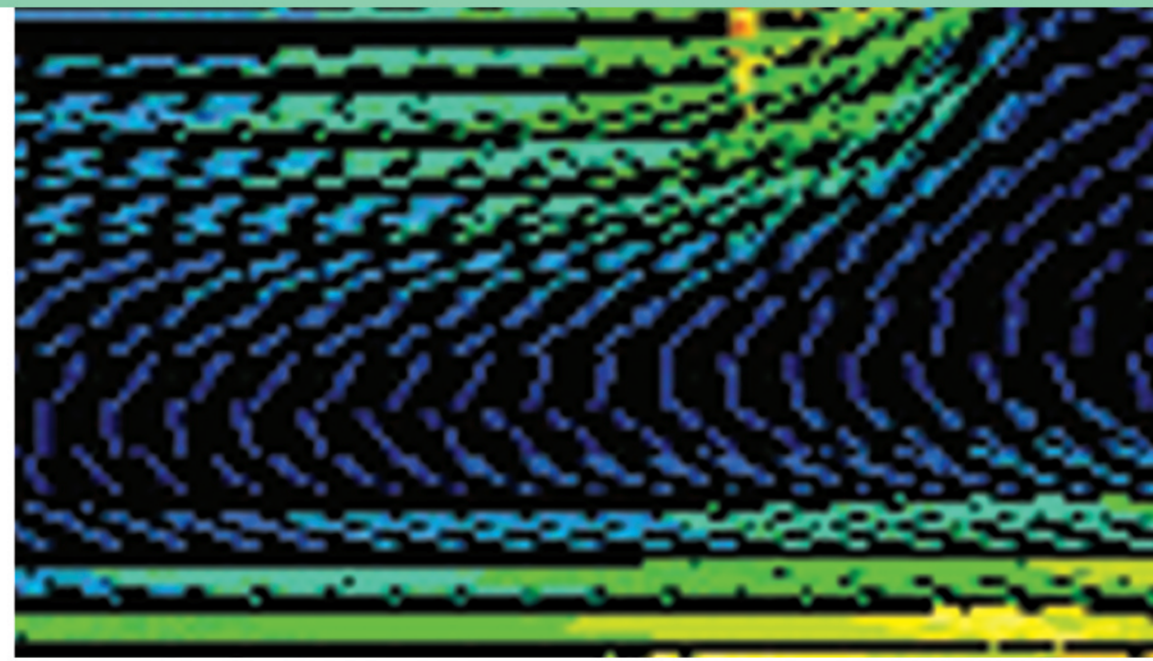
Architectural Engineering: Towards Practical Integration | Jelena Srebric, Ph.D.

Architectural Engineering:

Towards Practical Integration

Jelena Srebric, Ph.D.

Visiting Professor, The Pennsylvania State University



Public Lecture in
'Workshop on Airflow Simulation in Architecture
Using Computational Fluid Dynamics'

Faculty of Architecture, Thammasat University

Council of Deans of Architecture Schools of Thailand

Cooperative Research Network, Ministry of Education

Architectural Engineering:

Towards Practical Integration

Jelena Srebric, Ph.D.

Visiting Professor, The Pensylvania State University

Public Lecture in
'Workshop on Airflow Simulation in Architecture Using Computational Fluid Dynamics'

Faculty of Architecture, Thammasat University

Council of Deans of Architecture Schools of Thailand

Cooperative Research Network, Ministry of Education

Chalermwat Tantasavasdi and Non Arkaraprasertkul, Editors

Welcome Address by the Dean

Dear our expert, scholars, colleagues, and students:

It is a great pleasure for us today to have Professor Dr. Jerena Srebric here at Thammasat University. I would like to thank for her effort and invaluable time in giving us a special lecture and workshops on airflow simulation: CFD. I wish that this particularly event will lead to some academic collaboration in the near future.

Before we start the activity, I would like to explain the relationship between this expert invitation project and the 4+2 architecture program at Thammasat University. In most of developed countries such as the USA and Japan, the 4+2 program represents the trend of today's and future architectural education. In fact, our special knowledge-based and research-based programs are running in parallel to the world's architectural education systems. Our endeavor to have experts in architectural engineering field such as Dr. Srebric promoting new technology to us is therefore a part of our objectives for the knowledge-based education.

I wish that our colleagues from other institutes would try to proceed in the same direction as we do. Holistically, this would raise the overall standard of our architectural education and practice as a whole for the nation.

Finally, as the host of this 'Expert Invitation' project, we would like to express our appreciation to the Thai government in providing funding for the project through the Cooperative Research Network—CRN of the Higher Education Commission, Ministry of Education.

Thank you for your attention.

Professor Vimolsiddhi Horayangkura, Ph. D.
Dean, Faculty of Architecture
Thammasat University

Project Background

Natural Ventilation is regarded as one of the most effective passive cooling strategies for buildings in hot-humid areas such as Thailand. Architects can imagine the airflow patterns within and around buildings but lack the tools to prove their assumptions. The standard architectural methods being used nowadays for building airflow and ventilation design have a great deal of technical limitations. Computational Fluid Dynamics (CFD) is internationally considered the most powerful tool for building airflow simulations. It has been widely used not only in aerospace and mechanical engineering fields, but also in architectural research and practice. Despite its capability, CFD is a relatively new technology in Thailand, especially in architectural area.

Sponsored by the Thai government through the network of Energy and Environmental Conservation, Professor Jelena Srebric, a renowned expert with copious invaluable research and publications in the field of airflow simulation, is requested to lead a workshop on CFD in Thailand. Participants include Thai educators, researchers, professionals, and students in architecture. The activities involve public lectures and a training workshop on CFD program, both constructed for general attendants with architecture background.

This booklet illustrates the contents of the introductory yet significant public lecture, entitled ‘Architectural Engineering: Towards Practical Integration.’ The lecture focuses on the collaboration among building designers, especially between architects and engineers. Leading to new education systems and practices in the area of architectural engineering, such partnership is also proved to work effectively and therefore represents the future direction of building industry.

03

Research in Building Thermal Science

The motivation for research in building thermal science in the United States is that buildings use 1/3 of total energy and 2/3 of electricity. It is a huge number as the United States spends $200 billion per year for buildings' cooling, lighting and operation. The Department of Energy estimates that 30-70 percent of this money can be saved. So they are heavily investing in this kind of research, because they want this number to go down. There is another very important point: people especially in the States, spend 90 percent of their time indoors. Very few activities do happen outdoors. You spend a lot of time in buildings, so it had better be a pleasant environment, visually and thermally.

Another important thing is that about 6-10 percent of productivity loss was measured or estimated due to poor Indoor Air Quality (IAQ). It is another big topic these days in the United States. My research interests are really in the development of CFD as a design tool. There are a lot of obstacles and many good reasons for this technology not yet to be there as a comfortable design tool. It is still under development. This technology traditionally came from different industries. It came from the military industry; it also came from aerospace engineering. There are huge differences between a spaceship or wings of an airplane, and a building. So we do not need a completely new model in order to be able to comfortably apply this technology to building design.

I do simulation and measurements of indoor and outdoor environments. Now, I actually do not focus my work any more on indoor. I work most of the time right now on the simulation of outdoor environments. I work on ventilation and indoor air quality and I also do building energy analysis.

I have here a list of the most recent projects that I have been working on. I have a project on designing healthy and energy efficient buildings using computation fluid dynamics. This project is about coupling of CFD and thermal simulation. I have heard that some of the researchers here do work on similar projects. I work on an indoor environmental design tool for entire buildings. One of the big problems of CFD is that it is very slow. It is so slow. How many of you have done CFD simulation? Nobody? Is it slow when you do the simulation? It is not only slow in terms of calculation, but also slow in making input. How many days do you sit in front of a computer trying to make things work for you? So, there are many things about CFD that are against you and need improvements. I work on moisture control. This project has just ended this month. I have seen a lot of buildings being damaged by moisture in Thailand and I just cannot stop thinking that there is something that should be addressed. I am sure that there are a lot of people working in that area. I personally have interest in that area, so if anybody would like to discuss about the moisture control, please see me after the lecture.

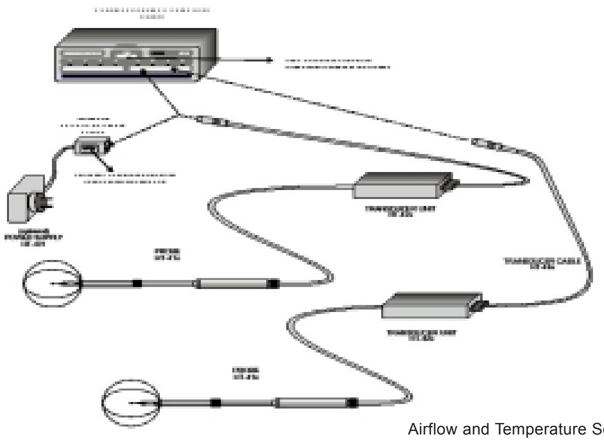
I also work on sustainable technology but from a different perspective. I educate students because if I go outside and try to work on a sustainable technology, it is much less powerful than if I teach hundreds of students and they go to their workplaces and do it. So, that is the strategy.

I am the director of an environmental chamber. Actually we are going to switch the name to 'building environment research laboratory' because it is getting too big. It is not only an environmental chamber itself. I have two career awards from National Science Foundation (NSF) and National Institute of Occupational Safety and Health (NIOSH). So, I do work with medical doctors trying to make buildings better for occupants. I have six graduate students: four Ph.D. students and two master's students. I have two undergraduate research assistants working in the environmental lab facility that I am managing. I am developing something called an 'HVAC playground.' I try to interest my students in HVAC systems. If you want to see more details, you can always go to my webpage. I try to update it regularly and that is the best way to communicate to the rest of the world since if I reply on one-by-one basis, it would take me too much time. I also collaborate with aerospace engineering, horticulture, landscape architecture, civil engineering and education. In the past four years, I managed to establish the collaboration within Penn State University.

4

This is an environmental chamber facility (Figure 4). It is a fairly big metal box that is nicely insulated from the rest of the building, so that we can study inside what is going on with different systems. This is a rack with an acquisition system and some old computers that we keep there in the laboratory.

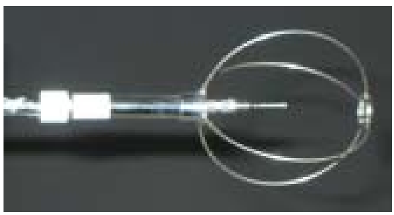
We have over 150 different sensors (Figure 5). I cannot bring information about everything because you might probably fall asleep. We have something called a 'tracer gas monitor.' It uses the gas that occurs naturally so you can follow exactly how the contaminants spread throughout the building if you want to study something like that. We have airflow and temperature sensors measuring low air velocity—a very difficult job. Just one of these instruments—I have around 20-30, probably 25—costs around $3,500. I bought them from Eastern Europe because they offered a better price. So, to have the lab that I am showing you right now, the value that we estimate approaches half a million dollars—very expensive facilities.



Airflow and Temperature Sensors

Tracer gas monitor

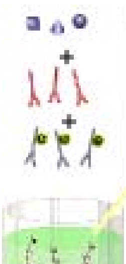
Total over 150 Sensors

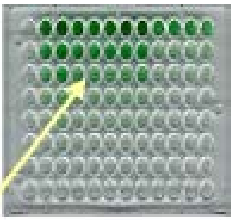


5 a

Optical Particle Counter (0.33 – 2 μm+)

Environmental Chamber - T (t), %RH (t), CFM





ELISA – Immuno Assays

Positive ELISA Test

Luzchem UV Photo-reactor
Defined UV Irradiance Conditions

5 b

Besides airflow and temperature measurement, we have instruments for measuring particle concentration. Because there are a lot of little things flying around us, we need to know how many and what the sources are. Another important thing that we do is study what happens with Volatile Organic Compounds (VOC). You know, the smell of the new things or when you enter your car on a raining day; some people even like that smell. The smell is coming out of plastic, which actually could be harmful. If you get long-term exposure to high concentrations, your health could be damaged.

We have lighting lab space (Figure 6). Our program is the only one that has a Ph.D. degree with two advisors in lighting system. We have this intelligent lighting control system, and this is a demonstration/research facility. Now, it looks even much fancier than the photo that I have here. This facility, the same as my HVAC playground, was built with donations from industries and manufacturers. It is actually our students putting components together. So, if we design something and our students are capable of putting its components together, it means we are doing a good job in educating them.

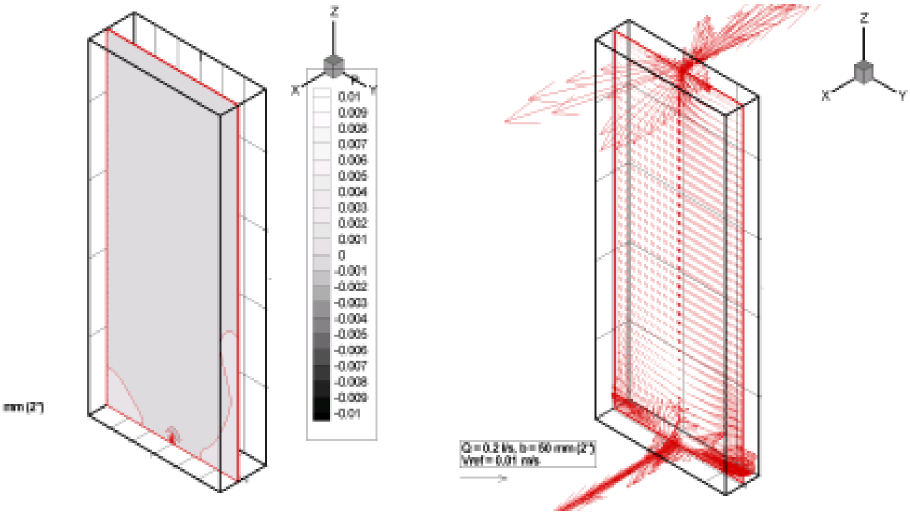
A little bit about building envelope research—I am just bringing a few topics here before we go back to CFD shortly. Topics are wind/seismic/impact resistant glass curtain wall, building enclosure performance-wall and roof systems: membranes and earthquake problems. Because one problem with the structural system is that the design of sub-structure engineering exists there for a period of time, a lot of things are known.

6

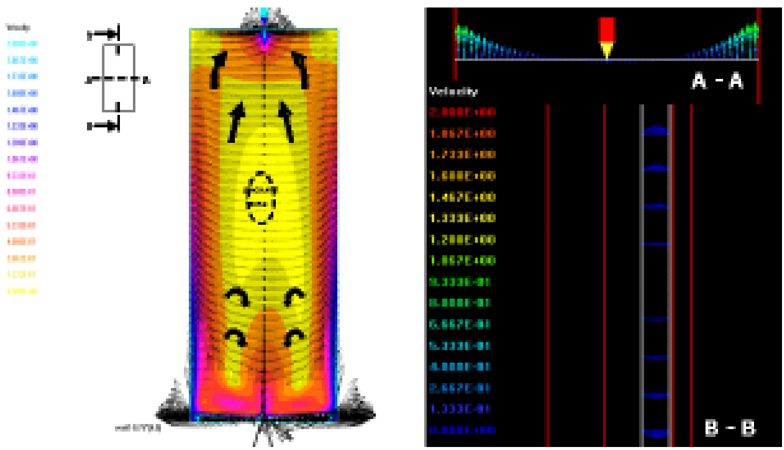
7

This is one thing that I can show you that I have been working on collaborating with structural engineering, partly in the architectural engineering. This is the test facility (Figure 7), built based on existing houses in the United States. You have air gaps, a screen wall and air gap behind a screen wall, and then the rest of the structure of the building. It is a very typical design. There has always been a question of how to ventilate and remove moisture from that little section. We have here the wet surface, which is always completely wet. We run air through the air gap and see how much moisture is removed. The front surface is Plexiglas, so it is completely enclosed. You have enclosed surfaces at the front and the back representing completely moist wet surfaces. We now look how this is going to be ventilated. Here I have the pressure distribution (Figure 8) because the pressure is the driving force for the ventilation and I have arrows here showing you computational fluids dynamic result.

Now we look at the different cases; we have over a hundred cases simulated. We look at the airflow for higher and lower flow rates (Figure 9 and 10), on the different surface, in order to be able to better design the system. We have the air coming in here, the way that is coming through your building envelope. We notice that for the high flow rate (Figure 9), actually there is a huge stagnant area. You see this yellow shaded area; that means very slow air velocity, at almost no air


Z
X
Y
0.01
0.009
0.008
0.007
0.006
0.005
0.004
0.003
0.002
0.001
0
-0.001
-0.002
-0.003
-0.004
-0.005
-0.006
-0.007
-0.008
-0.009
-0.01
mm (2")
Q = 0.2 l/s, b = 50 mm (2")
Vref = 0.01 m/s

8


A - A
Velocity
2.000E+00
1.867E+00
1.733E+00
1.600E+00
1.467E+00
1.333E+00
1.200E+00
1.067E+00
9.333E-01
8.000E-01
6.667E-01
5.333E-01
4.000E-01
2.667E-01
1.333E-01
0.000E+00
B - B

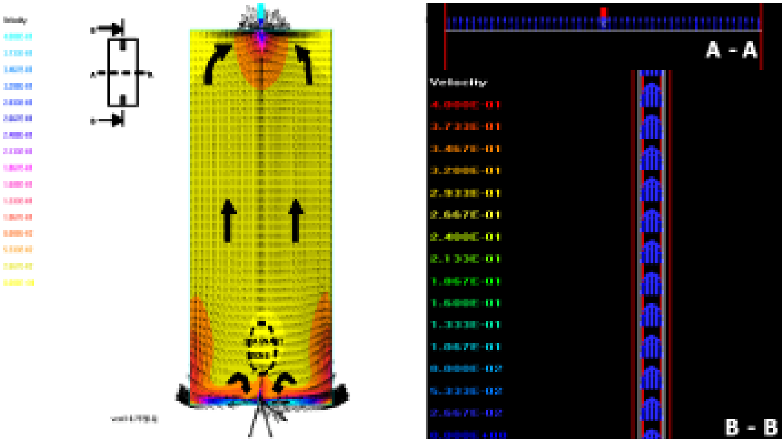
9

movement. One big surprise that we discovered is that actually at the lower flow rate (Figure 10), we have a better drying effect in that wall because there is no stagnant area in this wall cavity.

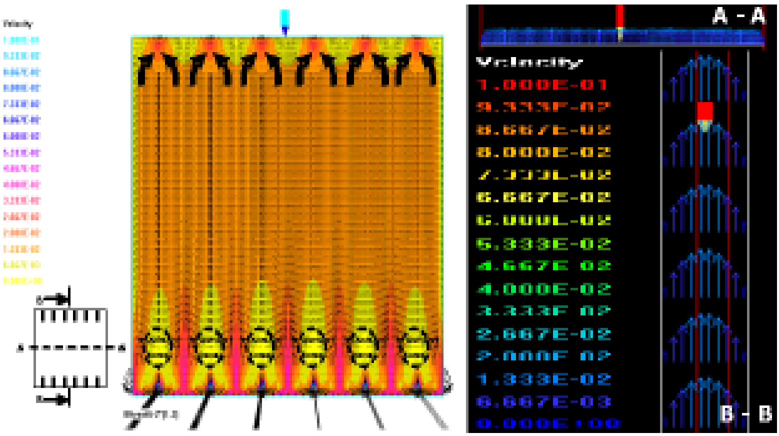
Then we came out with this multiple opening (Figure 11) because we really want the system to work for both high and low flow rates. This is one of the examples of mechanical engineers and architects, in architectural engineering, collaborating. I did not know anything about the wall system but now I know because I learn through the project. I would never have imagined this to be a problem at all if my colleague from the structural side did not approach me to work with him.

The image here, with the arrows, shows the pattern of wind through that air gap that I was discussing. This color here: red, is approximately one meter per second. This 0.1 meter per second is the velocity in that air gap. It is a very slow wind because we look at that the worst scenario case when you have very little wind, you hope that the enclosure would be dry and the moisture would be extracted from that enclosure. That is why we design multiple openings. We can get better airflow through that gap so that our wall can be dry even with very low wind. That was the whole idea. We also have the concentration of the moisture and other very detailed simulations. The only important thing that comes out of this is not to present the result of what we have done at the meantime, but the final design that makes the wall easier to dry. It is the whole purpose of this CFD simulation.

This is not to tell you that I can demonstrate very fancy animation. It is not that I have done hundreds of simulations, it is how we did improve design of the buildings. That is the purpose. CFD is just a tool, so you should not be scared of using it. You can learn how to use it and the more time that you spend with it, the better you will be. When I use CFD and present my work, there are always people from aerospace engineering or from physics just heavily criticizing my work being this or that. I do not bother, so I suggest you to do the same thing.


A - A
Velocity
4.000E-01
3.733E-01
3.467E-01
3.200E-01
2.933E-01
2.667E-01
2.400E-01
2.133E-01
1.867E-01
1.600E-01
1.333E-01
1.067E-01
8.000E-02
5.333E-02
2.667E-02
B - B

10


A - A
Velocity
1.000E-01
9.333E-02
8.667E-02
8.000E-02
7.333E-02
6.667E-02
6.000E-02
5.333E-02
4.667E-02
4.000E-02
3.333E-02
2.667E-02
2.000E-02
1.333E-02
6.667E-03
0.000E+00
B - B

11

chemical and biological agents around large urban areas. We wanted to see if we can distribute smart sensors around the campus that would pick up concentration levels and send information to CFD about the wind and current stage of the contaminant concentration, then do a very quick CFD simulation and tell the campus population how to evacuate. We can avoid high concentration areas by this smart technology. It is under development. We still do not have it but I can show you a little bit of what we have done so far. This is actually the project that I sit with three physicists and we do not understand each other.

For CFD, it is so important to validate. If you show me the result, I do not trust you. If I show you my result, you should not trust me. It is very important to do benchmark validation and measurements for some cases, so you can show me with confidence that your program is good and you know how to use it. That is called 'benchmark validation.' That is why I have the laboratory because first I want to test one case and then do other hundred simulations that are their variations. Basically, you need to have some kind of validation. You see the little models sitting

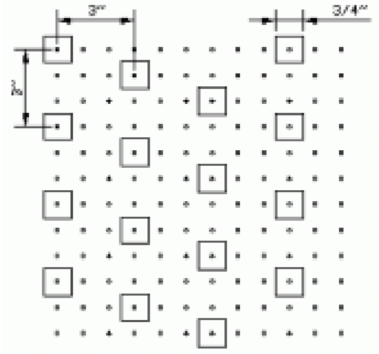
22

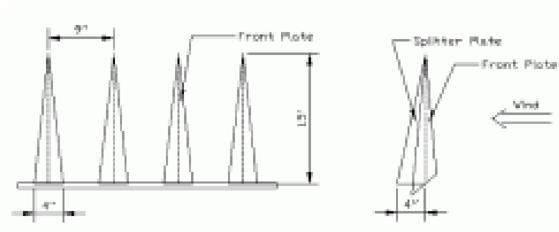
here just about to go to wind tunnel (Figure 23). This is a test facility in aerospace engineering. They let me test our buildings within this test section here. Basically, you have a huge fan that runs the air around. I put the little model here and I pass the velocity around the building.

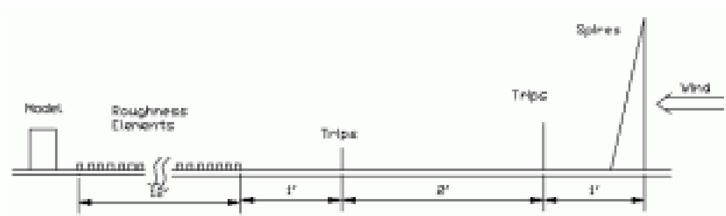
Here is something called ABL: Atmospheric Boundary Layer (Figure 24). We have to introduce this thing inside the wind tunnel. You see the piles and little bricks here; we have to build those ourselves because aerospace engineers do not need Atmospheric Boundary Layer, since they do not work with layers, and they work with airfoils and spaceships. So, we have learned by ourselves and did a lot of adjustments in the wind tunnel.

23

24


3"
3/4"


Front Plate
Splitter Plate
Front Plate
Wind

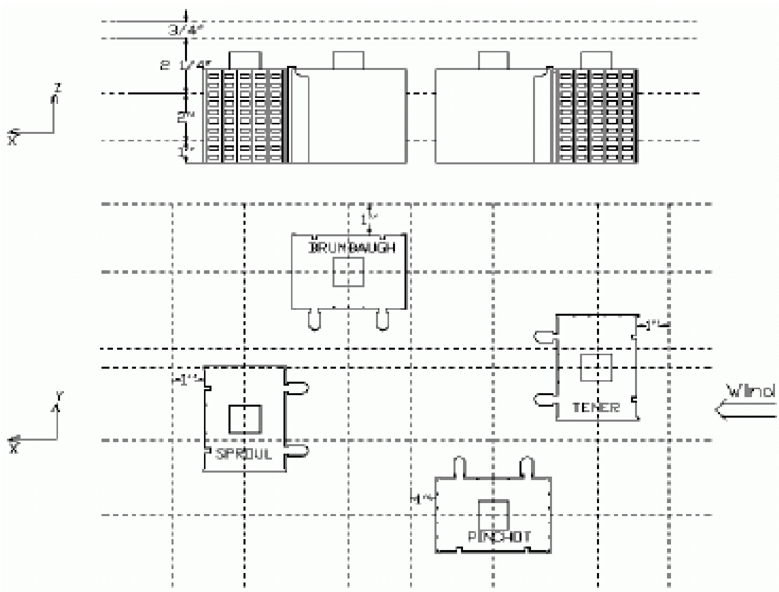

Model
Roughness Elements
Trips
Trips
Spires
Wind
1'
2'
1'

Here are the four buildings (Figure 25). I am showing you different cross sections where we will do the measurements. My students have conducted a huge number of measurements which I do not have data with me right now.

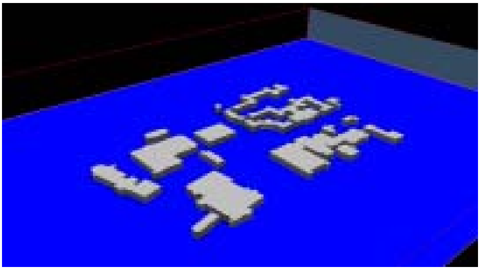
Here is the campus simulation (Figure 26). The four buildings are situated right here. We are still building that simulation and it is not done yet. I am just showing you a very new result. Here is the source of pollution and you can see how it spreads throughout the campus as we have wind blowing in that direction. The red and especially the black are the high concentration areas of contaminant spreading throughout the campus. You can see that most of these buildings are affected, but people will be safe in other areas. The blue area is unaffected by contaminant. And you see the red represents very high concentration that will go back to blue again. In fact, the wind is blowing and the concentration goes down with the time.

From the different cross-sections, the vertical cross-section (Figure 27), you see how contaminant goes around the building. This blank area is a very high concentration area. The program does not represent this with any colors; it is just too high in numbers. And you can see how the entire buildings are actually contaminated.

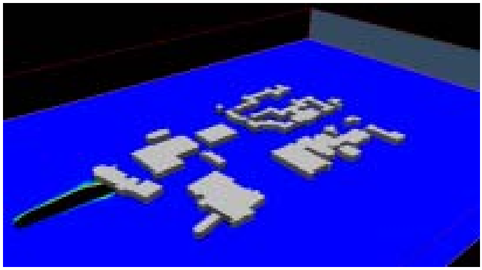
We have airflow velocity and temperature simulation, but I am showing you just the concentration result and how the contaminant disappears with time or actually being diluted with wind. The simulation of the space like this one would take at least 3-4 hours on a very fast computer. On a little bit slower computer, it can take an entire day. Or if we talk about the simulation of an entire campus, we are talking weeks. So, if I am going to make the evacuation efficient, my simulation has to go down to few minutes. This is what I am working on right now. I am trying to make the CFD simulation work accurately within a few minutes.



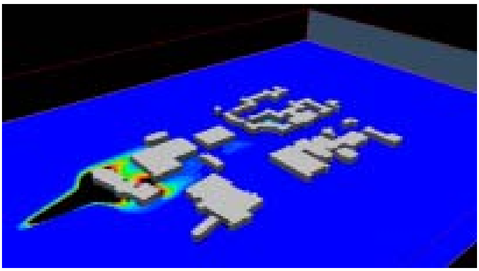
25



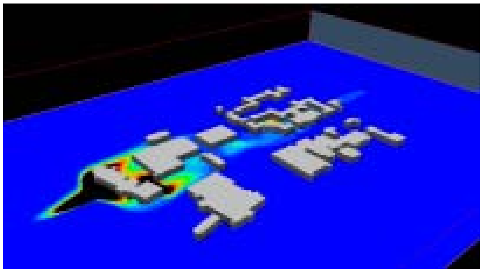
(a)



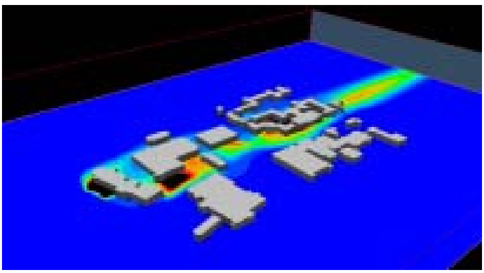
(b)



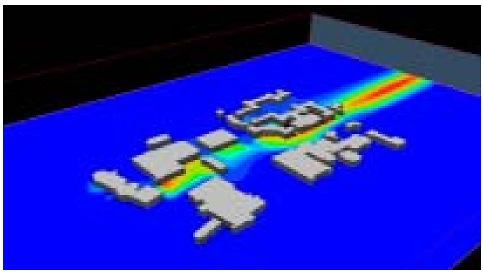
(e)



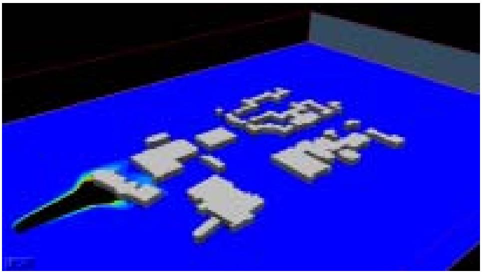
(f)



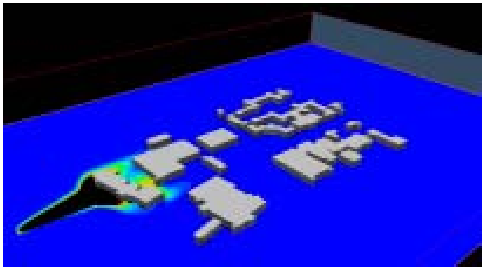
(i)



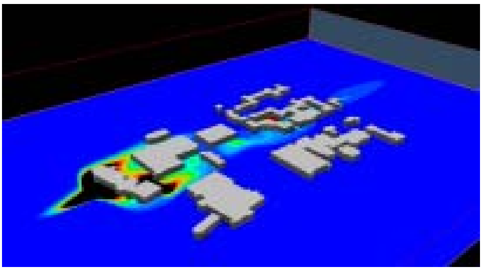
(j)



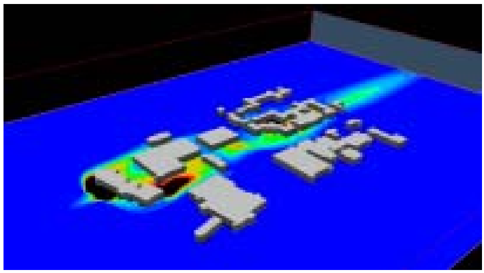
(c)



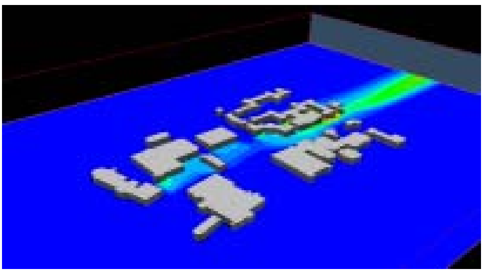
(d)



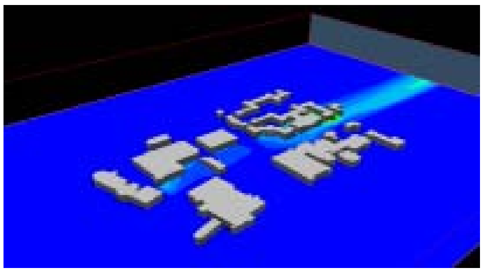
(g)



(h)

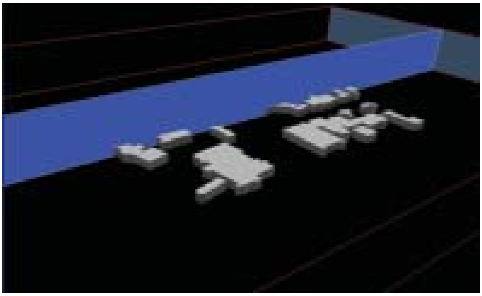


(k)

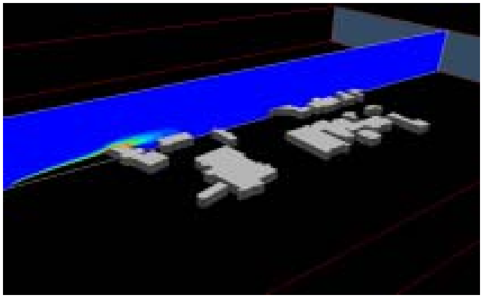


(l)

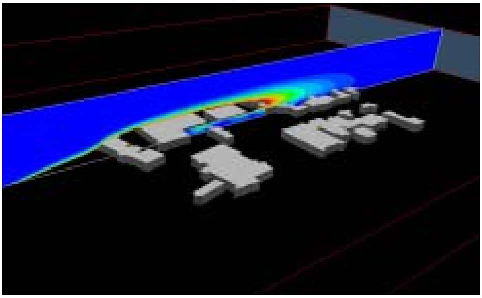
26



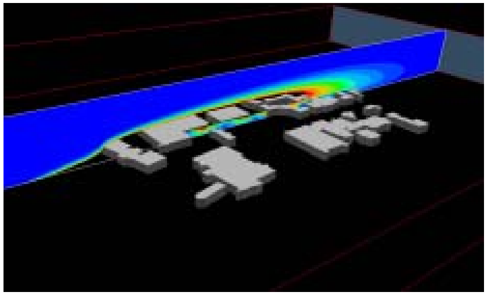
(a)



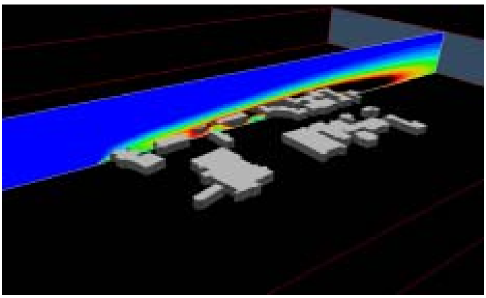
(b)



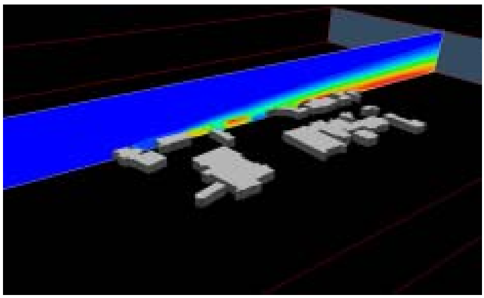
(e)



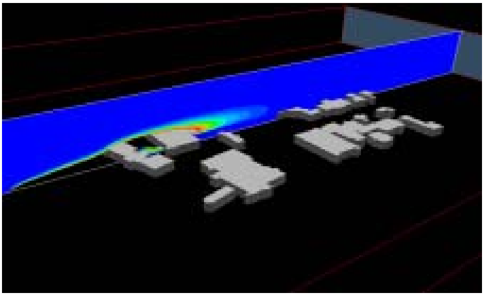
(f)



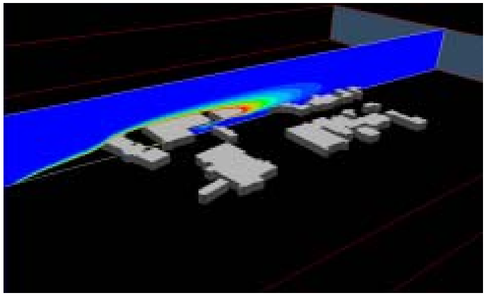
(i)



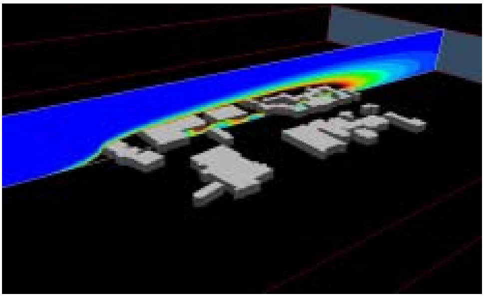
(j)



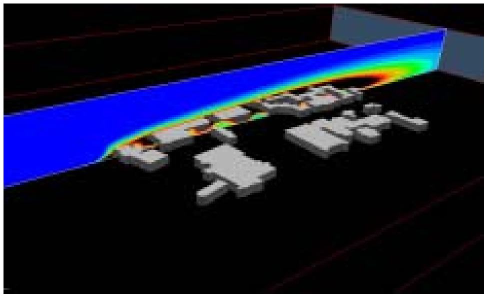
(c)



(d)



(g)



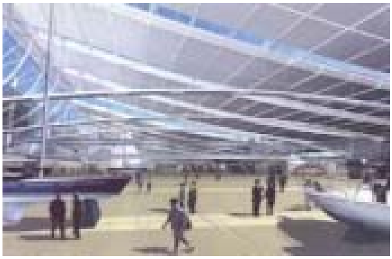
(h)

27

08

Indoor Applications

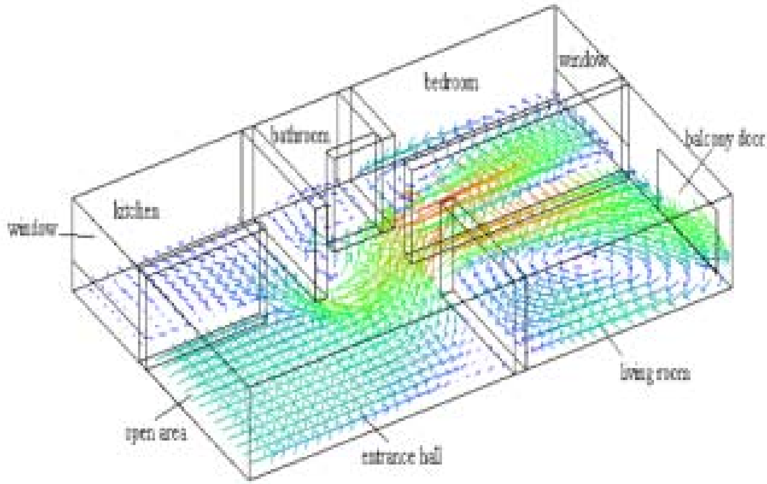
28



29

I am just talking about different applications of CFD right now. I will go back and forth between indoor and outdoor. Here are some indoor applications of CFD. It can be used to study natural ventilation and night cooling, double-skin fa□ades, ceiling cooling panels and slab cooling or heating and displacement ventilation—all kind of different mechanical systems.

Here is the David L. Lawrence Convention Center in Pittsburg, Pennsylvania (Figure 28). We have a very nice shell roof that imitates the slope of the bridges. It is a huge space (Figure 29). One interesting thing is that they use natural ventilation. I look at the plan what they are doing inside; they are using natural ventilation and natural light. I always take my students to see and admire what they have done there. The natural ventilation was designed and planned with CFD computational results. Of course, the results have assisted natural ventilation, which means they are using the regular HVAC system from time to time, but their main cooling capacity, especially in spring or fall, can definitely rely on natural ventilation. So, this is a relatively new concept because we believe that natural ventilation is only suitable



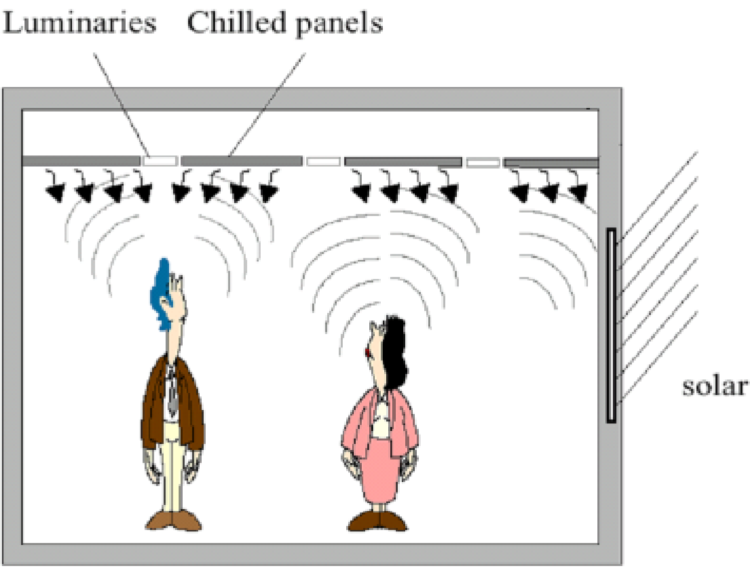
30

for apartment buildings. This project has shown that even a large building such as a convention center, can be properly designed to utilize natural ventilation.

This is what I was really talking about. If you have airflow simulation, you can look into apartment buildings. This is an apartment building that is actually built in Shanghai, China (Figure 30). I actually worked on this together with an architect. We design the distribution of different rooms such as kitchens, bathrooms and balconies based on the wind coming from this window. This is the more traditional work of natural ventilation and CFD.

Double-skin fa□ade (Figure 31); you basically have 2 pieces of parallel glass. This is one layer of glass and you have inside another layer of glass. You can use the wind to go through this or you can help it with a little fan. You can actually cool down the building and take some of the solar load off before it comes inside the building. I am just thinking that you have such a pretty high wind here; this double skin fa□ade could probably work in Thailand. This is the Helicon Building at Finsbury Pavement in London. We are still struggling to convince the owners of the buildings to install systems like this one.

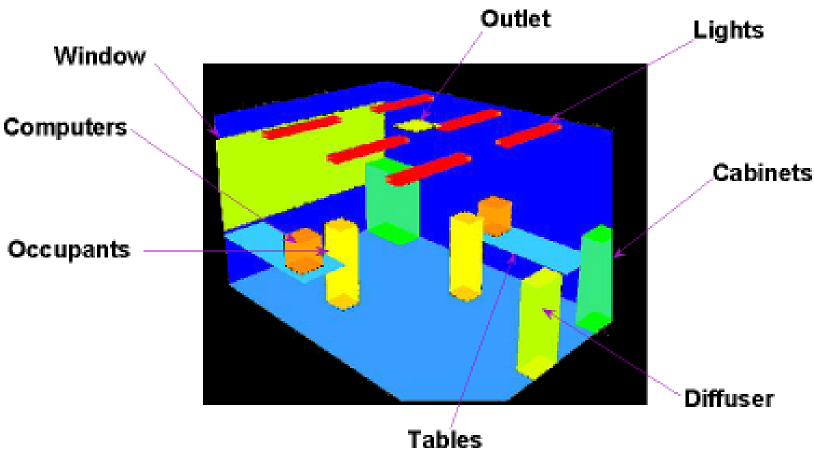
31



32

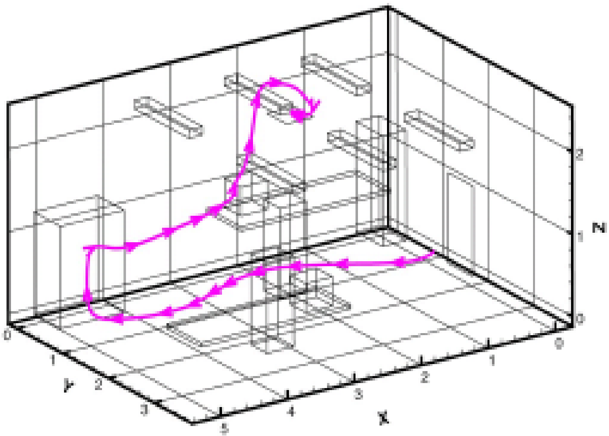
Cooled ceiling (Figure 32); I would definitely not recommend this in Thailand, but it is very popular in Europe and in the US if you are retrofitting buildings. Instead of the dropped ceiling, you can use a panel with cold water. So, it does not take as much space as an HVAC system does. You send cold water there and its radiation actually cools down the space. The reason why it could not work in Thailand is that the cold surface would have lot of condensation because you have a very humid climate. The system is pretty successful in climate that has less humidity.

This is something that I would like to show you in a little bit more detail. I do not know if you have heard of displacement ventilation. A standard mixing diffuser would produce a kind of drought, like when you feel the cold air going around your body. If you sit here, you cannot go anywhere. You have to suffer with drought for several hours, which is not very pleasant. So, this displacement ventilation is a system that mimics natural ventilation (Figure 33). Here are the diffuser, the human simulator, some furniture, computer, lighting system and outlets. Here is also a window like that of typical office space.

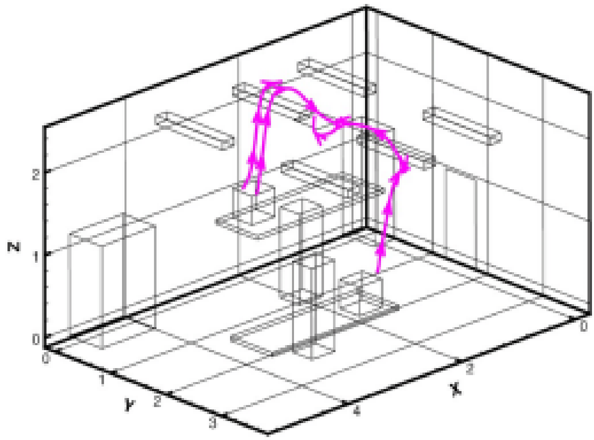


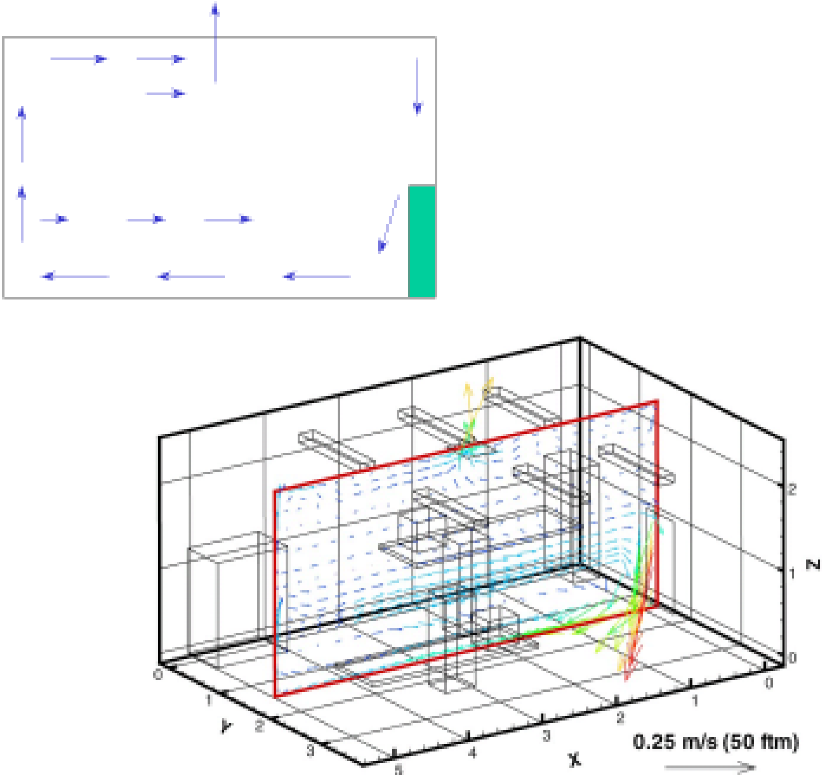
33

Displacement ventilation is basically composed of a huge perforated plate. It is just a plate sitting here on the floor, and it works like an opened window when the weather is calm and the outdoor air is slightly cooler than that inside of your house. That is what displacement ventilation is all about. It is just a very slow velocity, sending the air that is slightly cooler than your environment. It can be very pleasant for the occupants. What happens is that the particles of air go around and go immediately to the exhaust so there is no mixing of different pollutions (Figure 34). If this thing, a computer, is producing some kind of pollutant, you know where it goes, and it goes directly to the exhaust. So, the system is fairly popular because it takes contaminants directly to the exhaust.



34

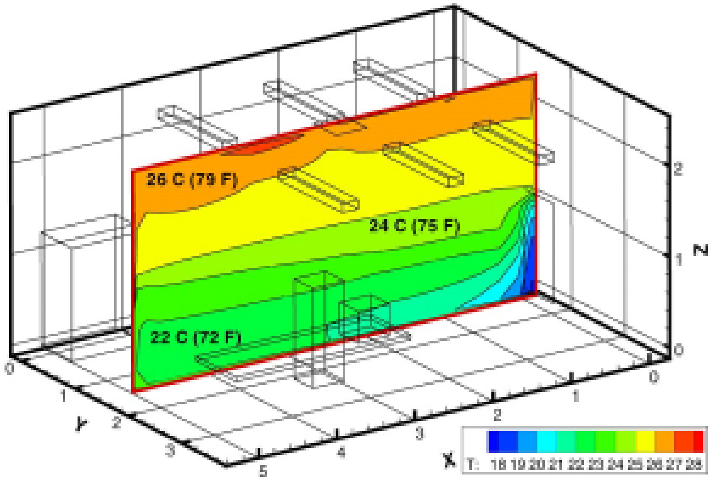




35

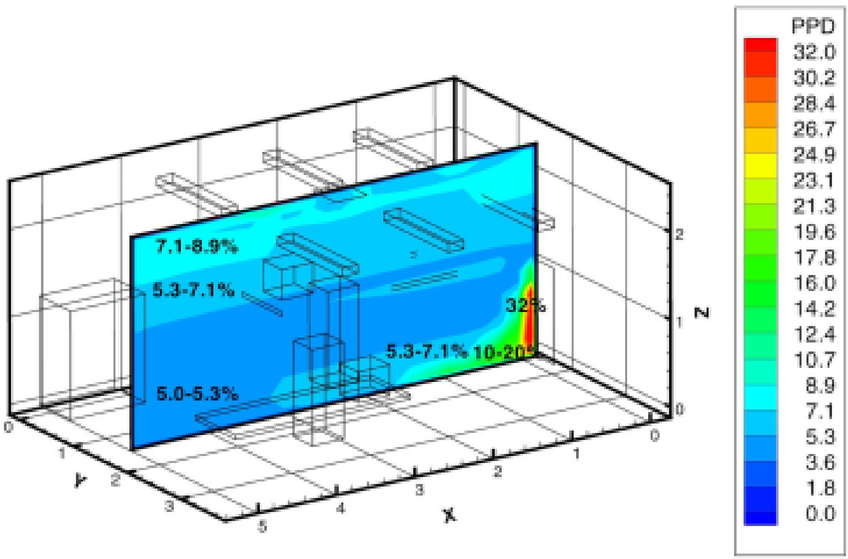
Here is an observed airflow pattern in an environmental chamber (Figure 35, top), the one that I had shown to you previously. And here is the CFD result for velocity (bottom). Basically when we compare this observed with calculations and we look at the details, we have got exactly the same result.

Here is the temperature with the displacement ventilation (Figure 36). You can see here at the floor level, you have around 22°C. 24°C is where occupants are. It is 26°C at the ceiling and it crumbles to 30-36°C if you have a high ceiling. It is a suitable system for atria. You can save a lot of energy if you use displacement ventilation because it promotes natural stratification, which would happen even



26 C (79 F)
24 C (75 F)
22 C (72 F)
T: 18 19 20 21 22 23 24 25 26 27 28

36



7.1-8.9%
5.3-7.1%
32%
5.3-7.1% 10-20%
5.0-5.3%
PPD
32.0
30.2
28.4
26.7
24.9
23.1
21.3
19.6
17.8
16.0
14.2
12.4
10.7
8.9
7.1
5.3
3.6
1.8
0.0

37

with the low wind and natural ventilation. That is why I compare displacement ventilation with an opened window. And of course, where it is very close to the diffuser, you have a very low velocity and basically should not spend too much time there.

We look at something called Percentage of People Dissatisfied (PPD) (Figure 37). This is the percentage of people who would be unhappy in that environment. It is from a perspective of thermal comfort. If you look at the numbers that I have here, basically these are really good numbers except here where you have 32 percent. 20 percent is acceptable and considered to be a very good design. If you have 20 percent of PPD and below, you have made a very good design. You see that the lowest number here is 5 percent of people who will be unhappy no matter what I do with my system. Have you been aware of that? Did you get an evaluation of your work? We do the evaluation for teaching. No matter what I have done in my teaching, there always are portions of students who are very unhappy with my course. I stop worrying after I have been learning about this factor. This is based on extensive statistical analysis, so no matter what you do, you will always have people that are unhappy with your building. And that is fine; let them be, as long as the greatest majority is very happy with the design.

As you have seen a little bit of airflow simulation and contaminant simulation around those campus buildings, you can look inside a specific place. You can look if the contaminant comes from windows and outside or from other source 2 and 1 that I have here (Figure 38). This is actually a military command center, where you have a corridor in between and two offices with people inside. You have a copy machine, computers and so on. I have a source of contamination underneath the desktop of a poor guy over here and I also have a source of contamination in this corridor. We look at the concentration levels at the breathing plane, which is approximately 1.2 meters from the floor area.

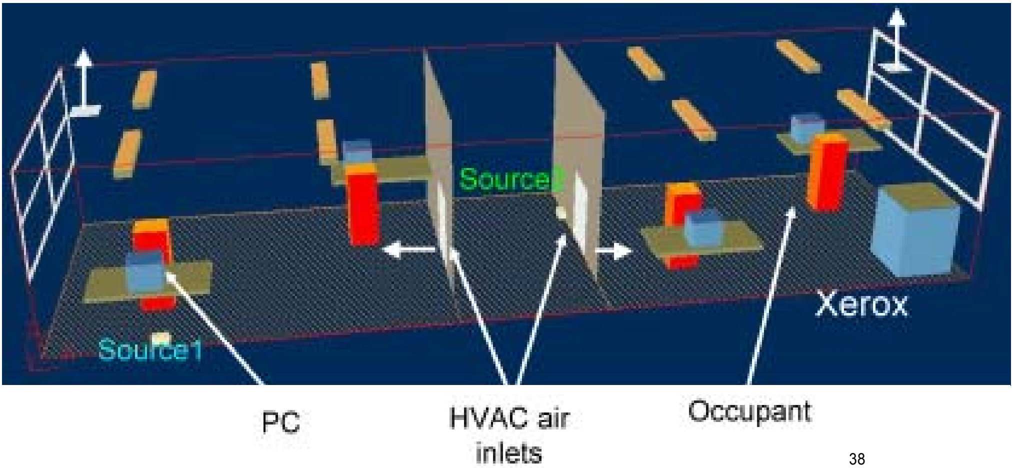
This is an HVAC system with displacement ventilation. The CFD result shows that the high level of concentration is impacting this person, because he is too close to source 1. The contaminant actually goes around the entire building. It is spreading around and unfortunately, the HVAC system works undisturbed, but the people are gone. And the more time goes by, the air will be cleaner because the air-conditioning system is working on cleaning the building.

Can we improve the design in such a way that our air-conditioned system helps protect the building? We did something called ‘Emergency Air Supply (EAS)’ exhaust system (Figure 39). It is an additional HVAC system, which is an additional expense to your building. But trust me if you try to protect important buildings, the owners are willing to throw in the money. We have here the additional system; you can see the displacement ventilation, my original exhaust and my original diffuser.

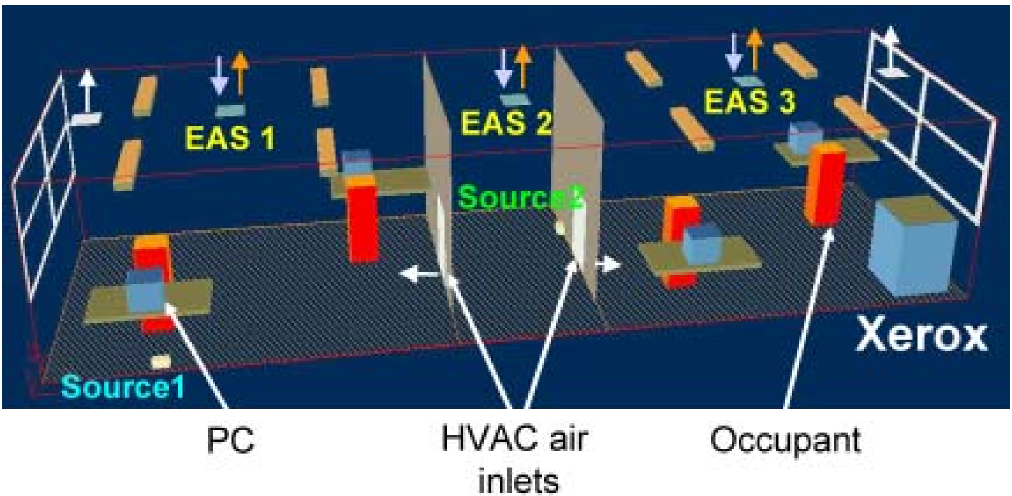
Again, we can see how this system works. We run another CFD simulation demonstrating the same scenario for source 1 and 2. The system is now doing a pretty good job by trying to help the building’s occupants. It is completely safe around most areas. For the person who is too close to the source, it is not very worthwhile designing the system for. But for the rest of the occupants, it is very effective. We use CFD actually to define the flow rate, so you can do the very neat thing with CFD.

This is some conclusion for this particular simulation. Without emergency air supply, the chemical and biological agents can spread easily around the building. In fact, the corridor works as a duct for the HVAC system. It spreads the contamination throughout the building so the corridor is the weakest point of the building. When you evacuate, where will you go? The first place to the outside of the building is definitely the corridor. What we prove with this simulation is that it is the worst place to go to.

Questions that are answered through this type of simulation are: how fast will different chemical and biological agents be distributed to space? How many sensors are enough to detect chemical and biological agents in time and how to set them up effectively? And is it possible to isolate contaminated air in specific zones of a building? All of these answers can help design better buildings.


Source2
Source1
Xerox
PC
HVAC air inlets
Occupant

38


EAS 1
EAS 2
EAS 3
Source2
Source1
Xerox
PC
HVAC air inlets
Occupant

39