WITH WHAT CONFIDENCE CAN AN ARCHITECT USE A CFD CODES PACKAGE AS A BUILDING ENVIRONMENTAL DESIGN TOOL? CASE STUDY OF THE SEARCH FOR AN INDONESIAN ROOF CHIMNEY.

Prasasto Satwiko
Jurusan Teknik Arsitektur, Fakultas Teknik, Universitas Atma Jaya Yogyakarta, Jl. Babarsari 44, Yogyakarta 55281, INDONESIA.
Michael Donn
CBPR, Victoria University, Wellington, NEW ZEALAND

ABSTRACT

This paper examines the role that computers might play in the education of architects about Environmental Science in buildings. It results from an analysis of the performance of a solar driven roof chimney in an indonesian house. It suggests that as computers become cheaper, so the range of analyses that designer will be able to do of the operation. of their building will be expanded.

The results of the comparisons between the natural ventilation performance predictions of the commercial CFD program and the reality measured in scale models are also encouraging.They suggest that the direction and relative velocity of the air flows through the building are well modelled by this type of softWare.

INTRODUCTION

Building construction in Indonesia is developing rapidly thanks to the improvement of the Indonesian economy. New buildings are built in new styles. The trend is to recall local architecture and adapt it to modern life styles. The trend can be criticised for the failure of some new designs to achieve thermal comfort with energy and environmental conservation.

It would appear that most Indonesian architects do not understand tropical architecture analytically and scientifically. This introduces difficulties when they try to modify local tropical architecture to new styles.

Curricula of architecture education and financial capability of most universities may be the source of much of the problem. Although addressing building technology, most universities only include a small amount of building science in their architecture curricula. Qualified experts are few. Moreover, expensive laboratories are not available. Fortunately, computer technology has grown rapidly so that powerful computers are available at low prices. Building science teaching can take advantage of this.

The Indonesian roof chimney research sought to develop a natural ventilation system for a hot humid climate. It simulated the system's performance with a commercial Computational Fluid Dynamics (CFD) codes package. This research is presented as a model of a new approach to education and potentially to practice of building science in Indonesia.

ARCHITECTURE AND BUILDING SCIENCES IN INDONESIA

As with other countries, architecture education in Indonesia emphasizes qualitative (philosophy, aesthetics, etc.) instead of on quantitative (analytical, numerical, etc.) subjects. Architectural design emphasizes aesthetics values over building science considerations. Most universities give Structures and Construction Technology classes for 8 semesters, plus some other classes in material technologies and building materials. These are building technology subjects. Taking our definition of building science as the physics of buildings, including acoustics, natural and artificial lighting, and natural and mechanical ventilation, we find these subjects comprise only a small part of the whole curriculum. Most universities present just the basic concepts of building physics and only a small amount in an analytical and scientific way.

Some lecturers argue that most architectural students are not ready for complicated mathematics (which is usually required to solve problems in building science). It is "well known" that most architecture students are reluctant to work with numbers. Computer development offers a solution in Indonesia for problems in building science education. Computers can process complicated numerical calculations and produce simplified results. Some institutions have started introducing software such as the DOE2 program. However, these programs are not popular. They are usually used in research and only by staff. Architecture students are not made familiar with them.

Architecture students use computers for graphics. They are familiar with drawing (AutoCAD, etc.) and animation (3DStudio, etc.) software. They usually develop their skill informally as amateurs or hobbyists and have personal computers. Most architecture schools, on the other hand, do not have sufficient capital to establish a computer laboratory. Simply speaking, most architecture schools are left behind by their students.

One might typify architecture students as follows

- they prefer graphics to numerical presentation

- they understand both real and imaginary spaces well

These two characteristics are also required to understand phenomena in building physics (such as acoustics and air flow) well. Introductory building science classes can take advantage of these characteristics. By using Graphic User Interface (GUI) software, the classes can be made more accessible and hence more interesting. Development of design tools like simulation programs needs to address more than just the the user interface. Teaching a simulation program like SUNCODE to groups of senior level architecture and building science students each year makes one a strong advocate for simpler GUI interfaces to such software. But, as SUNCODE also demonstrates well, merely improving the user-friendliness of the input is not sufficient. The raw output of many computer based environmental design programs is obscure. There is a need for translators of "bricks and mortar" into building physics, and interpreters and data analysers to provide benchmarks of the performance. What good is a kWh figure for the cooling energy use without some means of measuring whether it is good or bad?[1]

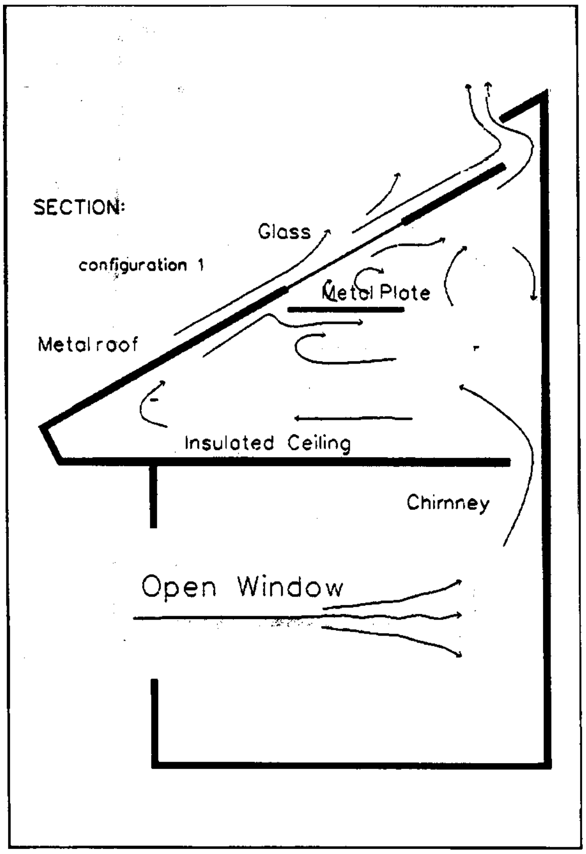

Figure 1 Section through roof chimney prototype.

The research reported here used commercial Computational Fluid Dynamics (CFD)^2 software. It used an older version of the program than is currently available. Although it had no GUI or other user friendly device, it was relatively simple to use. Its complicated calculation process made possible the study of the effect of variations in building design on air flows that would have been impossible with full-size models. It was responsive to changes of the design. This saving of time, cost and energy with graphic output and detailed numerical results is important if software like this is to be applied in education or practice.

If they are to be encouraged to design energy and environmentally responsive buildings, Architecture students (and professional architects) need to know air flow patterns inside and outside the building early in the design. They need immediate information of the effect of changes in the design on these flows, . CDF programs have the potential to fulfil their need for this type of information. It is clearly the type of information that cannot be conveyed by a few simple diagrams in a text book.

THE MODEL OF AN INDONESIAN ROOF CHIMNEY

A roof chimney (or a *stack* roof) is a roof which is designed in such a way that solar radiation can efficiently amplify the buoyancy of the air to induce indoor ventilation. The basic form of the roof chimney has long been known, and applied to vernacular architectures. In general, the induced air movement is not used directly to suck indoor air. Instead, it is used for ventilating the roof (such as in the double skin roof). A stack roof is usually designed in combination with a wind tower. In many types of ventilated roof, winds are considered to be more important than buoyancy. This is because wind induced ventilation flow is commonly stronger than stack induced flow, in particular, in low rise buildings. Stack induced ventilation (or buoyancy driven ventilation) is, so far, insufficient to create physiological cooling. Velocities associated with natural convection are relatively small, usually not more than 2 m/s.

The design of the Indonesian roof chimney was derived from consideration of improving the ventilation performance of the roof chimney (by maximizing the solar radiation potential) without ignoring technical and economic and social factors.

The basic form of the roof chimney prototype is as follows (see Figure 1):

- It follows the proportions of ordinary tropical houses with pitched roofs to shed rain water, and wide overhangs to shelter the openings.
- The chimney form is integrated into the roof

to avoid a radical architectural difference to buildings in Indonesia.

- The roof pitch is 30°. It is a simplified solution (or compromise) between conflicting requirements: too steep a roof will cause space inefficiency and higher radiation absorption on the glass; whilst, a flatter roof will introduce the risk of wind induced vacuum damaged, a shorter stack column and higher bending moment on the glass.

- Steel plates placed beneath the glazed opening in the roof have dimensions of 0.9 m x 9 m x 0.0006 m (width x length x thickness) each. The width of 0.9 m has been taken to avoid too wide glass (which might introduce a safety problem). The plates are nine metres long (the width of the house has been taken to maximize the area of horizontal steel plate). The thickness of 0.0006 m has been used to optimize the weight of the steel and its heat capacity.

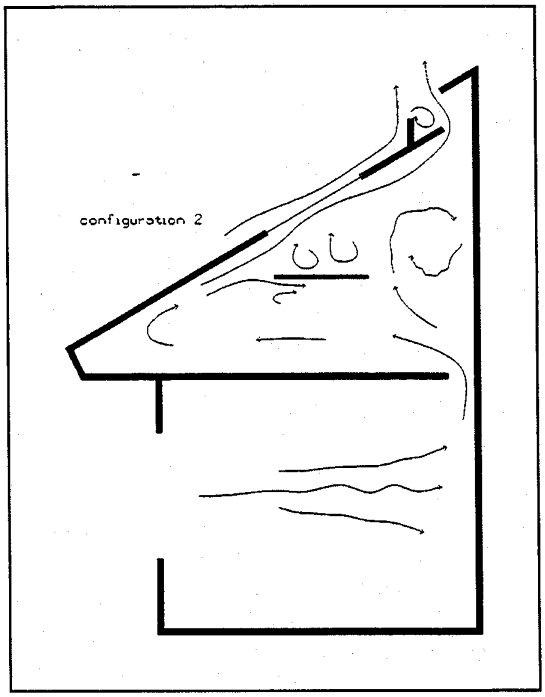

Figure 2 Prototype roof chimney: configuration 2

- The provision of an air space between the steel plate and the ceiling is to reduce the heating effect of the plates on the ceiling. It also gives a path for air to take the heat from the lower side of the plates. A close distance between plates and roof will minimize shading effect of the roof on the steel and reduce the glass area needed.

- The height of the upper and lower edges of the window openings are 2.0 and 0.80 metres respectively. This is the range of the occupants' level. Therefore, air moving from window opening to ceiling hole should sweep past the occupants.

- To maintain the flexibility of the room layout, the lower edge of the chimney shaft has been made two metres high. Thus, the shaft opening can be made as low as possible without necessarily dictating the furniture layout. The sixty centimetre width of the chimney is taken to allow a person to enter the attic (to clean the glass and steel plates).

- A symmetrical building form is applied because, at locations near the equator, the sun's position is on the northern side for six months and on the southern side of a building for six months.

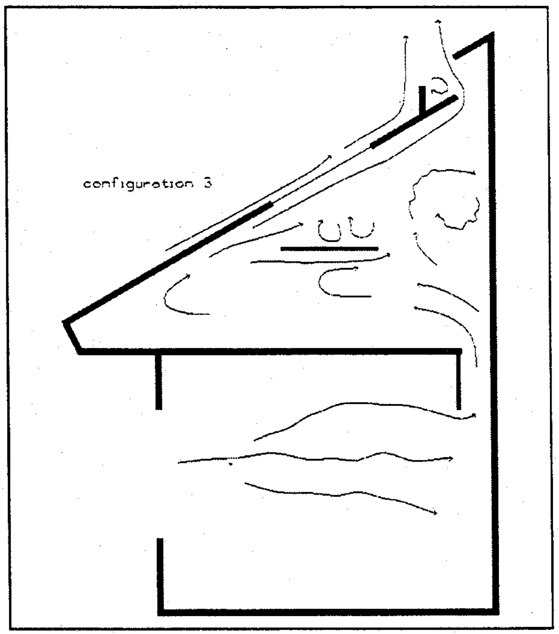

Figure 3 Prototype roof chimney: configuration 3

- The attic walls (between attic and outside) and the ceiling are made from insulated materials so that all the heat gained from the sun can be used to increase the attic air temperature but prevent the indoor air temperature from rising.

This basic form of the roof chimney involves convective and radiative heat transfer processes. Solar radiation received by the roof surfaces and the horizontal steel plate will be converted into heat. These warmed surfaces transfer the heat to the surrounding air through convective and radiative processes. The warm air rises (as it becomes lighter) and escapes through the upper opening in the roof chimney. It is replaced by air from the room which has a lower temperature. As a result, the outdoor air is induced to enter the room. As this outside air is normally warm, occupant cooling is expected to arise mostly from the increased movement of this air through the room.

Conductive heat transfer also occurs in this process, particularly in the ceiling between the room and the attic, and in the attic wall between the attic and the outside. By conduction the heat will flow from the hotter side to colder side of these surfaces. The colder side of the attic wall is the outside and it will release its heat there, whereas the colder side of the ceiling faces into the room below and it will radiate its heat there. Thus, radiation from the ceiling can warm the occupied zone if it is not well insulated.

EXPERIMENT USING PHYSICAL MODELS

The effect of scaling the air velocity was studied using physical and mathematical modelling. Experiments were conducted at first using three simple physical "models" of the roof chimney. The model scales were 1:4, 1:2 to 1:1. The mathematical relationship of the scales would suggest a scaling ratio for observed air speeds of 1.4142. The experimentally determined figure was 1.3. While there may have been a number of limitations on the modelling precision which affected this result, the agreement was considered sufficient to accept the model scale for scaling the results. The results indicated that the smaller scale (1:4) was accurate enough to be used to predict the air velocity in the real building. A full discussion of this whole experiment is published elsewhere.[3]

The findings of the experiments using physical models are as follows:

(1) A buoyancy driven air flow could be successfully induced by the roof chimney. Smoke released in front of the lower opening was sucked into the model. It proved that the stack effect which was created by the roof chimney could generate ventilation.

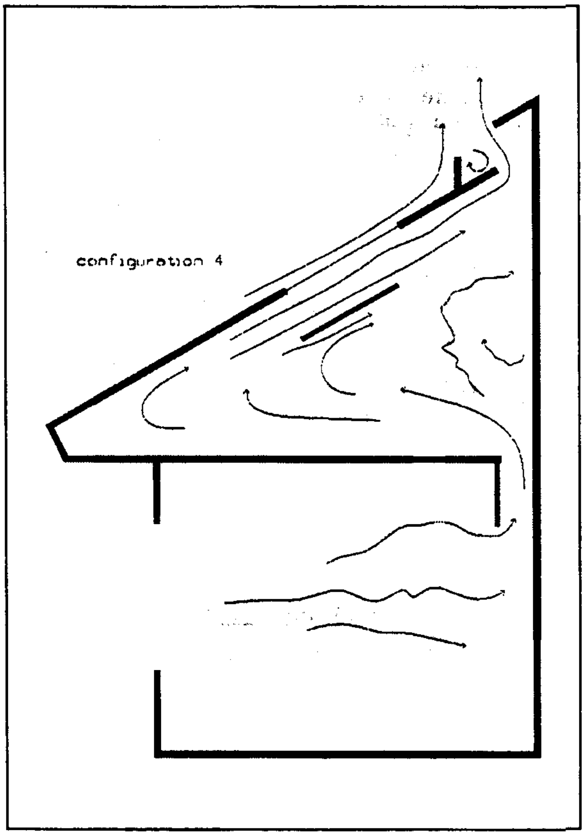

Figure 4 Prototype roof chimney: configuration 4

(2) Using a 1:2 scale model, air movement was induced at the occupant's level and could reach above the minimum velocity for physiological cooling (0.15 m/s). The smoke path indicated a horizontal flow of air from the lower opening toward the shaft (ceiling hole B) with a velocity fluctuating between 0.02 and 0.17 m/s (in Configuration No.3 - see Figure 3). Using a scaling factor of 1.4142 to predict the air speed in the full scale building, the figure came to 0.24 m/s which is sufficient to create physiological cooling.

(3) Configuration No.1 (Figure 1) had the worst ventilation performance of the four configurations. The air speed generated within the model room (up to 0.10 m/s) equalled 0.1412 m/s in the real building

Table 1. Range of speed in the model and in the real building.

Configuration No.	Range of speed in the model (m/s), measured in occupied zone	Range of speed in the real building (m/s)
1	0.00 - 0.10	0.000 - 0.141
2	0.01 - 0.16	0.014 - 0.226
3	0.02 - 0.17	0.028 - 0.240
4	0.03 - 0.15	0.042 - 0.212

which is still outside the comfort zone.

(4) The horizontal steel plate contributed to the attic air temperature increase. Configuration No.3 (Figure 3) showed that the higher temperature of the horizontal steel plate had made the air within the attic move more actively and develop a higher velocity.

(5) Air velocity development was affected by obstruction. Rafters created obstruction which caused poor air velocity development under the roof.

(6) The main air flow patterns within the models were almost the same for all four configurations.

(7) The effect on the air flow pattern of the depth of the shaft protruding into the room was small.

EXPERIMENT USING A CFD COMPUTER PROGRAM

A commercial CFD computer package for IBM PC computers was used to perform this analysis. The model study process used the procedure recommended in the manual for the program. This procedure was then systematised in order to ensure that the experiment could run efficiently. The aim was to minimise the potential for error arising from the difficulty of checking the input data, which was a mesh of numerical values overlaid on the irregular shape of the roof chimney.

The cells of the grid were arrayed on what the manual described as a Body Fitted Coordinate array, where the mesh cell size varies according to the size and significance of the flow domain. Temperature and velocity boundary conditions were applied to all surfaces of the model. The velocities at each of the openings in the model were calculated as part of the analysis. The model assumed laminar flow.

The CFD analysis reached the following conclusions:

(1) Buoyancy driven ventilation can be generated within the model. All four configurations showed the presence of ventilation flows. On the other hand, a test which used Configuration No. 3 with all of its surface temperatures the same (in this case zero) showed zero air movement. This confirmed that the simulation did model this aspect of reality accurately: without a temperature difference no air movement would exist.

(2) The air flowed through the occupant zone at a low speed. The highest range occurred for Configuration No.2 (see Table 2 and Figure 2.) It was from 0.0238 to 0.071 m/s at full scale. This is below the required air speed for physiological cooling.

(3) The intrusion of the chimney shaft below the ceiling into the room reduced the air velocity.

(4) The shaft constructions in Configurations No. 3 (Figure 3) and No.4 (Figure 4) give only a small advantage to the area of the occupant zone. It can be seen from the speed contour plots of Configurations No. 3 and No.4 that the presence of the shaft does constrain the air to flow horizontally for a little bit longer compared to Configurations No. 1 and No.2. before entering the attic through the lower hole of the shaft. However, when the speed contour plot of configuration 1 (or 2) was superimposed on the speed contour plot of configuration 3 (or 4) it was obvious that the difference was very small. This was because in configurations 1 and 2 with no shaft, the air also flowed horizontally. It started deflecting upward toward the ceiling hole only after it had travelled more than half-the width of the room.

Table 2. Range of air speed in the computer simulation and in the real building.

Configuration No.	Range of speed in the model (m/s)	Range of speed in the real building (m/s)
1	0.0216 - 0.0648	0.031 - 0.092
2	0.0238 - 0.0710	0.034 - 0.100
3	0.0192 - 0.0576	0.027 - 0.081
4	0.0177 - 0.0532	-0.05

(5) The main air flow patterns within the models were almost the same for all four configurations.

(6) The effect of the shaft on the air flow pattern was minor. The shaft does keep the air flow in the occupant zone. However, it reduces the air velocity.

(7) The CFD package produced very clear and detailed air temperature gradients, air velocity distributions and air flow pattern. The effect of changes (represented by Configurations No.1. to No.4.) could be seen clearly.

COMPARISON BETWEEN THE RESULTS OF PHYSICAL AND COMPUTER EXPERIMENTS

There was a consistent relationship between the physical and computer experimental results.The changes in the results due to the changes in the different configurations were consistent in both the physical and mathematical models. The physical experiments gave a clear idea of the phenomena occurring within the models (i.e., the fluctuating velocities, turbulence, and minor changes of air movement direction), which could not be seen in the computer experiment results.

In terms of the general air temperature gradient, air velocity distribution and air flow pattern, both the physical and computer experiments gave consistent results. It was presumed that therefore one could treat these as reliable predictors of full-scale performance. The different magnitudes of air temperature and air velocity produced by the computer experiment were expected since the data inputs were limited by the input assumptions (laminar flow, two dimensional, steady state, etc.), which differ from the reality observed in the physical experiment.

Both the physical and the computer experiments demonstrated that the roof chimney system successfully induced ventilation air movement although they gave different magnitudes of air temperature and velocity. Both showed the ability to be used as design tools. The computer experiments demonstrated that the CFD package could simulate the effect of changes to the air temperature gradient, air velocity, and air flow pattern with reasonable validity (when compared to the physical model). The ease with which the graphically represented results could be understood, and their reliability, led to the conclusion that CFD packages could be used as design tools yielding qualitative guidance as to air flows. This would be especially useful in the preliminary step of building environmental design where general ideas are usually sufficient.

The physical experiment found the highest air velocities (at the occupant zone) occurred in Configuration No.3. On the other hand, the computer experiment found the highest air velocity (at the occupant zone) in Configuration No.2. There is therefore, still some work to be done to ensure that the CFD results can be interpreted reliably as quantitative predictions of real air flows.

CONCLUSION

CFD based computer simulation has the potential to become an important building climate design tool. It is fairly easy to use once the hurdle of understanding the input and output data jargon is overcome. Any effects caused by changes in building details can be seen in a relatively short time. It is sufficiently reliable to be used in early design stage for qualitative study of the likely direction and pattern of naturally induced air flows.

A background in architecture study is useful for CFD simulation, as it works with architectural concepts in its building description. This knowledge is useful from the preparation stage where one is transforming the real model into a computer model, through to the post-processing stage where one is interpreting the graphic results. However, architecture study neither gives knowledge in fluid dynamics nor advanced mathematics. This made entering the control data for the simulation difficult. A convincing decision on data entry is difficult without help from experts.

The greatest difficulty with the particular CFD package used, arose from the lack of explicit examples in the manuals of the use of CFD for building climatic design .

There is no doubt the computer program can simulate the real phenomena. There are high similarities between results of computer experiments and of physical experiments. However, to use the numerical results convincingly as a final design guide, expert help is still needed. The question that remains to be answered is whether it is possible to package that "expertise" in the simulation program itself. In the authors' experience, the input process would be improved if one described the building in physical, architectural terminology (steel and thatch etc) not thermal elements in a calculation procedure. The graphical presentation of the results can be understood and used by most experienced and thoughtful architectural designers.

ACKNOWLEDGMENT

The authors would like to thank to the institutions and persons mentioned below for their support:

Dr. George Baird, School of Architecture, Victoria University of Wellington, Wellington, NEW ZEALAND

Badan Perencanaan Pembangunan Nasional (BAPPENAS), Jakarta, INDONESIA

Victoria University of Wellington, Wellington, NEW ZEALAND

Works Central Laboratories of Wellington, Wellington, NEW ZEALAND

Universitas Atma Jaya Yogyakarta, Yogyakarta, INDONESIA

REFERENCES

1 **Integrating Design Tools: Model Flexibility for the Building Profession**, (Amor, R., Hosking, J., Groves, L. and *Donn* M.) Symposium on Building Systems Automation - Integration, A/E/C Systems 92, Dallas, Texas, USA, 10-12 June, (1992)

2 **FIDAP Introduction**, Revision 5.0; Fluid Dynamics International, Illinois, 1990

3 **The development and modelling of a natural ventilation system for Indonesian conditions - case study of a roof chimney;** Prasasto Satwiko, MBSc Thesis, Victoria University of Wellington, 1994