Solar chimneys do draw but mainly they just suck

Colin Bidden Allison, AP.AIRAH Simultude, Elvin Chatergon, Fratelle Group

ABSTRACT

Solar chimneys are something of an enigma. They are not very common, and very little is known about their actual performance. They have been installed in various commercial or public buildings, and as long as they are not adding to the space cooling load, are assumed to work satisfactorily.

Very few studies have been done to rationalise their performance. Their use in a domestic application is quite scarce. Therefore, when it was proposed to use a solar chimney in a private residence in Perth, it was a fortuitous opportunity to estimate the actual, value – added ventilation contribution of the solar component.

A series of virtual experiments using a CFD analysis was designed to measure the actual solar chimney ventilation contribution. These experiments would be extremely impractical to perform in a real sense but the use of a capable CFD code proved to be an extremely useful design tool. The flip-side of this circumstance is that the CFD simulations could not be validated since they indicated that the solar chimney should not be constructed in the first place. At best, the expense of constructing a chimney that would have very little benefit was avoided.

Key words: Computational fluid dynamics (CFD), Solar chimney performance, natural ventilation

INTRODUCTION

Solar chimneys appear to be very clever devices that can be used to ventilate a building free of charge by using the energy of the sun. In construction they are preferably dark-coloured metal ducts, essentially located on the sunny exposure of a building.

Ideally they should run the entire height of the building, as tall as possible in order to capitalise on the stack effect. Some have a glass front and the remaining internal faces are blacked out.

Clearly this construction is expensive, and requires the remaining faces to be insulated from the interior of the building. Incident solar radiation heats the surfaces of a chimney. The heat is transferred to the air inside and then rises due to buoyancy convection.

If this convection process is potent enough, and the inlet to the chimney is suitably located, then air will be drawn from the building and vented to atmosphere at the chimney discharge.

As with a mechanical extract fan, it is essential to provide an allowance for make-up air to the building so that the convection process is not overwhelmed by flow restrictions. It can be a design challenge to integrate solar chimneys into the building architecture, at the same time ensuring that they perform physically.

Therefore it is reasonable to question what return on the effort can be expected. Unfortunately, it is not possible to generalise on solar chimney extract flow performance; a specific installation's performance requires individual assessment. It is demonstrated here how CFD simulations can be used to estimate a particular device's performance potential with a high level of confidence.

BACKGROUND

There appears to be a dearth of information pertaining to the performance of solar chimneys. Hence, it is difficult to compare installations on an equal basis and draw conclusions on the merits of a particular construction.

Some validation of CFD simulations and in particular the commercial CFD code FloVENT (henceforth "the CFD code in question") have been performed in the modelling of solar chimneys.

Nugroho et al (1) and Nugroho et al (2) performed several experiments on a solar chimney constructed from some PVC pipe.

They measured air flow and temperatures, and found excellent agreement with the CFD code in question's CFD model, in some cases obtaining exact matching with experimental data. They found that localised velocities within the chimney could reach as high as 0.5m/s.

Charvat (3) et al performed experimental and numerical simulations of several solar chimney configurations, which included vertical and inclined chimney surfaces.

They validated their numerical simulations performed with the Star CD CFD code, and obtained excellent agreement with experimental measurements. They concluded that the incident solar radiation had very little impact on chimney flow-rates, and ambient wind speeds had an overriding effect, in some cases negating airflow performance.

One can infer from these findings that firstly the use of CFD codes in predicting solar chimney performance is quite straightforward and has been proven to give good comparative results with measured experimental data. Secondly, it would appear from their results that the solar chimney performance is actually quite dismal. A maximum velocity of 0.5m/s as realised by Nugroho's experiments is insufficient to provide a meaningful ventilation rate in a practical application.

Clearly the air flow through a solar chimney is dependent on three variables. There is the ambient wind velocity, which causes an induction effect, sucking air from the top of a chimney.

There is the contribution of internal space loads, causing buoyancy convection, which tends to push air out of the chimney from the bottom. And then of course there is the buoyancy of air within the chimney due to the solar-heating effect.

The first two driving factors are not attributed to the solar effect, and depend on other highly variable parameters. Therefore they should be eliminated from the determination of the "solar" performance of the chimney. As far as it is known there are no studies available that have proceeded to isolate the performance of the solar chimney in this manner. Therefore this study has immense value in rationalising the performance of a solar chimney in the context of the Australian geographical conditions.

METHODOLOGY

The actual double-storey residence where the solar chimney was to be constructed was a marvellous piece of architecture.

However, to construct a 3D CFD model to replicate this building would have been extremely time-consuming, and as it turns out completely unnecessary. Instead a simplified 3D representation of the actual residence was created.

This consisted of a large room representing the ground floor, and above that a similar-sized room representing the first floor. The height of each room was consistent with the proposed architecture, since any stack affect would influence the buoyancy convection.

The first-floor facade wall was stepped in by 600mm to provide a shoulder for the chimney to sit on. The inlet to the chimney was thus located in the first-floor slab.

The vertical length of the chimney was equal to the first-floor height plus an additional 2m above the roof slab to represent similar geometric proportions to the actual proposed chimney.

In order to create a benchmark control case, a second opening or vent (without a chimney) was placed alongside the solar chimney inlet but separated by a distance of about 4m. Furthermore, a vertical opaque screen was located to shield this vent from any solar influences that may have affected the airflow through it. This control case provided a comparison between induced flow drawn through the chimney, and the airflow that would be naturally vented without the benefit of a chimney attached to it.

In the opposing façade wall of the structure to where the chimney was located and along its entire length, at ground level, an inlet opening was created. This permits the induction of cool ambient air.

In this case, it was assumed that the air was precooled by passing through a geothermal ground loop at 23°C. (The simulation of the geothermal ground loop was not performed as part of this study, but had been previously performed by others. Their results showed an expected inlet temperature to the house of 23°C).

Since this opening was large in comparison to the chimney inlet and control vent inlet, it is unlikely to present a significant air-flow resistance to either the chimney flow or the control vent flow.

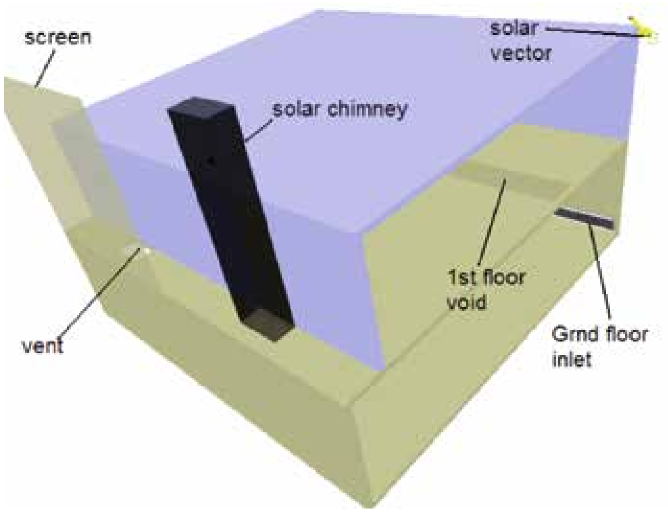

Figure 1: 3D geometry of the simplified house representation. Note that the vent in the first floor is not present in this scenario.

Fluid transfer between the two levels was facilitated by a long void about 1m wide in the first-floor slab. This simulated the presence of a staircase.

This void also provided an access route for buoyant air to ascend into the first floor rather than venting through either the solar chimney or the control vent. Finally, an additional vent was placed in the roof slab to permit the venting of hot air, which would be expected to accumulate under the roof unless it was allowed to vent somehow. The resulting geometry is shown in Figure 1.

The geometry was then meshed with a Cartesian grid. The mesh size varied from 150mm in the general bulk-fluid-flow area, to about 30mm inside the chimney and at the vent locations, to account for higher velocity and temperature gradients occurring there.

The solar radiation module, as well as the radiant heat-transfer module and view factor calculation was invoked. In addition some planar heat sources were located on the ground-level slab to simulate internal heat-loads occurring on the ground floor. These heat sources could be turned on or off depending on which modeling scenario was being considered. Potential heat loads for the first floor were ignored.

The CFD code in question was used to perform the simulations.

BOUNDARY CONDITIONS

Next the required boundary conditions were specified according to three different modelling scenarios, which are summarised in Table 1.

Table 1: Schedule of modelling scenarios

m/s	Heat load W/sq m	Solar time	Ambient temp °C	Inlet air velocity m/s
1	0	Feb 21, 1pm	23	0
2	120	"	23	0
3	120	"	38	0.5

Scenario 1 is a theoretical condition where no internal heat sources exist, and there is no resistance to the inlet flow. The ambient temp and hence transmission is minimal in order to emphasise the heating effect of the sun. Therefore flow through the chimney is due solely to the buoyancy forces derived from the solar radiation.

Scenario 2 has realistic internal space heat loads, which will facilitate a natural convection process in the room, independent of the chimney.

Scenario 3 is the most likely summertime scenario. The pre-cooled air from the geothermal ground loop is forced into the space mechanically with an initial velocity of 0.5m/s. In reality the inlet air would be distributed through several grilles and a fan would be required to draw air through the labyrinth and then discharge it into the room. This forced convection, and the buoyant air has to vent out of the chimney and the other two vents in a proportion based on the resulting building pressure distribution.

The solar azimuth and altitude were specified according to the geographical location of Perth and at a solar time of 1pm in February. The house was orientated so that the solar vector shone on the front of the house where the chimney is positioned. The thin opaque screen wall placed in front of the control opening was unlikely to influence any induced airflow through any of the vents, but was sufficiently large to shield the control vent from the sun so that it remains uninfluenced by any solar effects.

The house walls were specified as adiabatic (no heat transfer) so that no solar load was permitted through the various house facades. This ensured that only the heat transfer occurring in the solar chimney was responsible for any induced flow drawn through it.

The proposed solar chimney was 1200mm wide, 600mm deep and 5000mm high, constructed from 2mm thick mild steel, black in colour with an emissivity of 0.98.

A radiation view factor was specified for the solar chimney, according to a simplified radiation model consisting of elements no larger than 300mm square.

Since temperatures were expected to exceed 60°C within the chimney, a stratification turbulence component was included as an additional closure element in the modified k-e turbulence model. Figure 2 shows the predicted solar chimney surface temperatures as a result of the steady-state solar incident radiation. The maximum surface temperature of the metal duct reached about 50°C, which is consistent with what one would expect from a vertical steel sheet.

Boundary conditions summary

Chimney colour	–	Black
Emissivity	–	0.98
Chimney dimensions	–	1200mm by 600mm by 5000mm
Chimney construction	–	2mm thick mild steel
Inlet air temp	–	23°C
Internal heat load	–	120W/sq m

Finally, each steady state simulation was run to a convergence residual level of less than 5 per cent.

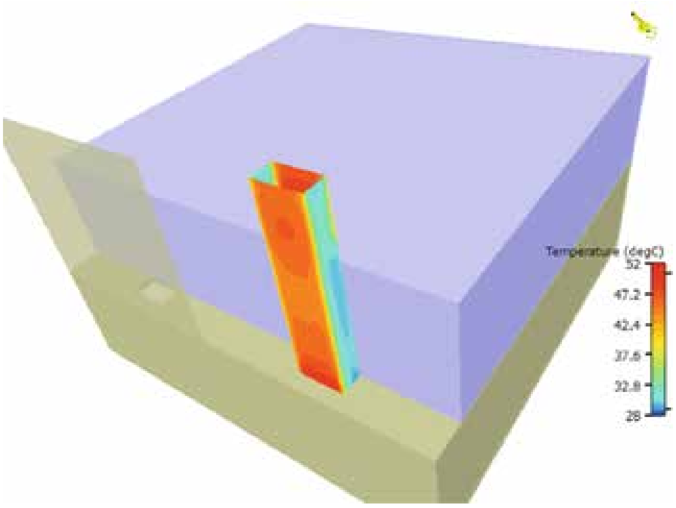

Figure 2: Surface temperatures of the Solar chimney due to solar radiation.

GRAPHICAL RESULTS

Only the graphical results for Scenario 1 are presented below by way of example, in order to demonstrate the type of comparisons that can be made. The rest of the results pertaining to flow through each vent has been tabulated in Table 1.

Figure 3: Velocity contours through vent.

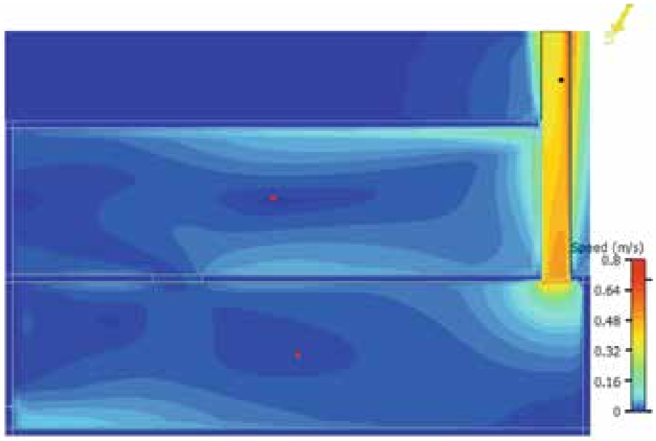

Figure 4: Velocity contours through solar chimney.

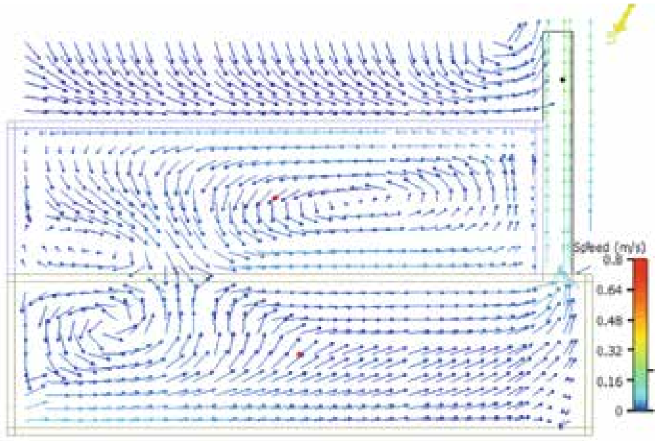

Figure 7: Velocity vectors through vent.

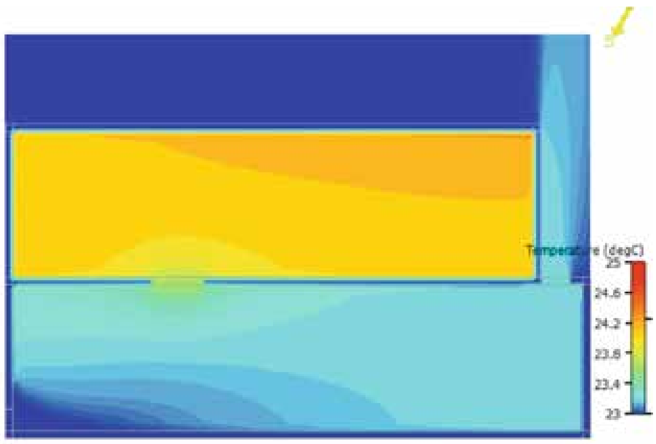

Figure 5: Temperature contours through vent.

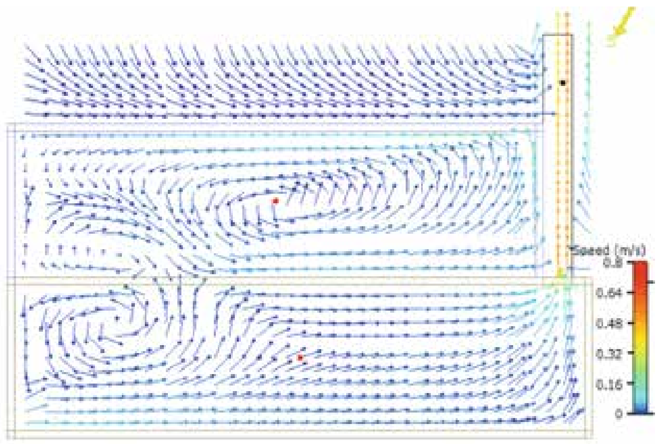

Figure 8: Velocity vectors through solar chimney.

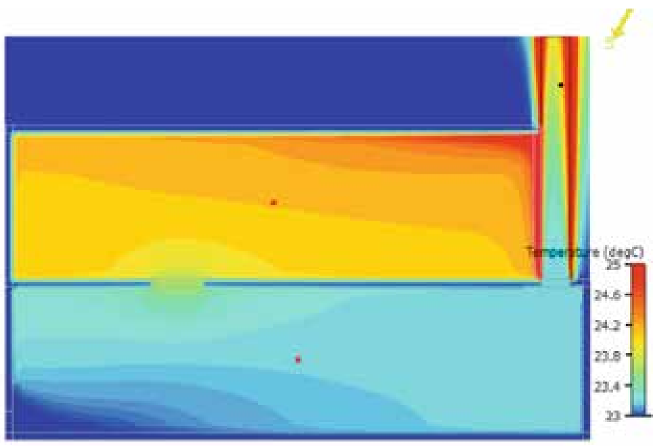

Figure 6: Temperature contours through solar chimney.

Table 2: Induced and vented flow rates in litres per second

Scenario	Chimney flow	Control vent flow	First-floor vent flow
1	314.9	0.087	–
2	957.2	505.3	932.5
3	939.8	988.7	1104.5

DISCUSSION

Induction of air due to solar effects alone(scenario 1) is approximately 315l/s, stemming from an average velocity of 0.4m/s, which is in agreement with the velocities recorded by Nugroho (1) (2).

This induced flow is only about 10 per cent of the ventilation quantity required to maintain internal temperatures below 27°C, were internal heat sources to be present. When realistic heat sources are introduced (scenario 2), then a total of 2400l/s is vented through the various openings (summation of row 2).

The solar chimney performs somewhat better than the open vent on the ground floor, venting almost twice that of the latter, but only for this case. One can infer that if you add the chimney solar contribution i.e. 315l/s to the vent flow of 505.3l/s, then you arrive at approximately the total chimney flow of around 957.2l/s. The actual proportions have changed due to the presence of the first-floor vent which wasn't present in Scenario 1.

The first-floor vent has a high flow rate due to the higher temperatures existing on the first floor. This is because the first floor is vented by air from the ground floor, which is initially heated by the ground-floor heat source, and therefore at an already-elevated temperature. Scenario 3 is probably the most likely to occur in reality. The cool inlet air would have to be forced into the space by a mechanical ventilation system. This is because the forces due to buoyancy will most likely be unable to overcome any duct pressure losses occurring in the labyrinth, or additional losses through ceiling vents or ducts connected to the chimney.

It is interesting to note that under this condition, all the vents have approximately the same flow rate. In fact the control vent actually has a higher flow rate then the chimney, presumably because of additional pressure drop through the length of the chimney.

This implies that there is minimal benefit if any in installing a solar chimney in this case, since the venting of air due to the other buoyancy and forced ventilation processes dominate. While a solar chimney does induce airflow, the space venting could just as easily be achieved by judiciously located high-level vents. The space temperatures on the ground floor are reasonable while those on the first floor are approaching uncomfortable. Therefore, in order to achieve reasonable conditions there, it is advisable to introduce some proportion of the labyrinth air directly to the first floor.

CONCLUSION

The CFD code in question predicted solar chimney temperatures that are consistent with expected practical results. Unfortunately, the induction of air through the chimney due to solar heating effects is quite low and these quantities are not very practical in this case. Under realistic operating conditions, all of the vents had similar flow rates. This implies that there is very little advantage of a solar chimney over a correspondingly sized open vent.

For the parameters simulated in this case one can conclude that that the expense involved in the elaborate construction of a solar chimney is questionable, and caution should be used to moderate enthusiasm in the implementation of these devices. However, that is not to say that solar chimneys might not work on a larger scale when applied to commercial buildings. A CFD simulation is the only practical method of evaluating whether they are effective, on a case-by-case basis. ▮

REFERENCES

1. Nugroho, Agung Murti, Mohd Hamdan bin Ahmad and Makmal Sain Bangunan. Possibility to use Solar Induced Ventilation Strategies in Tropical Conditions by Computational Fluid Dynamic Simulations. s.l. : University of Technology Malaysia, Department of Architecture.
2. Nugroho, Agung Murti. The Impact Of Solar Chimney Geometry For Stack Ventilation In Malaysia's Single Storey Terraced House. s.l. : Univerity Of Technology, Malaysia, 2007.
3. Charvat, P., et al., et al. Experimental and Numerical Investigations of the performance of Solar Chimneys. Brno University of Technology. Brno, Czech Republic : s.n.

About the authors

Colin Allison, AP.AIRAH (PhD) is the director of Simultude, thermal fluid modelling which provides CFD modelling services. Originally from Rhodesia, Colin has worked for various consulting engineering practices in South Africa and Australia. He is an Accredited Practitioner with AIRAH. Email: callison@simultude.com.au

Elvin Chatergon qualified in the UK as an architect, working on high-profile projects such as the Lower Lea Valley Masterplan in London for the 2012 Olympics. Elvin currently works for the Fratelle Group, and is involved with a number of commercial and residential developments in Australia, with an emphasis on environmental and ecological design. Email: Elvin@fratellegroup.com.au