EXPLORING RAPID PROTOTYPING TECHNIQUES FOR VALIDATING NUMERICAL MODELS OF NATURALLY VENTILATED BUILDINGS

Tong Yang, Richard A. Buswell and Malcolm J. Cook
Building Energy Research Group, Civil and Building Engineering, Loughborough University, Leicestershire, UK

ABSTRACT

An alternative to using numerical simulation to model ventilation performance is to model internal air flows using water-based experimental models. However, these can be time consuming and the manual nature of model assembly means that exploring detail and design variations is often prohibitively expensive. Additive, or Rapid manufacturing processes can build physical models directly from 3D-CAD data and is widely used in product development within the aero-automotive and consumer goods industries. This paper describes ongoing work exploring the application of such techniques for the production of physical models which can be used in their own right in water-based testing or for Computational fluid dynamics (CFD) validation. The findings presented here suggest such techniques present a worthwhile alternative to traditional model fabrication methods.

INTRODUCTION

Modelling natural ventilation is taking on an increasingly important role as sustainable building design solutions push the boundaries of the technique further. Using numerical simulation to model air flow in and around buildings is now regularly used as part of the design process. However, these models require validation and are sometimes unable to predict some of the more complex, time varying phenomenon.

Water can be used to model natural ventilation flows by using a salt solution (brine) to create a density difference relative to the water (Linden et al. 1990). This technique involves submerging small perspex models of a building, typically at a scale of between 1:20 and 1:100 into a larger tank of fresh water. Since brine is denser than water, the buoyancy forces acts downwards, so the perspex models must be inverted to enable a flow to be driven through the space. Using dye to colour the brine, an inverted image of what happens when a heat source drives a flow in air can be produced. Flow visualisation is an important part of salt bath modelling experiments. In most cases, the brine injected into a model is coloured using dye. This highlights areas of the flow where denser fluid is present, i.e. where warmer air would be present in a building at full scale.

Various modes of steady-state and transient buoyancy driven flows have been tested using the salt bath experimental techniques and analytic models have been developed and validated alongside for simple rectangular box geometries (Linden et al. 1990, Linden and Cooper 1996, Kaye and Hunt 2009, Hunt and Coffey 2010). Computational fluid dynamics (CFD) modelling is a valuable tool for undertaking parametric studies where a range of tests are needed in which small modifications between simulations are made. However, due to shortcomings associated with turbulence modelling, transient modelling, it is important that CFD predictions are validated rigorously, testing features such as stratification, flow patterns and flushing times.

One disadvantage of salt bath modelling is the time required for model production. This is very similar to manufacturing prototyping issues found in the aero/automotive and consumer goods industries. Over the past 30 years these industries have been developing rapid prototyping devices to reduce the design cycle time: Rather than producing a prototype through manual craft, rapid prototyping machines are able to produce a physical component directly from 3D CAD data, thus reducing the time for model production from days or weeks to hours. This vastly speeds up the design evaluation time, hence the name *rapid* prototyping. This field has since grown into what is now widely termed *additive manufacturing*, where not just prototypes, but end use parts are manufactured directly (Wohlers, 2004).

METHODS

Additive Manufacturing

There are a family of names used to describe essentially the same type of fabrication technology; rapid manufacturing, rapid prototyping, solid freeform fabrication and more commonly now additive manufacturing. These processes all operate in a similar manner by ‘printing’ 3D structures typically in a build volume of 500mm x 500mm x 500mm, although sizes vary depending on the process. A design is usually created using 3D CAD solid modelling software and the surface is tessellated in much the same way as a Finite Element Analysis or CFD mesh is generated. This model is then virtually ‘sliced’ into multiple layers. Each layer

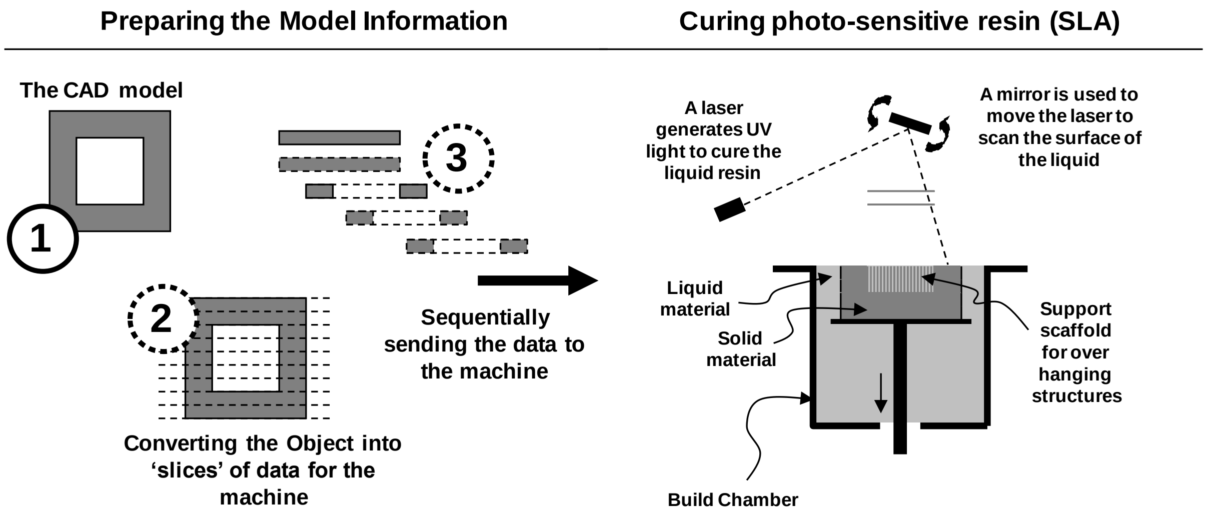


Figure 1: The principal features of the Stereolithography (SLA) process

forms a 2D plane on which some areas are solid material (the rest is not). Each slice is converted to a set of machine operations which controls the deposition of the material. As the layers build up sequentially they are fused or bonded to the last and eventually the entire 3D object is created.

There are many processes for rapid prototyping, each uses a specific set of materials: Thermojet, Selective Laser Sintering (SLS), Stereolithography (SLA), 3D printing (3DP), Fuse Deposition Modelling (FDM), are just a few. A good source of further reading is Castle Island (2011). All use different materials and vary in process. Figure 1 depicts the SLA process, which is of particular interest to this work since it can produce objects in a transparent resin, a key requirement for the visualisation aspects of the salt bath modelling technique.

The SLA process uses a range of UV sensitive resins, of which some are clear. The apparatus comprises of a horizontal mesh bed (on which the component is built) that sits in a vat of uncured, liquid resin. The mesh bed is lowered so that it is covered by one layer thickness of resign (0.1mm). A UV laser scans the surface of the resin where the solid material should be, curing it and making it solid. The mesh bed drops down to the second layer and the process is repeated until the component is complete, typically taking 12-16 hours.

After printing, some 'post processing' measures are required. The component must be baked in a UV oven to cure any excess resin that is on the surface of the model and any support structure must be removed. The viscosity of the resin will support overhangs up to about 30 degrees, which means voids in solid components less than 2mm can be created. Where larger overhanging sections are required, the part must be supported while it is cured and in the SLA process this is achieved by printing a very fine (~0.5mm) scaffold structure built up on every layer as the component is manufactured. This is simply broken off once the part is complete.

A limitation of the design is that this support structure needs to be removed, which can be problematic if very long, thin channels are formed, however, this is hugely outweighed by the virtually limitless geometrical forms that can be created. Almost any void shape in any shape of object can be created which when coupled with the fast build time opens up tremendous opportunities for the creation of functional models that can be used in the design process. In addition, once the principle geometry has been modelled, making changes and printing out alternative designs becomes trivial and opens up the iterative use of physical models in the design process that have an unprecedented level of detail and sophistication.

The following section demonstrates the possibilities of what this level of detail means for modelling an auditorium space for use in a salt bath modelling procedure. In order to fulfil the visualisation tasks associated with salt bath testing, a clear resin Accura60[1] was chosen as the model building material.

Auditorium Model

Modelling buoyancy-driven models using the salt bath approach has been reported by many researchers, including Linden et al (1990), Kaye and Linden (2004), Bower et al. (2008) and Kaye and Hunt (2009). This work has been seminal for

[1] http://www.3dsystems.com/products/materials/sla/datasheets.asp#Accura_60

determining the nature and behaviour of buoyant plumes. However, all these studies have used simple, generally orthogonal, geometries. Auditoria are designed for occupation by many individuals at various elevations and are particularly interesting in natural ventilation design because they present a number of fundamental design challenges, for example:

- ensuring stratification remains above head height;
- preventing downdraughts from outlet stacks;
- uncertainty due to solution multiplicity caused by turbulent and transient effects;
- understanding the impact on ventilation performance of a partially occupied auditorium

The detail of multiple inlets and individual occupants is difficult, if not impossible to model using traditional, Perspex style fabrication. Stereolithography offers the possibility of overcoming this by including internal channels to deliver fresh water and brine solution to many locations within the model.

The auditorium modelled here was based on a real design proposed for a college in Oxford, UK. A section with dimensions is depicted in Figure 2. The auditorium has a floor area of 14.0m×14.0m and stack height of 10.2m. Raked seating for 240 people was modelled in 12 rows of 20 people plus an additional person to represent the lecturer. In this study, heat sources from lighting and equipment are not modelled.

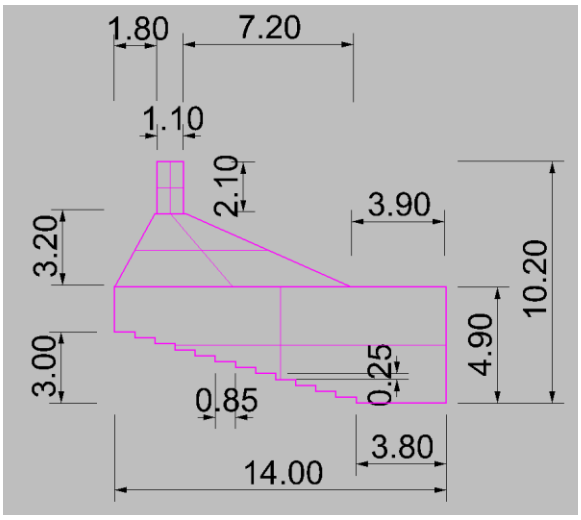


Figure 2: Section and auditorium dimensions.

A 1:50 scale model of the auditorium was built using the SLA process. At the model scale, the seating step has dimensions of 17mm in depth and 5mm in height, which is very difficult if not impossible to build using a traditional Perspex fabrication method. Minimum wall thicknesses of 2mm gave the model rigidness as well as translucency for visualisation.

The limitation in the size of the build chamber meant that a two-part construction was necessary. In addition, consideration was given to the orientation of the build to minimise the need for the support structure. This reduces the material used, build time and minimises the surface finishing required.

Figure 3 shows a 3D CAD rendering of the models and Figure 4 shows the detail that was modelled. Every seat was modelled and each individual was represented by a brine source emanating from one of the voids. Each plume is fed from a plenum that is connected to the brine/dye solution feed. In addition, at what would be ankle level, there is a secondary opening where ‘cool air’ could be introduced to investigate entrainment of outside air. These sections too are fed from dedicated plena connected to the surrounding fresh water reservoir.

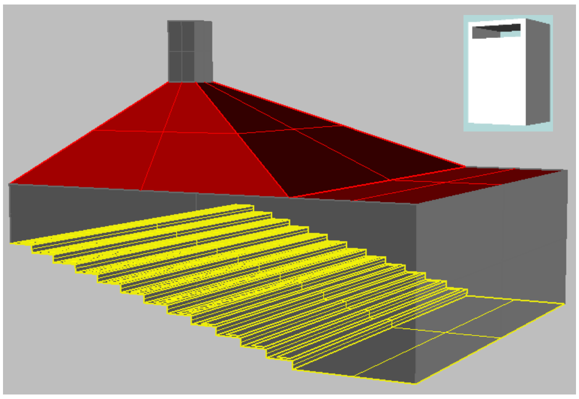


Figure 3: A CAD model of the auditorium (courtesy Michael Popper Associates LLP). Insert shows different stack opening design.

The whole auditorium model was printed in 3 parts, stack, ceiling space and seating/floor area with surrounding walls. Several design variations on stack height and stack opening sizes were considered, then individual model parts for each design were printed separately. Surgical tape (3M Blenderm) was used to join and seal model parts together to form a full auditorium model. Later, the model can be dissembled easily to enable testing of alternative geometries.

Figures 4 and 5 clearly show the level of detail that can be achieved using SLA.

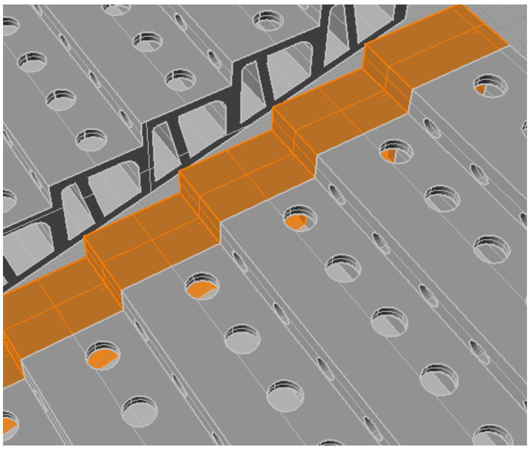


Figure 4: The CAD model detail – the built-in channels, ventilation openings and holes representing occupant heat sources and fresh air supply

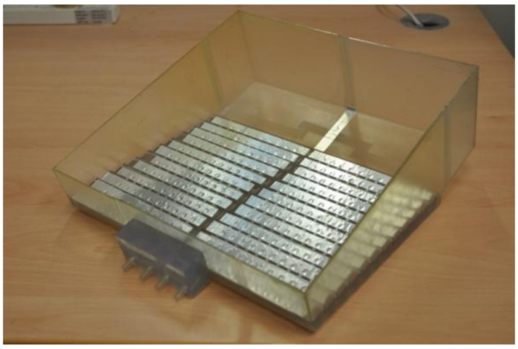


Figure 5: The SLA model detail – the seating area has four separated sections, individually connected and controlled via four supply pipes

Experiments

Saline solution was supplied via four pipes, each connected to one of the four seating sections (Figure 6). Ø6mm needle flow regulators (Legris, G1/8, D6) were fitted on each pipe to provide individual control and fine adjustment of pressure head to give the required flow rates.

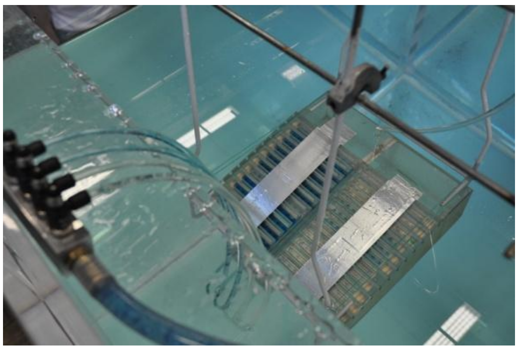


Figure 6. Fine control of supply saline solution for individual section to balance the system (Perspective view of model immersed in fresh water tank)

A saline flow rate of 1.0×10^{-6} m^3/s, was used for these experiments. This value was based on work for single plumes by Kaye and Linden (2004).

Buoyancy driven natural ventilation in the auditorium space was modelled. The issues investigated were:

- visualisation of the flow patterns;
- plume characteristics.

CFD Simulations

The full-scale auditorium space was simulated using the CFD software CFX (Ansys, 2010). It is assumed that there are 241 occupants in the auditorium and each person releases heat at a rate of 90W. Only heat transfer by convection was considered as radiation effects are not represented in the water model. Details of boundary conditions are listed in Table 1.

Table 1 CFD boundary conditions specified for the naturally ventilated auditorium

Location	Type	Value
Stack top opening	Relative pressure	0 [Pa]
	Temperature	18 [°C]
	Loss coefficient	2.69 [-]
Stack, walls, ceiling, stage, and seating steps	Smooth wall	Adiabatic
Under-seat vent openings	Relative pressure	0 [Pa]
	Temperature	18 [°C]
	Loss coefficient	2.69 [-]
Occupants	Heat release	90 [W] each

Mesh sensitivity tests were conducted based on best practice guidance on CFD techniques (COST, 2007). Temperature and velocity profiles on two vertical data lines (i.e. digital data probes) in the space were compared for three meshing configurations, namely, coarse, medium and fine (1.1, 1.6 and 2 million elements respectively). There was a maximum of 0.92°C and 0.09 m/s difference between the medium and fine meshes and less than 1.4% difference in mass flow rate predictions through openings, between medium and fine mesh. Consequently, the medium mesh was used for further simulation cases. The RNG k-ε turbulence model was used based on its widespread acceptance for accurately modelling indoor air flow (Chen, 2009).

RESULTS AND DISCUSSION

Visualisation

There was concern that, due to the slightly opaque nature of the resin used, that it would not be possible to visualise the flows without some treatment of the model surfaces. However, photographs taken using an SLR camera under typical laboratory lighting with a white screen placed behind the tank, provided good quality images which could be used to investigate qualitative aspects of the flow such as stratification and plume flow (Figure 7). It is likely that these images could be improved by using high intensity spot lights located above the tank. Other techniques being investigated include using an LED backlight on diffusive background paper, or a laser through a sharp diffusive lens to form a light sheet, which can be used to visualise the flow without the use of dye.

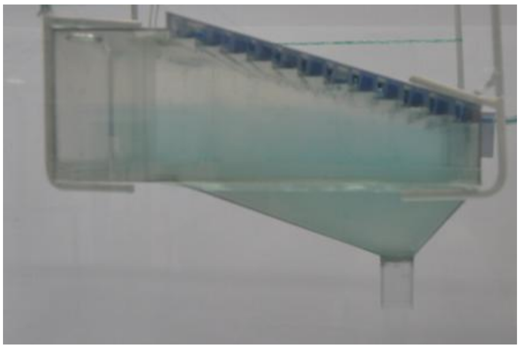


Figure 7. Flow stratification in the auditorium model (blue layer representing warm air in the auditorium space)

Plume characteristics

The experimental setup was successful in generating the required pressure head to drive a flow of brine into the plena, although it was not possible with the current configuration to establish uniform brine flow through all of the nozzles. Separate tests carried out using a single array of nozzles representing one row within the auditorium were more successful and generated buoyant, turbulent plumes as expected (Figure 8). One of the problems observed was the accumulation of dye particles in the plenum which is likely to have inhibited the flow of brine through the nozzles. This was thought to be caused by the salt reacting with disinfectant in the header tank. This will be the subject of further investigations.

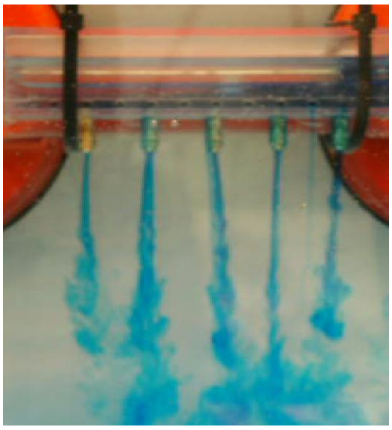


Figure 8. Turbulent buoyant plume generated in tank for one row of seating

It is likely that a much higher pressure head will need to be generated in order to generate the require flow through all of the nozzles. Alternatively, the use of one nozzle to represent a group of occupants may provide an acceptable solution.

Other Observations

Bubbles forming inside the model due to temperature changes in the tank water caused some problems for flow visualisation and inhibited the flow in some regions of the model. Future models will need to be designed to have a gentle slope to allow air bubbles to flow up and out of the model.

Some leakage of brine through seal between the plena and the main body of the auditoria was observed. This may be overcome by designing some overlaps between joint parts and more attention on surface polishing before gluing.

CFD results

Convergence was considered to have been achieved when the normalised Root Mean Square (RMS) residuals for all transport equations fell to below 5×10^{-5} and the global imbalance in momentum, mass and energy was less than 1%. In order to achieve this level of convergence, under-relaxation in the form of false time stepping was required and a high resolution[2] discretization scheme was needed (Ansys, 2010).

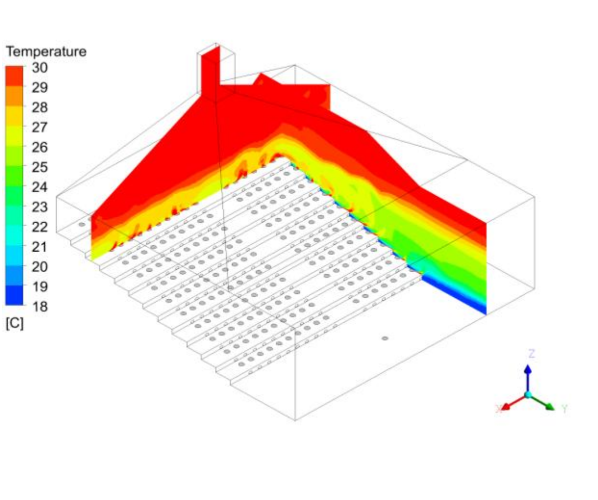
The CFD results predicted an air change rate of 5.5 ACH which is typical for this type of natural ventilation scenario. The predicted fresh air distribution and thermal stratification results are compared with the water model in Fig.9. Fig. 9(a) and 9(c) illustrate temperature variation and thermal stratification. The neutral pressure level can be found in Fig. 9(b) at the height where the inside-outside pressure difference is zero. This is located around the 3rd step from top row, which agrees favourably with the physical model results shown in Fig 9(d). The CFD simulations predict a much more diffuse thermal stratification than the water-based models. This was also observed in the work by Kaye et al. (2010) and highlights a possible limitation of the salt bath modelling technique.

CONCLUSIONS AND FUTURE WORK

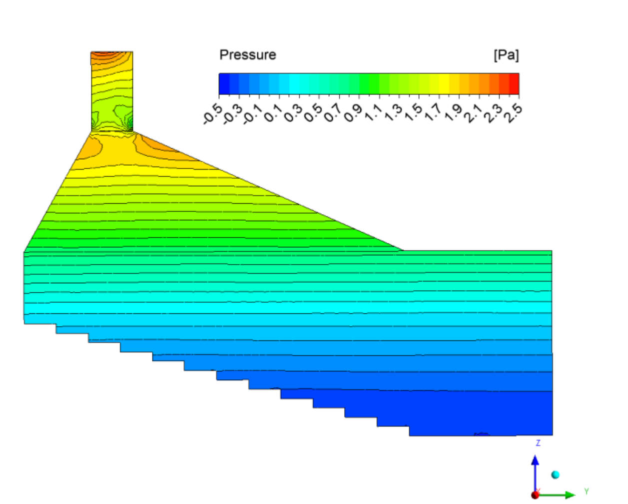
A 1:50 scale auditorium model has been designed and produced using rapid prototyping techniques and used to investigate buoyancy driven flows in water tanks. The model gave sufficient transparency to enable useful flow visualisation using a common static SLR camera.

Multiple nozzle arrays represented 241 occupants in the space were used, although it was found that the pressure head used was insufficient to ensure an

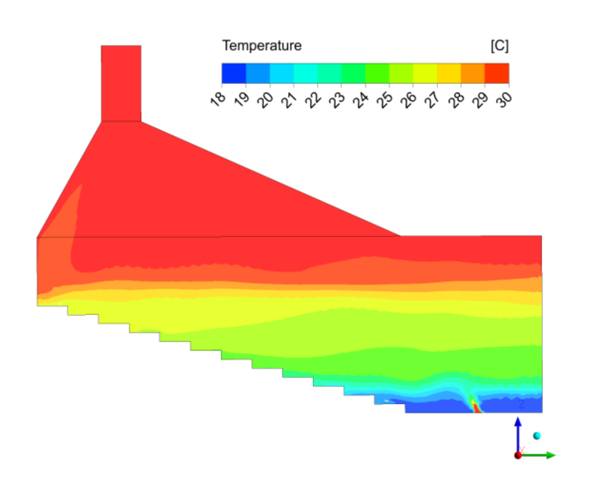
[2] A "high resolution" discretization scheme uses a local blending factor between 1st and 2nd order scheme, which provides a compromise between solution robustness and spatial order-accuracy of the discrete approximation.



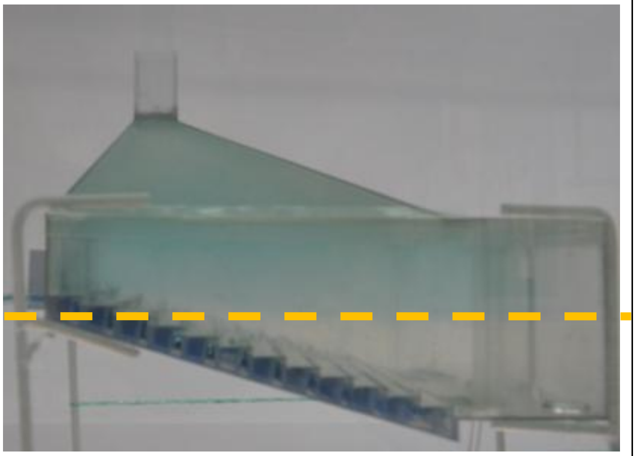
(a) Predicted temperature distribution showing thermal stratification



(b) Vertical pressure profile (neutral pressure level is indicated by the 0Pa level)



(c) Predicted temperature distribution over the central vertical plane showing thermal stratification



(d) Salt bath testing - flow stratification in the 1/50 scaled model (inverted image of Fig.7)

Figure 9. Temperature distribution and flow stratification in the auditorium – prediction by CFD and observation in the water model

uniform flow through all nozzle arrays. In order to improve this, a precisely controlled plume generation device, for e.g. a printer head strip with an electronic control system to create the desired buoyancy source strength and real-time control for transient natural ventilation scenarios could be used. Such additions to the model could be integrated into the 3D printing process.

It was also observed that, due to reactions between the salt and other substances in the fluid (possibly disinfectant), sediment formed quickly in the supply piping system and the plena, thus reducing the uniformity of the flow and the colour density of the brine. Dyes with finer particles such as nano particles could be used to overcome this problem, as well as testing alternative means of disinfection.

Future work will include tests to investigate the use of this technique to visualise the 3D structures of the buoyant plumes, e. g. turbulent eddy formation and break-up which will provide valuable data for CFD validation. The techniques to be explored will include fluorescent PIV (particle image velocimetry) and 3D stereo PIV.CFD simulations to be conducted include investigations into the differences between large area sources and multi-point heat sources and transient flows caused when operating conditions (boundary conditions) change.

ACKNOWLEDGEMENTS

This work was completed under the EPSRC/IMCRC funded Freeform Construction Project, Grant Number EP/E002323/1. In addition, the authors would like to thank John Webster (construction of experimental equipment), Philip Brindley and Mark East (model printing), Geoff Hodgson (photography), Robert Law (CAD modelling using Rhinoceros), Stephen Todd (salt bath modelling), and Michael Popper Associates LLP (original CAD geometry).

REFERENCES

Ansys Inc. 2010. Ansys-CFX and Ansys ICEM CFD manuals, Release 12.1 http://www.ansys.com

Castle Island 2010. 13/5/11-last update, *Castle Island's Worldwide Guide to Rapid Prototyping* [Homepage of Castle Island Co.], [Online]. Available: http://www.additive3d.com/ [2010, 16/5/11] .

Bower, D.J., Caulfield C.P., Fitzgerald S. and Woods, A.W. 2008. *Transient ventilation dynamics following a change in strength of a point source of heat.* J. Fluid Mech 614: 15-37.

Chen, Q. 2009. *Ventilation performance for buildings: A method overview and recent applications*. Building and Environment, 44 (4) pp848-858.

CIBSE, 2005. *Guide B: Heating, ventilating, air conditioning and refrigeration*. The Chartered Institution of Building Services Engineers, London, UK.

COST. 2007. *COST Action 732: Best practice guideline for the CFD simulation of flows in the urban environment*. Franke, J. et al.(Eds.) COST office. Hamburg, Germany.

Hunt, G.R. and Coffey C.J. 2010. *Emptying boxes – classifying transient natural ventilation flows*. Journal of Fluid Mechanics, 646, pp 137-168

Hunt, G.R. and Linden, P.F. 2001. *Steady-state flows in an enclosure ventilated by buoyancy forces assisted by wind*. Journal of Fluid Mechanics, 426, pp 355-386

Kaye, N.B. and Linden, P.F. 2004. *Coalescing axisymmetric turbulent plumes.* Journal of Fluid Mechanics, 502, pp 41-63

Kaye, N.B. and Hunt G.R. 2009. *An experimental study of large area source turbulent plumes*. International Journal of Heat and Fluid Flow 30(6): 1099-1105.

Kaye, N.B., Flynn, M.R., Cook, M.J. and Ji, Y. 2010. *The role of diffusion on the interface thickness in a ventilated filling box.* Journal of Fluid Mechanics, 652, pp 195-205

Linden, P. F., Lane-Serff, G.F. and Smeed, D.A. 1990. *Emptying filling boxes: the fluid mechanics of natural ventilation*. J. Fluid Mech 212: 309-335.

Linden, P.F. and Cooper, P. 1996. *Multiple sources of buoyancy in a naturally ventilated enclosure.* J. Fluid Mechanics, 311, pp 177-192

Wohlers, T. 2004, *Rapid Prototyping, Tooling & Manufacturing: State of the Industry*, Wohlers Associates, Colorado, USA.