FORUM

Achieving air-change effectiveness for Green Star IEQ-2 Office Design with CFD simulations: Diffuser performance

Colin Allison, AP.AIRAH, of Simultude   Ben North, M.AIRAH, of Waterman AHW

ABSTRACT

There has been a substantial motivation for building developers to apply for a Green Star rating under the Green Building Council of Australia's (GBCA) Green Star rating scheme. One category is the IEQ-2 demonstrating adequate air change effectiveness (ACE) at the design stage. For systems that are not of displacement ventilation type, this can only be demonstrated through computational Fluid Dynamics (CFD) modelling of the ventilation system and calculated in accordance with ASHRAE F25-1997 methodology. Extensive CFD modelling of actual case studies has produced some interesting findings regarding the current beliefs of designers to achieve ACE. In particular this relates to how the ceiling diffusers are modelled. There are currently no Green Star guidelines available to assist CFD modellers with how to correctly perform this. Therefore CFD studies are not compared on an equal basis. Also, the GBCA rating tool prescribes conditions which seem to achieve a contrary objective of the Green Star intention. In light of these shortcomings, it is suggested that there be a review of the current Green Star rating tool for the IEQ-2 credit.

Keywords — Computational Fluid Dynamics (CFD), Air Change Effectiveness (ACE), Indoor Environmental Quality (IEQ), Local Mean Age of Air (LMA, Green Building Council of Australia (GBCA)

INTRODUCTION

It is apparent that there are certain ill-conceived perceptions regarding ventilation designs best suited to achieve the required ACE in buildings. For instance a significant number of ventilation designs applying for IEQ-2 specify swirl diffusers as well as slot return air light troffers. However, the results of numerous CFD simulations of ACE using the commercial FloVENT CFD software, suggest that this is not an ideal design solution. While swirl diffusers have a high air entrainment rate at nominal flow rates, this does not necessarily apply at lower turndown ratios typically used for variable air volume (VAV) systems. Therefore the advantage of choosing swirl diffusers in order to enhance the chances of achieving the required ACE is lost. Regrettably, simulations must be performed at the minimum turndown ratios to comply with the credit criteria, and at these flow rates, the diffuser performance is uncertain. Obviously diffuser performance is a key element of the overall design effectiveness. If the virtual diffusers in a CFD model are incorrectly calibrated then the results of the study are ineffectual. The incorrect calibration of diffusers is likely, due to the lack of performance data at low air flow rates. These shortcomings raise reservations over the practicality and validity of performing these studies in the first place.

Furthermore, the simulations show that short circuiting of supply air into the ceiling void via the return air slots reduces the ACE. Therefore the specification of return air (RA) slots integrated into light fittings is not only expensive, but actually minimizes the ACE.

AIR CHANGE EFFECTIVENESS

According to the GBCA's Green Star requirements of IEQ-2 version 2 (and now version 3), two points can be achieved if it can be demonstrated that 90% or 95% of the Net Lettable Area (NLA) will have an ACE of greater than 0.95 when measured in accordance with ASHRAE F25-1997 (1). These measurements must be performed at minimum turndown ratios for a VAV system, at a height of 1m above floor level. Either cooling or heating mode can be selected for performing the simulations. The results of numerous CFD simulations performed in cooling mode, are presented in order to provide some insight to assist in the design of effective distribution systems. Of course a CFD analysis will still be needed to ratify the design for Green Star points, but at least the air distribution concept ought to have a greater chance of achieving the required ACE.

MODEL CONSTRUCTION

There are currently no guidelines or demonstrative requirements issued by the GBCA, pertaining to this type of simulation. It stands to reason that the subject is open to interpretation and could be very easily manipulated to the advantage of the designer to achieve the favourable result. These factors serve to undermine the intentions of the GBCA rating tool.

There are a few considerations to be taken into account when performing a building CFD simulation. The specification of internal heatloads, diffuser calibration, and the modelling of the ceiling void all have an important affect. Most importantly it is essential that the CFD code be capable of realistically simulating diffuser performance, since this is the basis of air change.

It will be shown that diffuser performance in a CFD model can be highly speculative. When there is a significant difference in temperature between supply air and room air, coupled with low discharge velocities, the buoyancy forces become more dominant than momentum forces. The fidelity of the turbulence model is imperative in its ability to simulate this flow transition.

The building model should be accurate enough to include architecture that may influence air flow. Columns, exposed trusses, beams or bulkheads where applicable, should all be included. Beams and bulkheads especially are highly influential on air distribution since the supply air is normally discharged at high level. Also important is the ceiling void. This is because the return air path is critical in determining the air change effectiveness. Typically the ceiling void is used as a return air plenum and therefore the flow distribution within the ceiling influences the ACE. Of course it is impractical to model the entire internals of the ceiling void including ducts, cable trays, sprinkler pipes, trusses etc. since this information is not readably available, and would unnecessarily complicate the model.

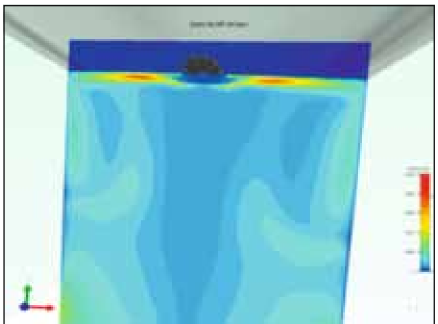
DIFFUSER MODELLING

The computational grid occurring throughout the domain, in this case the particular floor level being modelled is typically meshed with varying cell sizes. This is done for practical reasons, to limit huge demands on computational resources. In the occupied spaces the grid can be enlarged, but around diffusers, the mesh is refined in order to accurately capture the higher velocity and temperature gradients. Typically, the detailed diffuser geometry is not modelled since the entire model becomes far too complex and the meshing requirements prohibitive. Instead, depending on the sophistication of the CFD code, basic geometric shapes such as cubes or discs are used to represent the diffuser. Velocity profiles are then specified at the diffuser faces. This is an approach adopted by Nielson (2) (3).

He used measured data to specify velocities parallel to the diffuser face. However, it was too simplistic and did not adequately represent the variation in velocity profiles across the entire face of the diffuser. To overcome this Nielson (4) also proposed an alternative method called the velocity prescription method. In this case the velocity boundary conditions are specified as before, however, an additional measured velocity component is specified within the box boundary region to correct the prediction of velocity around the diffuser area. Both of these methods require measured data from each diffuser and are therefore impractical to apply. Chen and Moser (5) proposed a new momentum method in which the velocity vector is calculated based on the effective area, and not the opening area to have the correct velocity description. In order to maintain unity of supply air flow rate and air entering the room, it was necessary to describe the continuity and momentum equations independently. Chen et al (6) and Jiang et al (7) used the momentum method on detailed diffuser geometry, and reported good agreement with experiment, but most commercial CFD codes do not support the separate description of boundary conditions for continuity and momentum equations. Chen and Jiang (8) simulated a two-dimensional diffuser with complex geometry. While it was possible to describe accurately the boundary of a complex diffuser using state of the art techniques

Krantz RA-N3-350 Swirl Diffuser mounted in Krantz smoke test chamber ceiling, with collar size 3

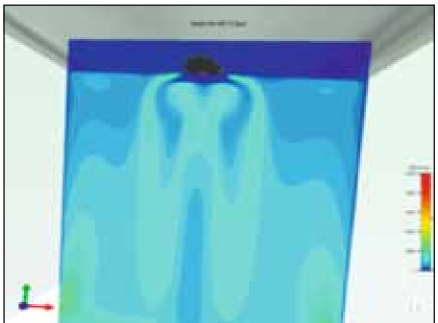
Several seconds after smoke discharge into chamber, 95 l/sec, – 12K Delta T



FloVENT Fan model Swirl Diffuser KETurb contours: Steady State solution, 160 l/sec, – 10K Delta T

Approximately 5 seconds after smoke discharge into chamber, 28.5 l/sec, – 12K Delta T

Approximately 20 seconds after smoke discharge into chamber, 28.5 l/sec, – 12K Delta T



FloVENT Fan model Swirl Diffuser KETurb contours, Steady State solution, 72 l/sec, – 10K Delta T

Figure 1: Selected images from Krantz video footage of Swirl diffuser smoke test performance, compared with KE Turbulence plots of the FloVENT fan model swirl diffuser at nominal flow and at 45% turndown.

the computational resources were prohibitive on a practical scale. Having reviewed these various approaches, Emvin and Davidson (9) concluded the following. Full representation of the diffuser is useful but expensive and very time consuming. The momentum method can give qualitative results but cannot properly represent the entrainment of the supply diffuser and may only work on uniform meshes. The box model is consistent and supposes to perform as well as the full representation but requires measurements. Einberg et al (10) compared measurements with a CFD simulation of a swirl diffuser using the commercial CFD code FLUENT. They simplified the diffuser geometry by using a disc, and specified a uniform velocity on the edge of the disc only. The velocity was given a horizontal twist angle but the axial component was ignored. The supply air was only 4 degrees below room ambient so approaching an isothermal jet condition. They used the Renormalization Group theory k-epsilon turbulence model, but had to include some additional swirl modification. Comparison with the measured results showed that while there was good agreement in velocity decay close to the diffuser, this agreement decreased rapidly with distance from the diffuser. Temperature comparisons were even more differing. This was attributed to deficiencies in the momentum calculation of the CFD model. They suggested that more sophisticated turbulence models such as the RSM (Reynolds Stress Model) or LES (Large Eddy Simulation) would produce better results. However, the RSM model is computationally intensive and employs 6 equations for Reynolds Stress Transport rather than the 2 equation k-epsilon type turbulence model. Likewise the LES turbulence model requires a transient analysis and a much finer grid. Hence the computational resources are prohibitive and not practical for most applications. Finally Huo et al (11) have developed a variation of the box method called the Jet Main Region Specification method. Here the analytical data is applied to a boundary surface located within the main jet region. This method shows significant improvement over the conventional box method.

Clearly diffuser modelling is not a trivial exercise. The modelling results can be just as capricious as the variety of methods used to specify the diffuser velocity discharge. Furthermore, there are other factors that affect diffuser discharge patterns such as the internal geometry from support brackets, and the size and orientation of the supply duct and balancing spigot position. Clearly some of these factors are beyond being included in a CFD model, either for practical reasons, or because they are unknown. And let’s not forget about the difference of diffuser performance between cooling and heating modes. Based on this perplexity, it is suggested that if the ventilation effectiveness is demonstrated using a global approach with generic diffuser performances, than minor variations in diffuser performance are not important. In this way ACE designs can be compared on an equal basis, so that the imperative factors such as quantity and location of diffusers or return air grilles can be assessed.

The commercial CFD code FloVENT has specially derived diffuser smart parts. The diffuser component is made up of a number of different smaller parts. In this way the velocity and airflow in each subdivision can be described independently. This provides more versatility in calibrating the entire diffuser flow as opposed to a single discharge surface. In the case of a square diffuser it can be modelled as a thin cuboid attached to the ceiling. The discharge velocity is determined by a specified discharge angle, and flowrate based on the effective area. These parameters are adjusted until the velocity discharge pattern matches that specified in the manufacturer's catalogue. A swirl diffuser is more complex because it has an axial, radial and tangential component. Various flow rates will result in the flow being either discharged horizontally across the ceiling, vertically downwards towards the floor or somewhere between these two extremes. It is essential to accurately calibrate the diffuser, as it will be seen that the resulting ACE depends on whether the discharge is downward or horizontal. Swirl diffusers are more difficult to calibrate because there is a dearth of information specifying the performance at nominal flow rates, let alone at minimal turndown flows. Swirl diffuser performance cannot be categorized in terms of throws and terminal velocities. In fact, diffuser manufacturers Krantz do not publish tables of throw and terminal velocity since in the context of swirl diffusers, this information is meaningless. Instead, the swirl diffusers for this project were calibrated using video footage of smoke tests provided by Krantz.

Figure 1 shows a performance comparison between the swirl diffuser tested in a smoke chamber at operating conditions specified for the Centrelink Canberra Building perimeter zone (12), and the corresponding CFD calibration simulations. The simulations were performed using a virtual test chamber which had the same dimensions as the Krantz test chamber, so that wall effects were equally accounted for. A 3D fan model with 8 facets and a 76mm hub was used. The swirl was specified from the standard fan construction dialogue by selecting a vertical deflection of 35 degrees and a twist of 45°C. Room design temperature was 22.5°C and perimeter supply air temp was 12.7°C. At the nominal flow rate of 160l/s, the flow is discharged rizontally along the ceiling and remains there until it hits the walls of the chamber. In the case of 45% turndown the CFD model settings were unchanged except for the turndown flow rate of 72l/s. At this condition the discharge flows initially along the ceiling. However, without sufficient momentum, there is less coanda effect to maintain as protracted contact with the ceiling. Approximately 20 seconds from initial discharge, the slower moving air separates from the ceiling and cascades gently downwards. Subsequent footage (not displayed) shows that upon reaching the floor it spreads out building up a layer of young fresh air. This layer of fresh air actually has a positive effect on the resulting ACE for two reasons. Firstly, the younger air is delivered directly to the 1m level where ACE is measured. Secondly because it doesn't travel predominantly along the ceiling it has a much smaller chance of short circuiting back into the ceiling void via return slots.

Krantz diffusers are generally regarded as having good induction performance. In fact internal test results (13) have determined that they have induction ratios of up to 31:1. However, paradoxically, in a strange irony, diffuser dumping will actually contribute to a higher ACE. This is because ACE is measured at low level. Diffuser dumping delivers the fresh air to the measurement level sooner. Surely this outcome is not what is anticipated by the intentions of the GBCA IEQ-2. This phenomenon is demonstrated in the following case study.

ACE CASE STUDY: NORTHBRIDGE BRISBANE

It should be noted from the outset that the modelling results presented in this case study have not been compared with any physical measurements, since these were not available. In most cases the purpose of conducting modelling is to curtail the need to facilitate expensive and often impractical physical measurements. The results are therefore not completely verified but rather intended to stimulate discussion on the topic of applying CFD modelling to ventilation systems in buildings.

Figure 2 shows the typical floor plan of the Northbridge (now Santos Place) office tower – 32 Turbot Street, Brisbane, Qld, which has achieved a 6 Green Star rating under the Commercial Office Design rating tool. Applications are under way to achieve the same for the "as built" credits. The building is mechanically ventilated with the use of Krantz RA N3 350mm diameter swirl diffusers. The return air path is via light troffer return air slots and several egg crate grilles positioned around the core areas. The air is drawn to return air ducts located above the toilet block towards the rear of the core. There are four air handling units (AHUs) serving the NW, SW SE and centre zones.

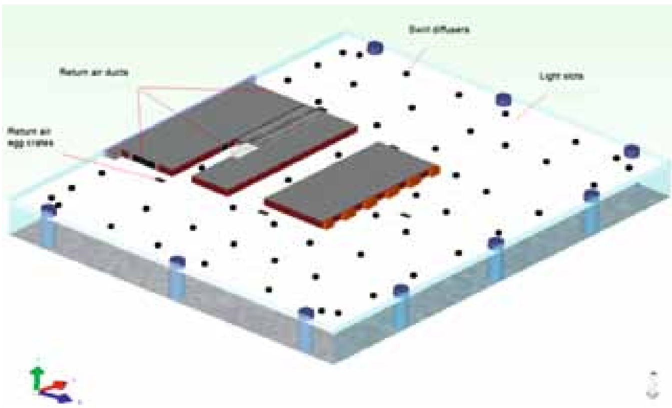


Figure 2: Northbridge Level 7 to Level 18 Typical Floorplan

The mechanical design specified a VAV system using swirl diffusers with a 40% turndown ratio. In Figure 3 to Figure 5, the simulated ACE values are shown at heights of 1500mm, 1000mm and 500mm above floor level. The ACE0.95 criteria of 95% is expressed as Local Mean Age of Air (LMA) corresponding to the zone time constant, which in this case is 1385s. It is plotted on a two colour scale so that the area of compliance can be easily distinguished. In this case the blue zones have a LMA0.95 lower than the zone time constant, and the red zones have a LMA0.95 which exceeds the time constant. Clearly, the ACE improves at lower levels. This is because diffuser downwards flow of supply air allows a layer of fresh air to be built from the floor upwards. In addition, downwards flow minimizes the possibility of supply air short circuiting back into the return air via the light slots. Supply air short circuiting compromises the ACE because it implies that elsewhere aged air is not withdrawn as it should be. So while diffuser downwards flow improves the ACE, it more than likely has a detrimental effect on thermal comfort requirements. This may contradict the intentions of the GBCA Green Star rating.

Figure 3: LMA at 1500mm affl

Figure 4: LMA at 1000mm affl

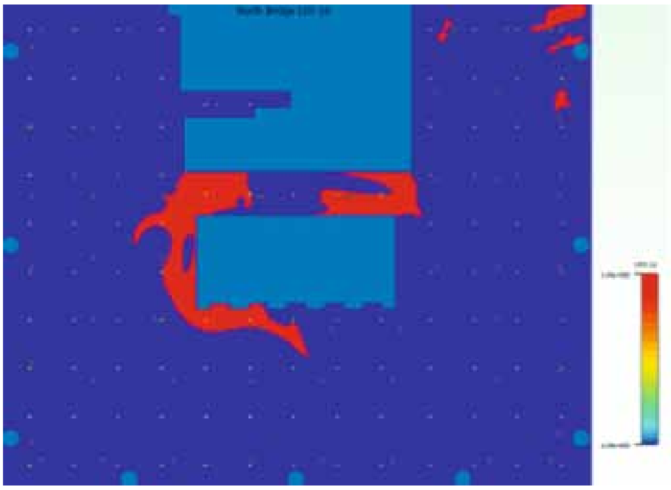


Figure 5: LMA at 500mm afflremoved

Exploring this further, and performing the simulation at the maximum flow rates, an entirely different ACE is achieved. The following graphics show the ACE at the maximum (ie no turndown) diffuser air flow rates. In this case the LMA has been adjusted to account for the increase in air supply and the corresponding time constant

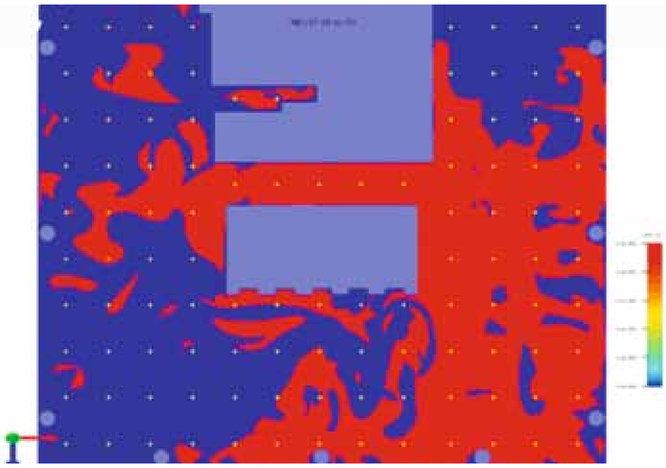


Figure 6: ACE at 1000mm height at maximum diffuser flow rates

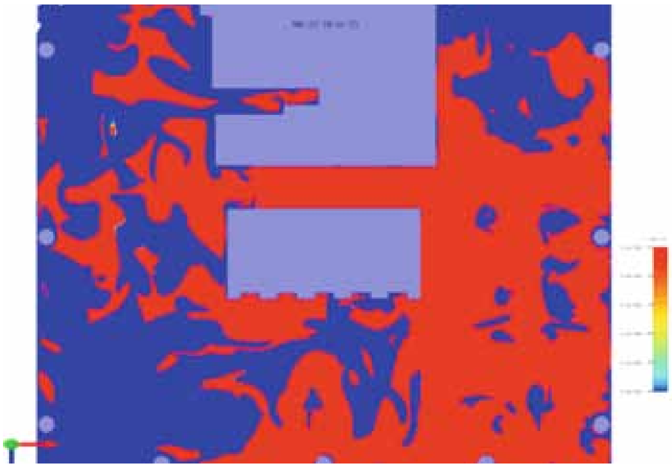


Figure 7: ACE at 2000mm height at maximum diffuser flow rates

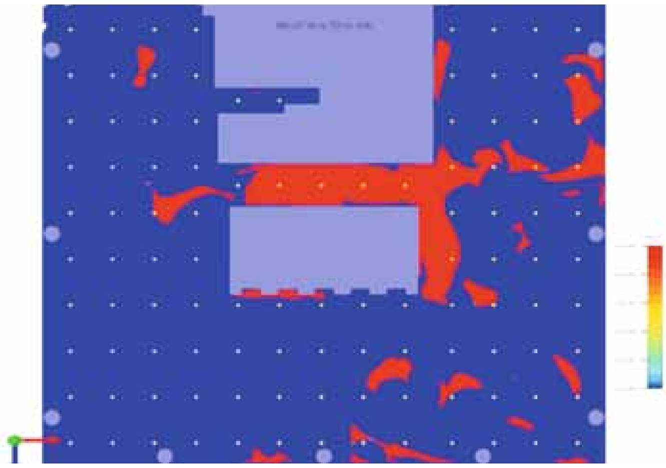


Figure 8: ACE at 1000mm height at maximum flow rates, with slots removed

is 623s. Figure 6 and Figure 7 compare the ACE0.95 areas at a height of 1m and 2m above the floor. The ACE in this case has worsened significantly, achieving only about 55% area compliance at 1m and the same 2m above the floor. The implications of this appear substantial. While the specified ACE is achieved at minimum flow rates, it is demonstrably poor at maximum flow rates. It could be argued that the optimum ventilation efficiency should be achieved at maximum flow rates, when the building experiences maximum heat loads or occupancy loads and when maximum fresh air quantities are supplied. The waste of energy due to poor ACE is obviously larger at maximum flow rates. Of course some will argue that maximum flow rates only occur over a small percentage of the time. Nevertheless, it can be demonstrated that by achieving a good ACE at maximum flow rates at the 2m level, it will almost certainly guarantee a good ACE at the 1m level. Unfortunately the converse cannot be said.

Finally, it can be shown that to immediately improve the ACE of the prior case of maximum flow rate, the return air slots should be eliminated. Figure 8 shows this situation where the return air is drawn into the ceiling void entirely by the 5 egg crate return air grilles located around the central core region. The ACE0.95 at the 1m level is greatly improved and this is now a good design achieving compliance over about 80% of the floor area. An ACE of 95% area would appear unrealistic and virtually impossible to achieve under these conditions. Note however, that by eliminating the slots, the ACE at minimal turndown rates and measured at the 1m level will also improve to above 100% of the area. Unfortunately as demonstrated, these results are arguable as to their meaningfulness.

CONCLUSIONS

It has been demonstrated that it is important to correctly model the performance of the diffuser as this is tantamount to air diffusion and hence ACE. The GBCA guidelines for measuring ACE would appear misguided and it is suggested that they be reviewed and amended to focus on the maximum flow rates rather than the minimum. Under these conditions, ACE values of 0.95 at the 1m level appear impractical and very difficult to achieve. It is suggested that a good ventilation design should be capable of achieving an ACE of around 75%-80% at the 1m level. It is suggested that return air slots in light fittings are counterproductive in achieving good ACE and their usage should be questioned carefully in a design attempting to achieve this. ▮

ACKNOWLEDGEMENTS

The authors would like to thank Krantz Pty LTD for supplying video footage of the swirl diffuser performance. They would also like to thank FlowSim Pty LTD for providing the FloVENT CFD code.

REFERENCES

1. ASHRAE. *Fundamentals. 1997.*
2. *The box Method-a Practical procedure for introduction of an air terminal device in CFD calculation.* Nielson, PV. Denmark: Aaborg University, 1997.
3. *The velocity characteristics of ventilated rooms.* Nielson, PV, Restivo, A and Whitelaw, J H. 1978, Journal of Fluid Engineering, Vol. 100, pp. 291–8.
4. *Description of supply openings in numerical models for room air distribution.* Nielson, PV. 1992;98(1), ASHRAE Transactions, pp. 963–71.
5. *Simulation of a multiple nozzle diffuser.* Chen, Q and Moser, A. Ottawa: 12th AIVC Conference, 1991. pp. 1–14.
6. *A database for assessing indoor air flow, air quality and draught risk.* Chen, Q, Suter, P and Moser, A. s.l.: ASHRAE, 1991. Vol. 97(2), pp. 150–163.
7. *A Comparison of displacement and mixing diffusers.* Jiang, Z, Chen, Q and Moser, A. s.l. : Indoor Air, 1992,2(3), pp. 168–79.
8. *Simulation of a complex air diffuser with CFD technique.* Chen, Q and Jiang, Z. Yokohama : Fith International Conference on Air Distribution in Rooms, Room Vent'96, 1996. Vol. 1, pp. 227–234.
9. *A numerical comparison of three inlet approximations of the diffuser in Case E1 Annex20.* Emvin, P and Davidson, L. Yokohama : s.n., 1996. Fith International Conference on Air Distribution in Rooms, RoomVent'96. Vol. 1, pp. 219–26.
10. *CFD Simulation and measurements in near zone of a high induction swirl diffuser.* Einberg, G, Koskela, H and Holmberg, S. Coimbra : s.n., 2004. Roomvent 9th International Conference.
11. *A systematic approach to describe the air terminal device in CFD simulation for room air distribution analysis.* Huo, Y, et al. Building and Environment, Vol. 35(2000), pp. 563–576.
12. *Krantz Swirl Diffuser* KPS4577 Video Archhive. Krantz Germany, 2008.
13. Krantz PTY Ltd. *Difusser Induction: Dadanco IDL20*; Krantz IN-V2.2;Krantz Swirl. s.l.: Krantz, 2005.

About the authors

Colin Allison (PhD) is the director of Simultude, Thermal Fluid Modelling, which provides CFD simulation services to the industry using the commercial CFD code FloVENT. During his PhD candidature at the University of Adelaide, he performed extensive CFD simulations of air flow and heat transfer through various heat exchanger surfaces. Email: callison@simultude.com.au

Ben North is the regional director of Waterman AHW(Qld), a consulting engineering company providing engineering and ESD design and consulting to the construction industry. With a background in mechanical engineering design and multi-discipline engineering project management, he has experience in various sectors ranging from institutional, commercial office and residential, to industrial manufacturing and technology. North holds a RBP (Vic), BSA (Qld) is an OMIEAust, AM.AIRAH and M.ASHRAE. Email: BNorth@wahw.com.au