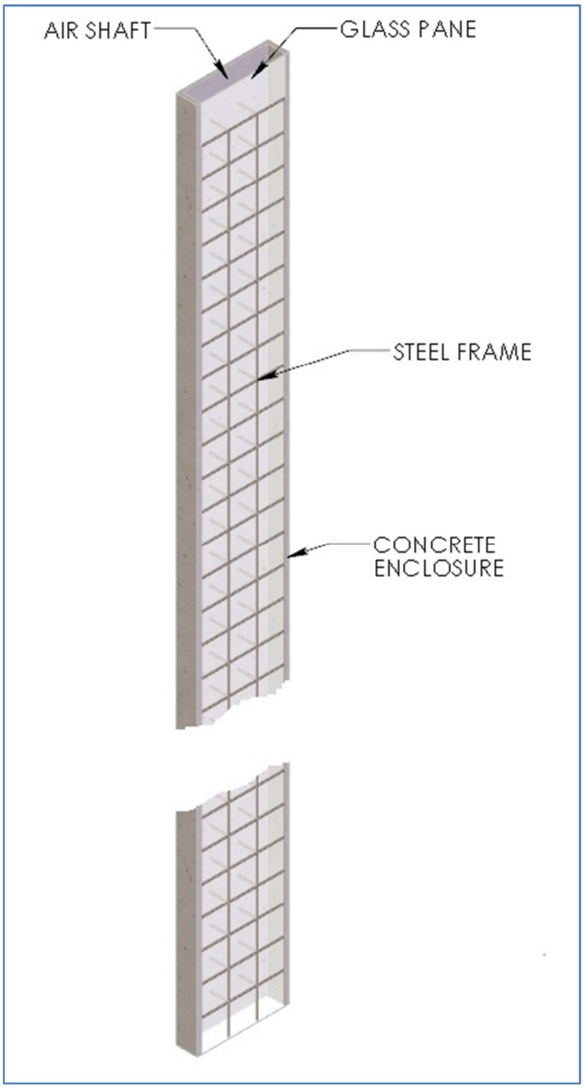

Figure 1: 3D representation of triple lift shaft

METHODOLOGY

This paper presents different runs of modelling which had a various goals. They are presented in table 1 below and further described in later section of this paper.

Table 1: Summary of modelling methodology

Scenario	Simulation Goal	Glass Properties	HTC_{conv}	Design T (^oC)
Unventilated Peak Load	Measure peak temperature of the shaft if left unventilated.	**Single Glazing:** Glass U value = 5.5 W/m2 SHGC =0.72 Combined frame + glass U value = 6.2W/m2 SHGC = 0.66 **Double Glazing:** Glass U value = 1.7 W/m2 SHGC =0.25 Combined frame + glass U value = 3.2 W/m2 SHGC = 0.22	Not taking into account lift movement.	40^{o}C
High Temperature Event Frequency Thermal Model	Determine yearly frequency of temperature raising above the design temperature with a thermal model.			
High Temperature Event Frequency CFD Model	Determine yearly frequency of temperature raising above the design temperature with a CFD model.			
Floor Exhaust Air and Outside Air Ventilated Scenarios	Determine the shaft temperature sensitivity to various colour schemes and outside temperatures. Assess the effectiveness of a ventilation only design.			
Moving Mesh Model	Generate the internal HTC_{conv} to use in shaft CFD models	N/A	Measured.	N/A
Bottom up vs. Top Down Ventilation	Compare the effectiveness of air distribution location to maintain design temperature.	Glass U value = 1.64 W/m2 SHGC =0.39 Combined frame + glass U value = 3.7 W/m2 SHGC = 0.39	Taking into account lift movement measured from moving mesh model.	40^{o}C or 35^{o}C
Transient Top Down Ventilation	Determine the impact of the shaft concrete thermal storage on its cooling requirements.			35^{o}C

UNVENTILATED PEAK LOAD TEMPERATURE ANALYSIS

Simulation

The goal of this simulation is to find out the temperature the shaft would reach if left unventilated. This was commissioned in order for the whole design team to understand the challenge presented by the unique characteristics of the shaft.

Only a steady state simulation was required for the purpose of this early stage model as the solar radiation and temperature were reasonably constant during the afternoon of a design day and the results are unambiguous. The outside temperature was set to be constant at 32 ^{0}C in line with the general building design criteria for its location.

The Solar Radiation settings for the model were chosen according to the following methodology:

- A test reference year (TRY) weather file (standard weather file for energy simulation) was analysed in order to find a reasonable estimate of peak solar gain on a Western façade in Brisbane.
- The 10th of December was selected in this data file as a day with peak solar radiation. The combined solar heat flux is above 800 W/m^2 for 7 hours on this day.
- The actual glass SHGC is dependent on the angle between the solar radiation and the glass solar characteristics were established for the different sun-glass angles through the afternoon.

The actual transmitted solar radiation is approximately 500 W/m^2 for more than 5 hours. At 5 PM, the peak combined solar radiation transmitted is 569 W/m^2 and corresponds to an outside diffused solar radiation of 100 W/m^2 and direct solar radiation of 818 W/m^2. The steady state model was run for these 5 PM solar radiation values.

The wind velocity is set as defined by the US National Fenestration Rating Council (NFRC) summer conditions (2.75 m/sec). It considered to be a worst case scenario for a façade under solar radiation that is warmer than its surroundings. Wind velocities are expected to be significant on a peak day since Brisbane gets thermally driven sea breezes from the pacific ocean on hot days.

Discussion and result analysis

Tables 2 and 3 shows four sets of average temperatures for different components of lift shaft. The steps in the graphs correspond to different colours of the frame ranging from clean white (lowest temperature set) to black (highest temperature set).

Table 2: Average Temperatures in the Different Parts of the Shaft (single glazing)

Colour of Frame	White	Light Grey	Dark Grey	Black
Temperature in Air Shaft (°C)	61	63	65	67
Temperature of Concrete (°C)	48	48.5	49	50
Temperature of Glazing (°C)	56	56.5	57	58
Temperature of Steel Frame (°C)	69	74	80	85

Table 3: Average Temperatures in the Different Parts of the Shaft (double glazing)

Colour of Frame	White	Light Grey	Dark Grey	Black
Temperature in Air Shaft (°C)	55	55	55	55
Temperature of Concrete (°C)	40	40	40	40
Temperature of Glazing (°C)	62	62	62	62
Temperature of Steel Frame (°C)	57	58	59	60

These tables clearly indicate that the temperature in the shaft are not acceptable and that ventilation or air conditioning is necessary.
They also show that the colour of the different elements in the lift shaft have a significant impact on the temperature profile of the shaft with single glazing but not so with double glazing for this particular unventilated scenario.

HIGH TEMPERATURE EVENT FREQUENCY THERMAL MODEL

Simulation

For the purpose of this project, a High Temperature Event (HTE) is defined as an hour during which the temperature in the lift rises above 40 ^{0}C.

The goal of this simulation is to determine the yearly frequency of the shaft temperature rising above a temperature threshold taking into account the thermal mass of the shaft walls and the rest of the building. It was commissioned in order to provide the lower end frequency of HTEs and provide an element of comparison to the CFD results.

A simplified thermal model was built in the IES Virtual Environment software using standard office construction material and standard National Australian Built Environment Rating System NABERS (Department of Environment of NSW, 2011) heat load profiles and schedules. The model takes into account the solar shading effect of the surrounding buildings on the shaft temperature.

In order to attempt to model the effect of the frame on the temperature in the lift shaft, the solar reflectivity of the shaft walls was set to 0 (Integrated Environmental Solutions, 2009). This is an approximation as the concrete walls would not heat up as much as a black metal frame due to its high thermal mass and the fact that the concrete walls are adjacent to 23 ^{0}C air conditioned spaces within the building.

The heat transfer in the shaft is wrongly calculated as if the shaft was a small space with a uniform temperature distribution. This is a software limitation that precludes from take into account lifts movement and stack effect in the shaft

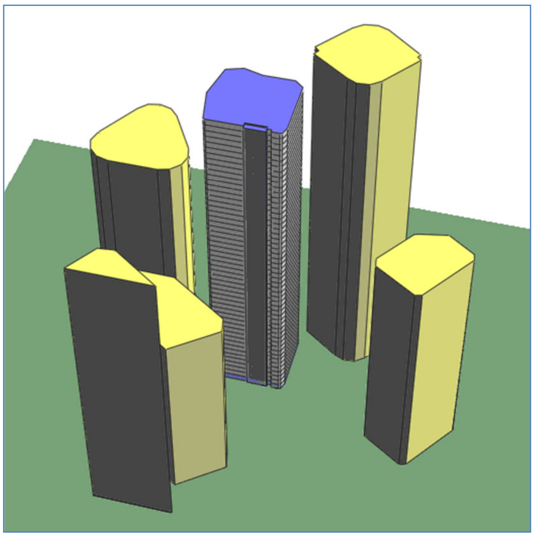

Figure 2: IES whole building model with surrounding buildings

This model is set up in the same way as a whole year energy model according to the NABERS protocol.

The weather file used is the Brisbane 1986 TRY file, which is used for NABERS, Green Star, and energy modelling to assess regulatory compliance with energy efficiency requirements. The weather in this file would be considered to be mild, in order to give average yearly energy consumption values and as such the simulation results can only be used as an indication of what would happen during a mild year.

Results

Single glazing:

The modelling and estimation of the lift shaft temperature indicates that the space exceeds 40 ^{0}C for approximately 281 hours per year or 3.2 % of the year. On a typical summer day (maximum temperature 30.5^{0}C at 1:00 pm) the temperature in the shaft is superior to 40^{0}C from 2:00 pm until 8:00 pm.

Double glazing

The lift shaft temperature is exceeds 40 ^{0}C for 1 hour per year. On a typical hot summer day (max outside temperature: 30.5 ^{0}C at 1:00 pm) the temperature in the shaft exceeds 30^{0}C continuously and is only just below 40^{0}C between 5:00 and 6:30 pm.

HIGH TEMPERATURE EVENT FREQUENCY CFD MODEL

Simulation

The goal of this simulation is to determine the yearly frequency of the shaft temperature rising above a temperature threshold using a detailed CFD model. It was commissioned in order to provide the upper end frequency of HTE's yearly frequency.
The CFD model used for the peak load calculation was run in order to find out the amount of solar radiation that creates a HTE. Direct solar radiation and diffuse solar radiation combinations corresponding to a HTE are modelled at 32^{0}C and 25^{0}C dry bulb outside temperature. A simple linear interpolation between these two points is then used to determine the number of HTE per year .

The angle dependent solar characteristics of the glass are calculated according to the ASHRAE Solar Heat Gain Coefficient (SHGC) method.

Discussion and result analysis

Single glazing:
The number HTE per year was 1027 hours.

Double Glazing:
The number HTE per year was 709 hours.

The results show that HTE are a very serious threat to the thermal comfort of the lift passengers and to the lifts equipment in an unventilated shaft. At this stage, it became clear for the whole design team that Air Conditioning was necessary to maintain comfortable conditions in the lift carts. The results also proved to sceptic non-experts that dynamic thermal modelling and CFD lead to very different results when it comes to this shaft.

FLOOR EXHAUST AIR AND OUTSIDE AIR VENTILATED SCENARIOS

Simulation

The goal of these simulations is to compare the relative impact of different design temperature, colour and ventilation schemes.

All the combinations of the settings below are modelled:
3 outside temperatures:

- 32 °C standard AC design
- 36 °C critical AC design
- 42 °C extreme AC design

2 colour schemes:

- All white, with steel structural frame and concrete walls painted in white
- Black and grey, with steel structural frame painted in black and exposed concrete walls.

3 ventilation schemes:

- No Ventilation
- Ventilation Occupied Hours, 300l/sec/floor of floor exhaust air
- Ventilation After Hours, 1100l/sec/floor of outside air

The solar radiation is set as per peak load conditions: 800 W/m^2 direct and 100W/m^2 diffuse.
For the ventilated scenarios, the ventilation air is exhausted at the top of the shaft and supplied by the floors outlets.
For after hour scenarios, the building internal temperature is set to 30 ^{0}C as worst case week end scenario.
Other settings are as per preceding simulations.

Each floor has three inlets which are 750mm long by 450mm wide. The inlets are spaced equidistant across all three lift runs in the shaft and placed at the top section of each floor.

Discussion and result analysis

The temperature stratification is very weak due to convection currents in the unventilated scenarios and due to the inflow of cooler air at every floor in the ventilated scenarios. Subsequently, the air average temperature gives an excellent measure of the thermal conditions in the shaft.

For single glazing, the temperature in the shaft is only maintained below 40 ^{0}C when the shaft is fully painted in white, with an outside of 32 ^{0}C and when ventilated by either with occupied hours ventilation or after hours ventilation. All other scenarios fail to achieve the 40 ^{0}C criteria.

With double glazing the lift shaft colour was found to have nearly no effect. The 42^{0}C outside after hour ventilated and the no ventilation scenarios fail to achieve the 40 C comfort criteria. All the other scenarios achieve shaft temperature below 40 ^{0}C, thereby satisfying the design criteria.

Floor ventilation scenarios results indicates that double glazing would be required in order to ensure acceptable comfort conditions in the lift, using 300l/sec/floor of floor spill air during occupied hours and 1100 l/sec/floor of outside air during after hours. The after hours strategy would fail when outside gets toward 40^{0}C.

Multiple cooling supply from the exhaust air of the building floors was investigated early in the design. It was discarded after realising that the HVAC system of the building did not allow for efficient use of this type of supply when the building is partially occupied due to after hour timing or lack of tenants. This type of supply would have required to turn on the fan coil units at every floor used to cool the shaft, whether the floor was occupied or not. This strategy also created a fire code compliance issues.

MOVING MESH MODEL

The goal of this simulation was to measure the increase in convective heat transfer between the various solid part of the shaft and its internal air due to the lift movements. Including the effects of lift car movements within the lift shaft analysis required the use of a moving mesh technology. In this approach, the movement of lift car is explicitly specified within the CFD model and accounted for. This approach places a heavy burden on computational resources and time by its requirements of using transient analysis and heavy meshing requirements.
The moving mesh analysis is run as a separate adiabatic model to generate convective heat transfer coefficient data that were then utilized in the final CFD models used to determine the cooling plant sizing. The three reasons for this decision were that:

- A heat transfer models required for the final CFD analysis was not compatible with the moving mesh model.
- The time step required for the moving mesh analysis was much shorter than the time step of the final CFD models.
- The moving mesh model geometry could be limited to 1/3 of the final model geometry because of the symmetrical nature of the shaft geometry.

The following approach was undertaken to assess the effects of lift car movements:

1. A lift car was placed at 1/3 of the lift shaft air-only domain;
2. The entire movement of the lift car from its starting position up the lift shaft was simulated in a transient fashion up to the lift car located on the top level;
3. The effect of the car movements on airflow patterns and convective heat transfer coefficient (HTC_{conv}) within the lift shaft was analysed and key parameters extracted (CD_Adapco, 2009);
4. Using the lift car movement results, the effect of the lift car movements was included within the lift shaft analysis.

This approach allowed us to assess the effect of lift car movement on the lift shaft environmental conditions, whilst using adapted computational methods to provide engineering advice within a timely fashion.
The velocity of the lift was modelled as per equation 1 below:

Equation 1: Definition of the velocity Vector of the lift car

$$V\begin{matrix} x \\ y \\ z \end{matrix} = \begin{matrix} 0 \\ 0 \\ MIN(4 * Time, 8) \end{matrix}$$

The time step employed on the moving mesh model is 0.01sec.

Discussion and result analysis

The figure below shows the effect of the lift movement on the convective heat transfer coefficients measured as area averages over the lift whole surfaces.

The HTC_{conv} are between 4 and 5 times higher when the lift is moving at design speed compared to when the lift is not moving and the only air movement is due to ventilation.

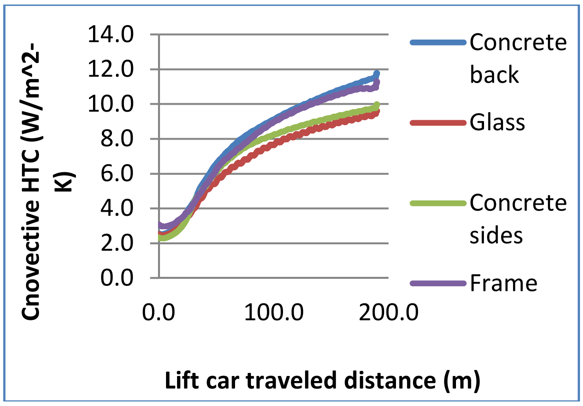

Figure 3: Area averaged convective heat transfer coefficient for the different surfaces in contact with the air in the shaft

BOTTOM UP VS TOP DOWN VENTILATION ANALYSIS

Simulation

The goal of this simulation was to finalise all the modelling by running models that took into account the lifts motion impact on the heat transfer and comparing ventilation scenarios that were possible to implement on the real shaft:

- Bottom up shaft ventilation (BUV) i.e. supply of air at the bottom of the shaft and exhaust at the top of the shaft ;
- Top down shaft ventilation (TDV) i.e. supply of air at the top of the shaft and exhaust at the bottom of the shaft ;

The glass was chosen due to a combination of high performance, high Visible Light transmittance and reasonable cost. See Methodology section for glass properties.

To maintain a practical working tolerance with the lift manufacturer 40°C temperature requirement, the modelling will be based on a maximum shaft temperature of 35°C.

In the simulations, the HTC_{conv} were "artificially" increased accordingly to reflect the impact the lifts would have on the heat transfer. However no lift is actually moving thus the mixing effect of the lifts is not accounted for.

The design criteria are set as follows:

- The outside temperature was set to be constant at 32°C in line with the general building design criteria for its location.
- The Solar Radiation settings for the model were peak solar radiation of 800 W/m2 direct radiation and 100W/m2 of diffuse solar radiation.

The models are set up to measure the temperature extremes in the shaft, the frame surface temperature, the condensation risk on the glazing near the ventilation inlets.

The models are run for supply flow rates at 11°C for two flow rates predetermined using hand calculation approximations. The temperature of the shaft are measured for both simulations, and the flow rate required to maintain the shaft below 35°C is determined with a linear approximation.

Discussion and result analysis

BUV models were run steady state. The temperature distribution is highly stratified and the highest temperature is easily measured at the top of the shaft.

TDV were run transient with a 1.2s time step based on Nyquist criteria. The flow looked like a column of smoke going downwards. Without lifts actually moving in the model, convective currents tended to go downward in the side shafts where the cooler concrete walls provided little resistance to the flow and upward in the middle shaft.

The figure below shows the cooling flow rates required to maintain the thermal condition in the shaft.

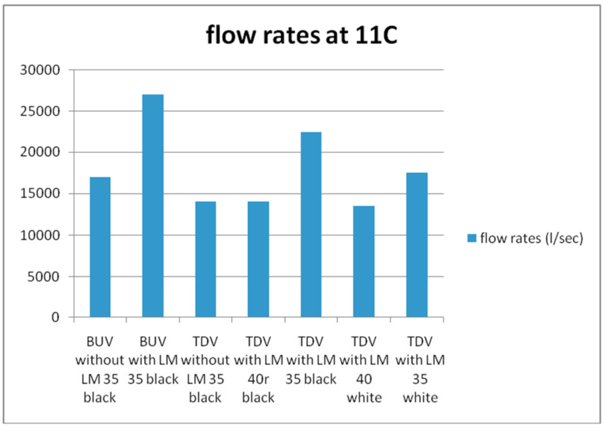

Figure 4: Cooling flow rates summary chart

where:
BUV: Bottom up Ventilation
LM: Lift Movement
TDV: Top Down Ventilation
35 or 40: maximum time average temperature at any point of the shaft more than 0.5m away from the nearest surface.

At this stage, the design team took the design to go ahead with the TDV approach.

TRANSIENT TOPDOWN VENTILATION ANALYSIS

Simulation

The goal of this simulation was to take into account the thermal mass of the concrete wall in the determination of the cooling flow rate. This could only be done using a transient model since thermal mass is by definition a transient thermal property. So far we had managed to take into account all major parameters of the analysis apart from it. A couple of months after finalising the project report we purchased a super computer along with an infinite number of core licence for our CFD software. It was now feasible to run a TDV over a whole day with a 1.2s time step.

The modelled scenario is based on the following parameters:

- From 7am until 3pm, diffuse solar radiation only is applied in the model. The diffuse solar radiation is set to 610 W/m^{2,} which is the highest diffusing solar radiation recorded in the Brisbane TRY weather file. Only 305 W/m^2 is reaching the shaft due the building shielding the shaft from half of the sky hemisphere.
- From 3pm until 7pm, the solar radiation is set to a combination of 800 W/m^2 of direct solar radiation and 100 W/m^2 of diffuse solar radiation, which is the highest global solar radiation recorded in the Brisbane TRY weather file. Both in the model and in real life, the actual amount of solar radiation

that reaches the shaft varies with the solar position.

The initial conditions for the transient simulation are taken from the results of a steady state model which represent a particularly hot night time conditions for Brisbane with an outdoor temperature of 28°C and an indoor temperature of 25°C.

Table 4: Summary of Boundary Conditions

TIME PERIOD		**TEMPERA-TURE (C)**		**SOLAR RADIATION (W/M2)**	
Start	End	Outside	Inside	Diffuse	Direct
7pm	7am	28	25	0	0
7am	3pm	32	23	610	0
3pm	7pm	32	23	100	800

The sun angle is based on the 1st of March. This date was selected based on the longest period of the day with an incident angle of direct solar radiation to the shaft below 30°.
The cooling flow rate control strategy is as follow: the cooling flow rate is set proportionally to the global solar radiation that hits the shaft compared to the design conditions in section 4 of this report. For example, section 4 peak cooling capacity is 10.6 l/s/m^2 for 850 W/m^2 global solar radiation. During the morning of this transient scenario, the global solar radiation that hits the shaft is 305 W/m^2, thus the cooling flow rate is set to 10.6 l/s/m^2 *305/850= 3.8 l/s/m^2.

Discussion and result analysis
The figure below shows the average temperature of the different shaft elements against time.

Table 5: Transient Temperature results

Time	**7am**	**3pm**	**7pm**
Temperature in Air Shaft (°C)	27	30	30.5
Temperature of Concrete (°C)	27	29	30
Temperature of Glazing (°C)	28.5	34.5	39.5
Temperature of Steel Frame (°C)	27.5	31.5	32

It clearly appears that the 10.6 l/s/m^2 cooling flow rate will be enough to maintain the average air temperature in the shaft below 35°C. In fact it appears that the 10.6 l/s/m^2 flow rate is able to maintain an average temperature of 31 °C leading to the conclusion that 10.6 l/s/m^2 is probably about 30% oversized.

CONCLUSION

Key findings of this analysis include:

- The lift movements within the shaft increased the convective heat transfer by a factor of 5 within the shaft, providing a significant effect on the U-value and SHGC of the façade,
- The final top-down air distribution is about 30% more energy efficient than a bottom-up air distribution to maintain a set maximum temperature in the shaft.
- Transient CFD modelling analysis over a whole design day (physical time) resulted in an additional 30% of cooling load reduction compared to a steady state model.

As expected, whole building thermal modelling and CFD modelling lead to dramatically different results. The thermal modelling assumptions are clearly not valid in large shafts.

Transient CFD should be used from day 1 of the modelling work on such shafts in order to include all the parameters that have a large impact on the cooling load. As demonstrated in this paper and expected, transient CFD leads to considerably lower cooling loads. For such applications, transient CFD analysis costs are recouped straight away by the lowering cooling plant capital costs.

ACKNOWLEDGEMENT

Christian Kohler and Robin Mitchell, Lawrence Berkeley National laboratory, University of California, US, for their helpful anwers to all our questions on Therm and Window 5.

Peter Ewing, CD-Adapco Australia, for his outstanding and practical help with the software.

Matt Dennis, for his help with the papers graphics.

REFERENCES

CD-Adapco. 2009. STAR-CCM+ software User Manual for versions 4.06 and 5.02

Department of Environment of NSW. 2011. Nabers Energy rating. Available: http://www.nabers.com.au/page.aspx?cid=533&site=2. Last accessed 29/08/2011.

Green Building Council of Australia. 2011. Green Star office rating. Available: http://www.gbca.org.au/green-star/rating-tools/. Last accessed 29/08/2011.

Integrated Environmental Solutions. 2009. Virtual Environment software User Manual for versions 5.0.