BCA ACADEMY

Design and Simulation for Natural Ventilation

(6th Intake)

School of Graduate Development and Management



The Education and Research arm of the Building and Construction Authority

Certification Course for Green Mark Professionals Core Module:
4-Day Course on Design and Simulation for Natural Ventilation

Course Instructors:

Mr Owen Wee, Surbana International Consultants Pte.Ltd.

Dr. POH Hee Joo and Dr. George Xu Xiangguo
Institute of High Performance Computing, A*STAR

Day 1:

8:30am – 9:00am	Registration	

9:00am – 5.30pm	• Introduction to passive design
	  • Tropical climatic condition
	  • Site topography and surrounding environment
	• Factors affecting ventilation design
	  • Building orientation and geometry
	  • Façade design and fenestration types, location, quantity and size
	• Natural ventilation strategies
	  • Single-sided ventilation
	  • Cross ventilation
	  • Stack ventilation
	• Achieving thermal comfort under tropical conditions
	  • Concept of thermal comfort
	  • Key factors affecting thermal comfort
	  • Thermal comfort prediction and evaluation
	• Thermal comfort index
	• Case study
		Mr Owen Wee

Day 2:

8:30am – 9:00am	Registration	
9:00am – 10:30am	**Introduction to CFD in Green Building Application** • Review about natural ventilation design • CFD-aided natural ventilation design of green buildings, and case studies	**Dr. Poh Hee Joo & Dr. George Xu**
10:30am – 11:00am : Tea break		

11:00am - 12:30pm	**Introduction to CFD in Green Building Application**
	• CFD methodologies recommended by BCA
	  o Residential building (RB)
	  o Non-Residential Building (NRB)
	• Available CFD tools and capabilities – a consultant's perspective
	• Case studies and examples of CFD simulation results used in optimizing natural ventilation in the built environment
		Dr. Poh Hee Joo & Dr. George Xu

12:30pm - 1:30pm : Lunch		
1:30pm – 3:00pm	**Introduction to CFD process** **Demo of CFD software** • Brief demo and software usage	**Dr. Poh Hee Joo & Dr. George Xu**
3:00pm – 3:30pm : Tea break		
3:30pm – 5:00pm	**Hands on with CFD software** • Tutorial for Practice 1 - Fluid Flow and Heat Transfer in a Mixing Elbow (step by step guideline about using ANSYS Fluent in ANSYS Workbench)	**Dr. Poh Hee Joo & Dr. George Xu**
End of Day 2		

Day 3:

9:00am – 10:30am	**Assignment briefing** **Introduction to CFD** • Mathematical Background • Turbulence Model • Wind Effect on Building Structure • Verification and Validation in CFD **Creating 3D geometry model**	**Dr. Poh Hee Joo & Dr. George Xu**
10:30am – 11:00am : Tea break		
11:00am - 12:30pm	**Meshing computational model** **Boundary conditions** **Numerical settings** **Post processing of simulation results**	**Dr. Poh Hee Joo & Dr. George Xu**
12:30pm - 1:30pm : Lunch		
1:30pm – 3:00pm	**Hands on with CFD software** • Tutorial 2 Simulation of natural ventilation around and across a Silsoe Cube	**Dr. Poh Hee Joo & Dr. George Xu**
3:00pm - 3:30pm : Tea break		
3:30pm – 5:00pm	**Hands on with CFD software** • Tutorial 3 Simulation of natural ventilation around and across a Silsoe Cube (continue..)	**Dr. Poh Hee Joo & Dr. George Xu**
End of Day 3		

Day 4:

9:00am – 9:45am	**CFD model demonstration** • **Natural ventilation scenario in BCA Academy 2-Storey Training Block – Engineering Perspective**	**Dr. Poh Hee Joo & Dr. George Xu**
9:45am – 10:30am	**CFD case studies** • **CFD Case Studies for Urban and Built Environment – Researcher Perspective**	**Dr. Poh Hee Joo & Dr. George Xu**
10:30am – 11:00am : Tea break		
11:00am - 12:30pm	**Hands-on Assignment 1 – Use CFD simulation tool (ANSYS-FLUENT) to perform ventilation simulation modelling and analysis in order to ensure good natural ventilation within the functional unit (100%)** • Simulation of natural ventilation around and across a Silsoe Cube (cont…)	**Dr. Poh Hee Joo & Dr. George Xu**
12:30pm - 1:30pm : Lunch		
1:30am – 3:00am	**Hands-on Assignment 1 – Use CFD simulation tool (ANSYS-FLUENT) to perform ventilation simulation modelling and analysis in order to ensure good natural ventilation within the functional unit (100%)** • Simulation of natural ventilation around and across a Silsoe Cube (cont…)	**Dr. Poh Hee Joo & Dr. George Xu**
3:00pm - 3:30am : Tea break		
3:30pm – 5:00pm	**Group presentation of simulation results** • Simulation of natural ventilation around and across a Silsoe Cube (end) **Closure** • Feedback session	**Dr. Poh Hee Joo & Dr. George Xu**
End of Day 4		

Part 1 : INTRODUCTION

Our World

Our Mother Earth

- The last 10,000 years have been a period of relatively stable climate;
- Over the paleoclimatic record, there have always been variations in climate;
- Changes have generally happened so slowly that forms of life have had time to adapt or migrate.
- The climate system consists of many subsystems (e.g. *atmosphere, hydrosphere, biosphere, lithosphere*), linked together in a non-linear fashion.



Source: NASA

Our Mother Earth

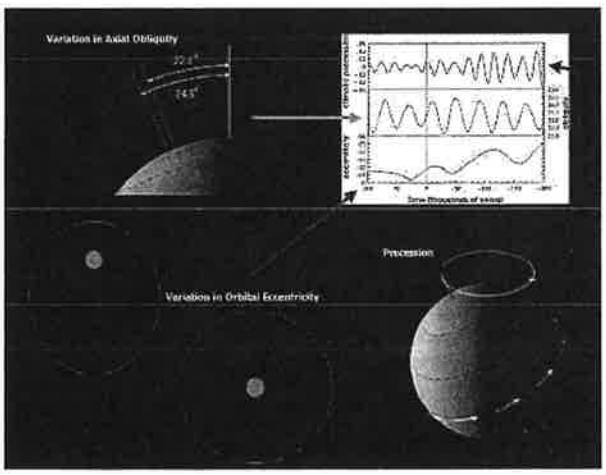
- To alter global climate one of the following climate forcings has to occur:
  1. Alteration of the Earth's orbit and movements (Milankovitch Cycles);
  2. Variation in the intensity of solar radiation (e.g. 11-year sunspot cycle);
  3. Shift in the geological equilibrium of the planet (e.g. volcanic eruptions);
  4. Variation in the equilibrium of oceanic currents (e.g. Gulf Stream, El Nino)
  5. Modification of the Earth's albedo (i.e. the reflectivity of the surface);
  6. Changes in the chemical composition of the atmosphere (humans).
- Climate forcings operate on different time scales, from geological eras up to more 'human' temporal dimension;
- Many complex 'feedback' mechanisms can amplify ('positive feedback') or diminish ('negative feedback') climate forcings;
- Earth's climate has varied many times, and Carbon (especially CO_2) has often played an important role in regulating its dynamic equilibrium.

Source : Nottingham University : Dr Sergio A.

Milankovitch theory

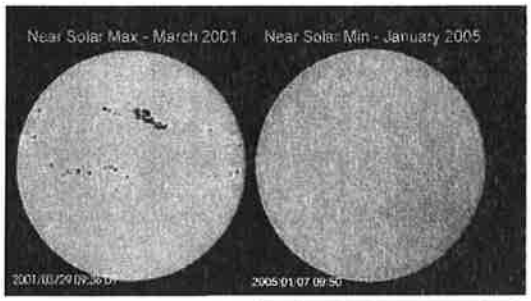
Milanković mathematically theorized that variations in eccentricity, axial tilt, and precession of the Earth's orbit determined climatic

There are two major cycles; one every 100,000 years and a weaker one every 413,000 years.



Solar cycle (sunspot cycle)

Solar activity is correlated to the number of sunspots on the Sun. As the number of sunspots goes up, solar activity occurrences go up And more solar radiation will reach earth



2013 was estimated to be the peak of solar activity for the 24th cycle

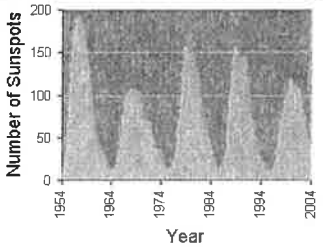


Image Source : SOHO/NASA/ESA

Geological disruptions

When **volcanoes** erupt, they emit a mixture of gases and particles into the air. Some of them, such as ash and sulphur dioxide, have a **cooling effect**, because they (or the substances they cause) reflect sunlight away from the earth. Others, such as CO2, cause warming by adding to the **greenhouse effect**



Earth Albedo

Albedo is a measurement of the amount of light reflected from the surface of a celestial object, such as a **planet**, satellite, comet or asteroid. The **albedo** is the ratio of the reflected light to the incident light: and has values between 1 to 0: a black object that absorbs all light and reflects none.

The ice cap are the most effective in reflecting radiation

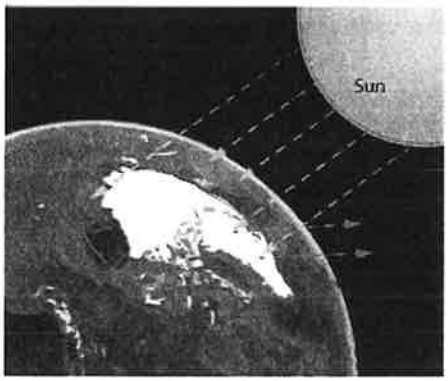
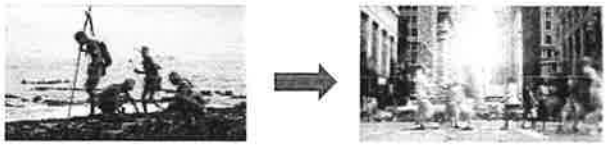


Image source :https://www.indiegogo.com



0.3

Human Activities ???





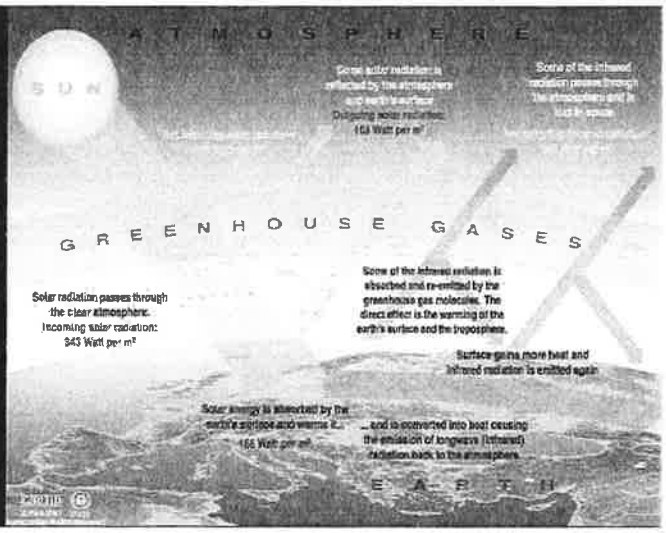
Where does the energy come from ?

Our Mother Earth

The energy reaching the higher strata of the atmosphere each second is 1,353 W/m²

The energy incoming per second to the terrestrial system – averaged over the entire planet – is 240 W/m²

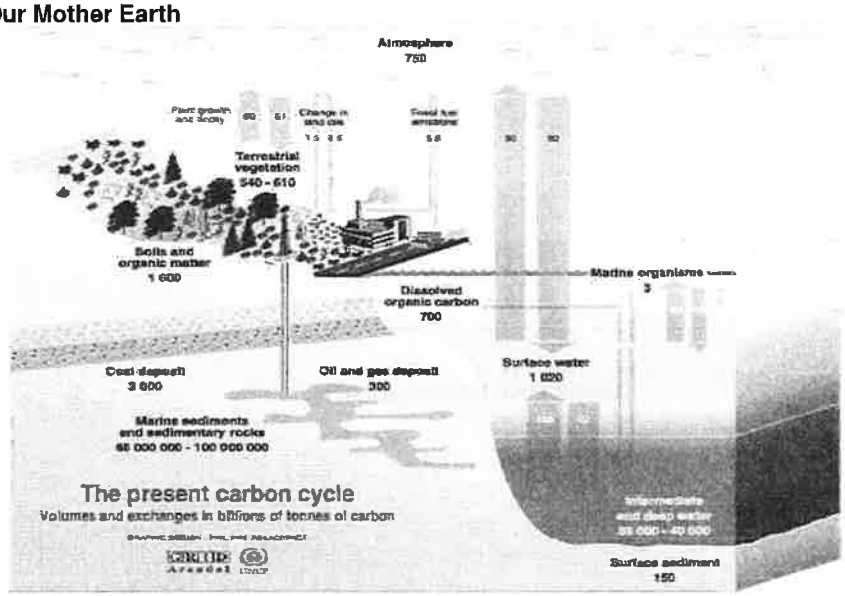
Without the natural greenhouse effect the Earth's surface would be about 30°C colder than the average +14 °C we enjoy at present



The Greenhouse effect

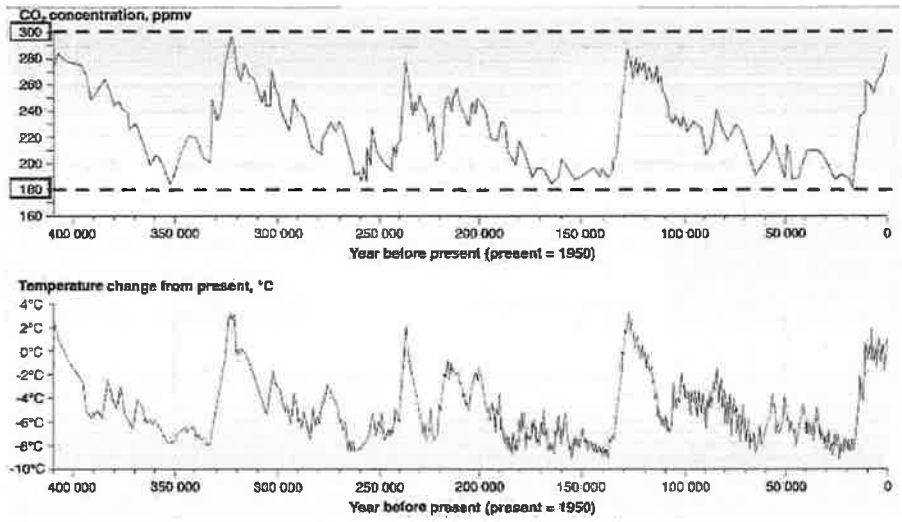
Source: IPCC, Climate Change 2007

Our Mother Earth



Passive design & natural ventilation 2017

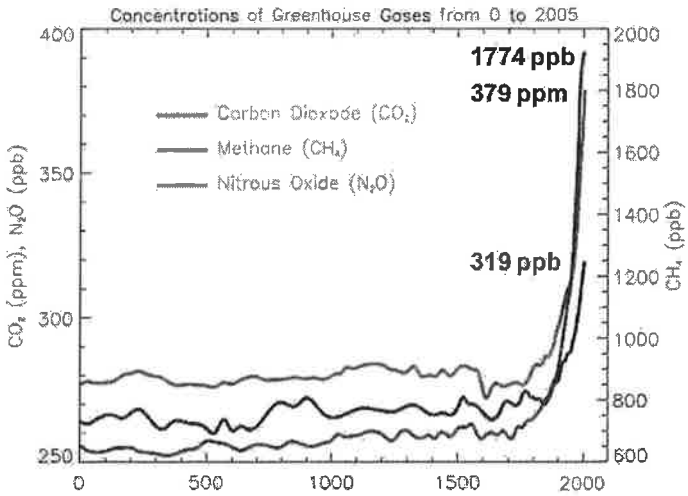
Our Mother Earth



Temperature and CO2 concentration in the atmosphere (from the Vostok ice core)

Source: IPCC, Climate Change 2007

Our Mother Earth : Last 50 years



Atmospheric concentrations of long-lived GHG gases over the last 2,000 years

Source: IPCC, Climate Change 2007

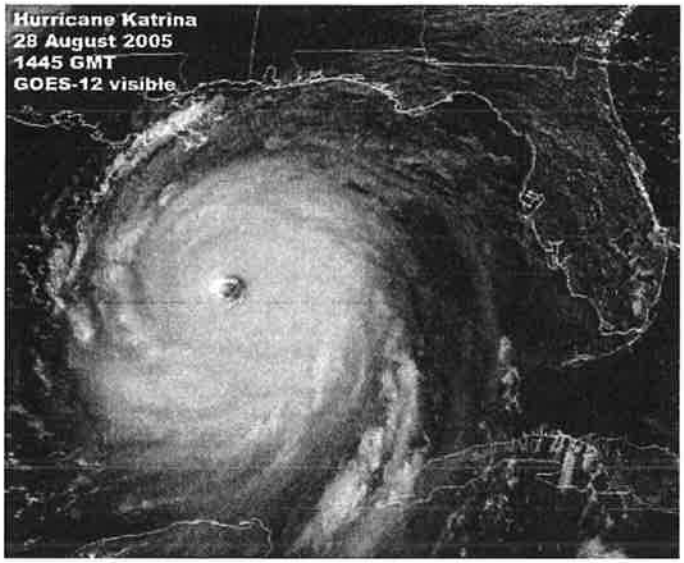
400 PPM WAS EXCEEDED IN 2015

Passive design & natural ventilation 2017

Our Mother Earth : Last 50 years

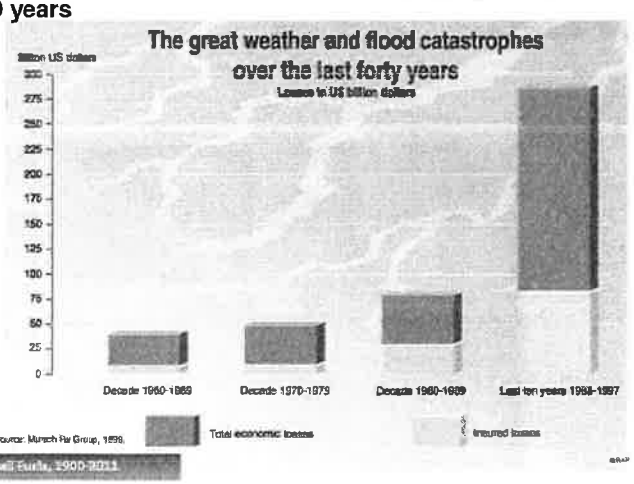


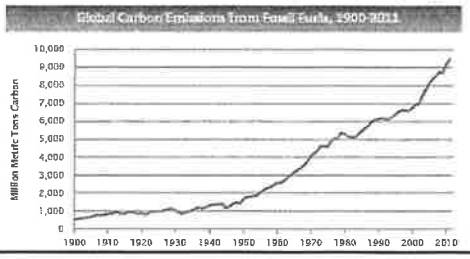
Our Mother Earth : Last 50 years



CLASS 5

Our Mother Earth : Last 50 years



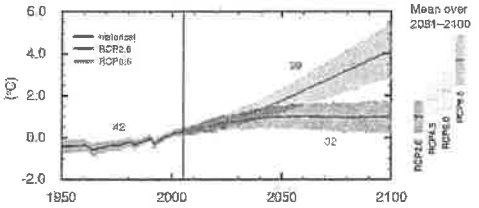


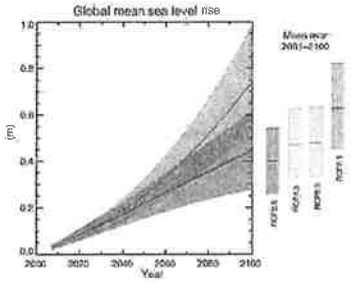
www.epa.gov

Our Mother Earth : Last 50 years

Why the Topic?

- The climate is changing and our activity is the cause
- Our resources are dwindling
- Our consumption is increasing in spite of this.
- We need to **work** together to reduce our impacts.





The graphs from the UN climate pane's 2013 report showing the best case and worst case scenarios for the climate projection of the earth. The report details the physical evidence for climate change and states with 95% confidence that humans are the "*dominant cause*". (IPCC WG1 AR5, 2013)

Source : SGBC , GSI

Our Mother Earth : Last 50 years

Warming of the climate system is unequivocal, and since the 1950s, many of the observed changes are unprecedented over decades to millennia. The atmosphere and ocean have warmed, the amounts of snow and ice have diminished, sea level has risen, and the concentrations of greenhouse gases have increased.

(Extract from Working Group I Contribution to the IPCC Fifth Assessment Report *Climate Change 2013: The Physical Science Basis* Summary for Policymakers – by the UN)

Each of the last three decades has been successively warmer at the Earth's surface than any preceding decade since 1850. In the Northern Hemisphere, 1983–2012 was *likely* the warmest 30-year period of the last 1400 years (*medium confidence*).

(Extract from Working Group I Contribution to the IPCC Fifth Assessment Report *Climate Change 2013: The Physical Science Basis* Summary for Policymakers – by the UN)

Our Mother Earth : Last 50 years

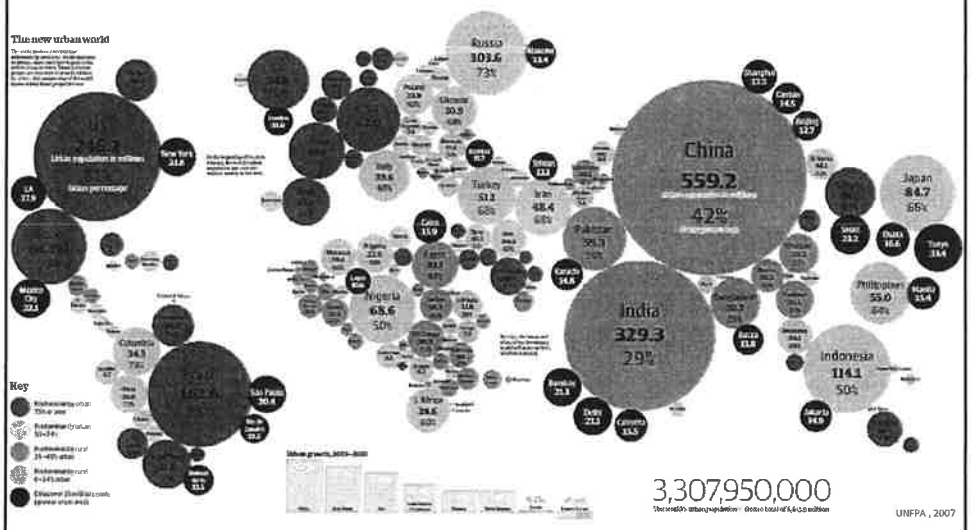
Conclusion

The observed warming of the atmosphere and the oceans, together with ice mass lost, support the conclusion that it is extremely unlikely that climate change of the past 50 years can be explained without external [human] forcing, and very likely that it is not due to known natural causes alone"

IPCC, Climate Change 2007, Fourth Assessment Report, 2007

Our Mother Earth : Last 50 years

Introduction : Population growing – 人口增长



Our Mother Earth : Last 50 years

People moving into cities



http://www.visitourchina.com/images/fileUpload/100507182013950.jpg

Our Mother Earth : Last 50 years

People moving into cities
– 人口增长，城镇化

More than 50% of the world population live in cities

United Nation , UNFPA , 2008

2011 marked the first year that China"s urban population was greater than it"s rural population

www.bloomberg.com

690 million vs. 656 million

www.edc.caln CanadaMark

中国每年新增城镇人口1000万人
In China,10 million people move into cities every year

中国改革发展研究院

World to have 8.3 billion people by 2030

United Nation , April 2013

Our Mother Earth : Last 50 years

People moving into cities

Overcrowding
拥挤



hatethefuture.tumblr.com

Recreation space
绿地与休闲



www.geomancy.net -

Slumps and pollution
平民区和污染



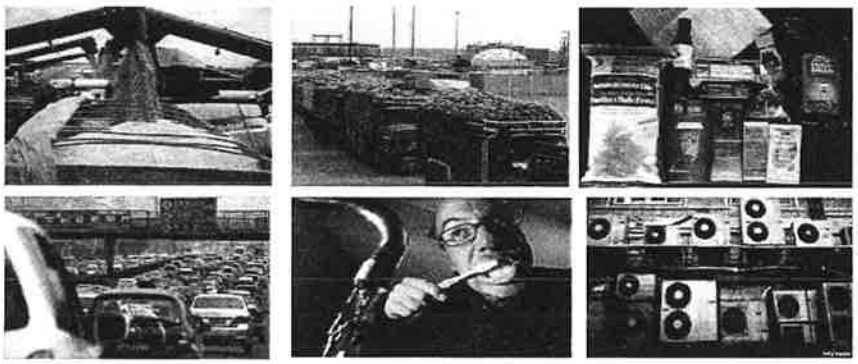
Source : ngoabroad

Resources
资源



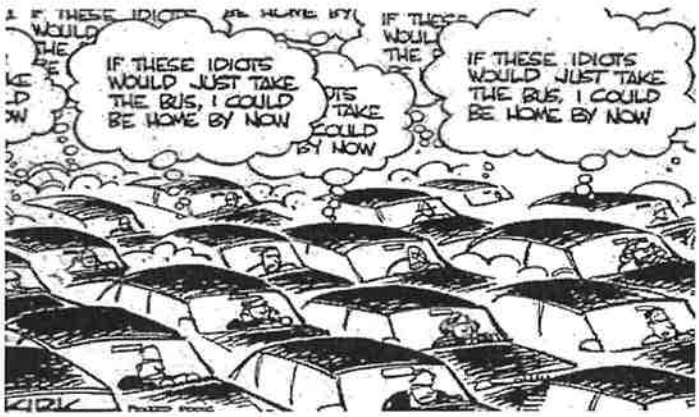
www.thebeijinger.com

Our Mother Earth : Last 50 years



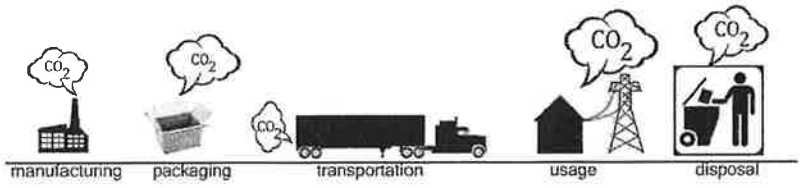
Life of high carbon foot print

Our Mother Earth : Last 50 years



Our Mother Earth : Last 50 years

Carbon Footprint



All activities release carbon dioxide or other green house gases, during the various phases in its life cycle. Carbon footprint is the sum total of all the green house gases emitted into the atmosphere during the entire lifecycle of the product/material.

Source : SGBC , GSI

Our Mother Earth : Last 50 years

What about Singapore ???

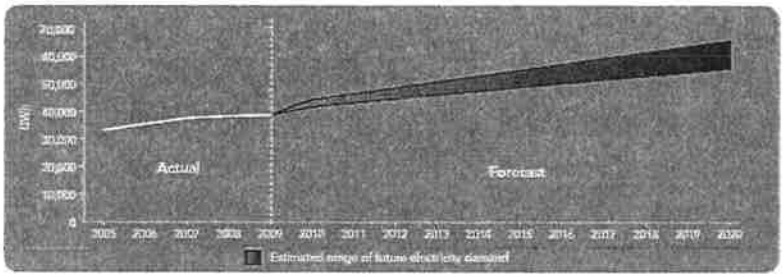


Fig 3.1- Estimated range of future electrical demand. Source:- EMA, 2010, pg 8

Estimated range of future electrical demand. Source:- EMA, 2010, pg 8

Our Mother Earth : Last 50 years

Impact on Singapore

- Rising sea levels – greater land constraints.
- Increase in flash floods
- Hotter dryer periods
- Change in wind direction and speed
- A less liveable environment

Oil	30 years
Natural Gas	50 years
Coal	190 years

Table Adapted from Vale, B&R (1991)

What about Singapore ???

Average Water Consumption (CuM)

Average consumption of Water (CuM)													
Premises Types	Jun-14	Jul-14	Aug-14	Sep-14	Oct-14	Nov-14	Dec-14	Jan-15	Feb-15	Mar-15	Apr-15	May-15	Jun-15
HDB 1-Room	9.8	9.6	9.9	9.5	9.3	9.6	9.3	9.2	9.2	9.0	9.7	9.8	9.6
HDB 2-Room	12.9	12.5	13.1	12.9	12.3	12.7	12.2	12.4	12.1	11.9	13.1	12.5	12.7
HDB 3-Room	14.7	14.4	14.9	14.5	14.2	14.3	13.9	13.9	13.8	13.8	14.6	14.0	14.2
HDB 4-Room	18.8	18.2	18.8	18.5	17.9	18.2	17.6	17.8	17.7	17.6	18.5	17.6	17.9
HDB 5-Room	19.7	19.5	19.8	19.6	18.9	19.1	18.5	18.6	18.7	18.6	19.6	18.4	18.8
HDB Executive	21.7	20.9	21.8	21.8	20.8	21.1	20.5	20.6	20.9	20.9	21.7	20.5	20.9
Apartment	16.1	15.5	16.1	16.1	15.9	15.8	15.2	15.3	15.1	15.4	16.0	15.4	15.2
Terrace	28.4	27.5	28.8	28.1	28.6	28.6	27.7	27.6	28.3	29.2	29.3	27.3	28.2
Semi-Detached	32.1	31.7	32.6	33.4	33.4	33.2	31.1	31.7	33.0	34.8	35.2	32.0	32.3
Bungalow	47.0	50.5	50.1	52.1	52.1	66.3	48.0	48.5	52.9	56.5	60.3	53.9	46.3

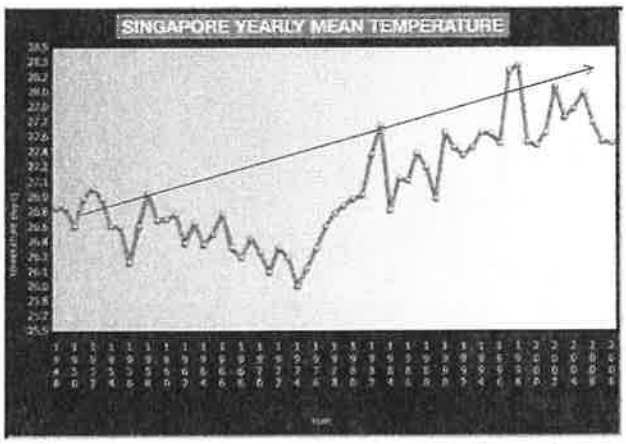
Note: Average consumption is computed based on total consumption divided by total number of accounts in the respective premises types.

Singapore's per capita domestic water consumption has been brought down from 165 litres per day in 2003 to the current 150 litres. The target is to lower it to 147 litres by 2020 and 140 litres by 2030.

http://www.singaporepower.com.sg/irj/servlet/prt/portal/prtroot/docs/guid/609fede-040d-2f10-15b4-de464caacd39

Passive design & natural ventilation 2017

What about Singapore ???



Air temperature measurements were made hourly. All hourly measurements made in a year are averaged to give the yearly mean temperature.

Source : NEA , Weatherwise , 2009

What about Singapore ???



July was earth's hottest month on record

Singapore recorded highest temperature in 10 years at 36.7 deg C on April 13 2016

Source 2016 NEA

Study: Most cities will be too hot to host summer Olympics by 2085

BERKELEY (California) • In 70 years, most cities in the Northern Hemisphere will be unfit to host the summer Olympics due to rising temperatures associated with climate change, according to a medical journal's findings published this week.

"Our study using climate change projection shows that there will be very few cities at the end of the century that will be able to hold the summer Olympics as we know them today," said researcher John Balmes, a professor of public safety at the University of California, Berkeley, who co-authored the paper.

The findings, which originally appeared in The Lancet, indicate that by 2085, only eight Northern hemisphere cities outside of Western Europe are likely to be cool enough to host the summer Games. San Francisco would be one of just three North American cities that could serve as hosts.

least 600,000 residents, the size considered necessary for hosting the Games.

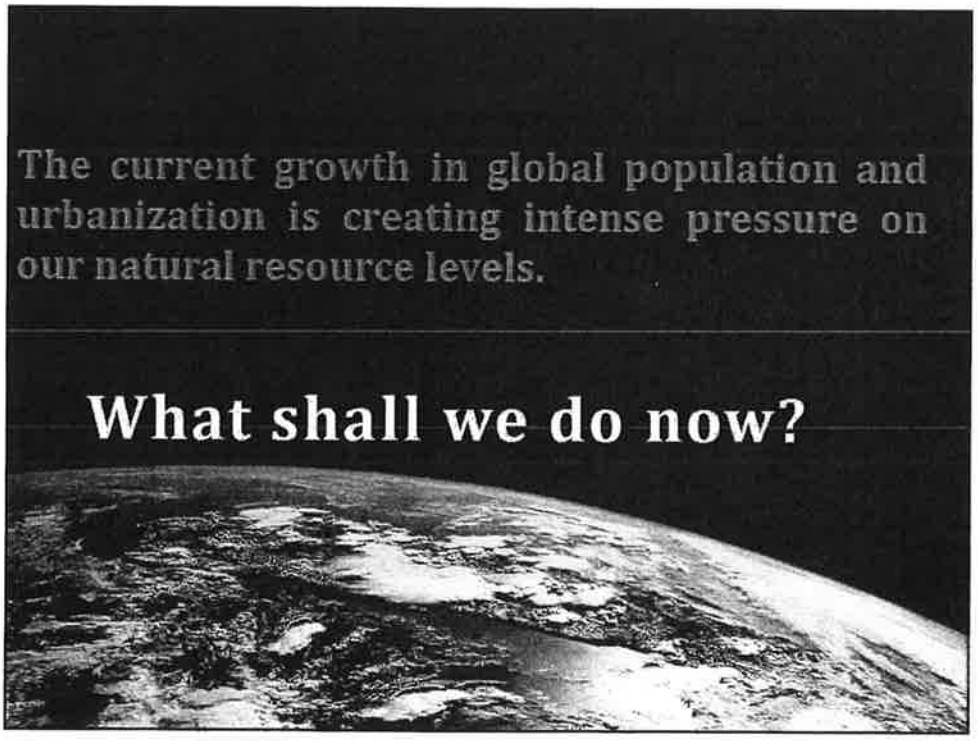
The findings assumed that any city with more than a 10 per cent chance of having to cancel a matathon due to temperatures exceeding 26 deg C would not be a viable venue.

"If you're going to be spending billions of dollars to host an event, you're going to want to have a level of certainty that you're not going to have to cancel it at the last minute," said UC Berkeley's Professor Kirk Smith, a co-author on the study.

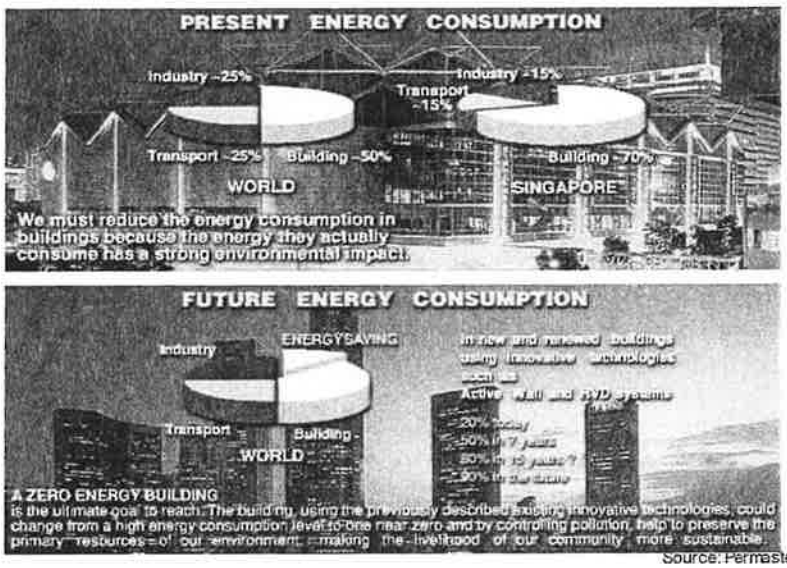
According to the projection models, by 2085, all of the cities that are or were in contention for either the 2020 or 2024 summer Olympics – Istanbul, Madrid, Rome, Paris and Budapest – would be unfit to host the games.

Tokyo, the host of the 2020 games, would also be too hot to ensure athlete safety.

Singapore Straits Time 19 Aug 2016



The Solution



Source: Permasteelisa, 2001

The Solution

Buildings use about:

40% of Global Energy

25% of Global Water

& 1/3

of Green House gas emissions

Source : SGBC , GSI

The Solution

- Residential and commercial buildings account for almost 39 percent of total U.S. energy consumption and 38 percent of U.S. carbon dioxide (CO_2) emissions.[1] Nearly all of the greenhouse gas (GHG) emissions from the residential and commercial sectors can be attributed to energy use in buildings

Source : -Centre for climate and energy solutions

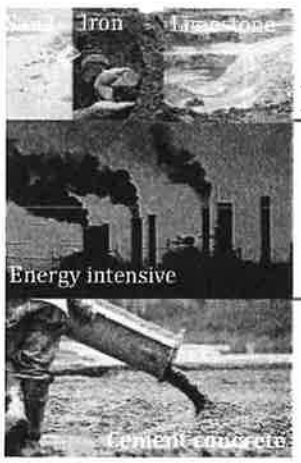
-China and India is unprecedented and estimated to represent half of the world's total market today. From a property owner perspective, resources, space utilisation, and long-term operational costs are increasingly factored in during planning. With buildings responsible for about half of all energy consumption and greenhouse gas emissions.....

-Source :- http://www.asiagreenbuildings.com Aug 2014

- Commercial building and residential building use up 51.4% of our energy with industries using up 42.6%

Source:- Singapore Energy Statistics 2015 , EMA

The Solution



The Cement manufacturing process

Raw Materials mined from earth

When processed through **energy intensive** and **carbon emissive** industrial process.

110-120 kWh of electrical energy required per ton of cement produced.

Forms **cement**, a predominant construction material throughout the world

Source : SGBC - GSI

The Solution

Embodied Energy



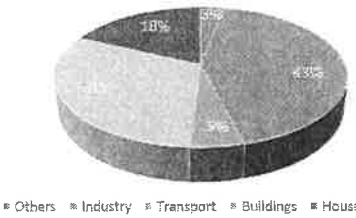
Embodied Energy is defined as the energy used during the entire life cycle of a product including the energy used for manufacturing, transporting, and disposing of the product.

Source : SGBC , GSI

The Solution

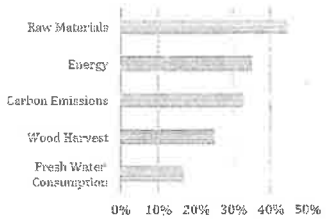
No other sector of industry uses more materials and energy, produces more waste and contributes less to material recycling than the building industry (Hegger et al. 2008)

Electricity Consumption in Singapore (2005)



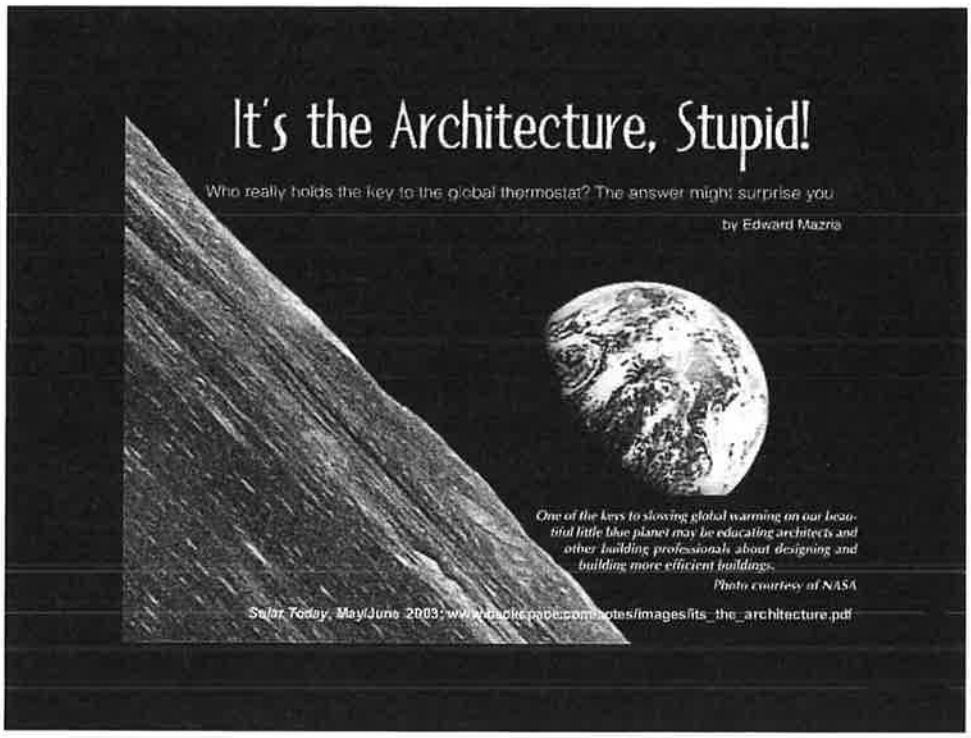
Global Building Consumption

(Kibert, 2016)



The built environment globally and locally consumes vast quantities of resources during their construction and over their life. With limited resources and the impacts of climate change being felt globally there needs to be collective action to mitigate this.

Source : SGBC , GSI



The Solution

What is Sustainable Development?

Bruntland Report

- *Development that meets the needs of the present without compromising the ability of future generations to meet their own needs*

(World Commission on Environment and Development 1987).

Caring for the Earth

- *Improving the quality of life while living within the carrying capacity of supporting ecosystems*

(IUCN/UNDP/WWF 1991)

The Solution

What is Sustainable Development?

1. How we live well with the environment:

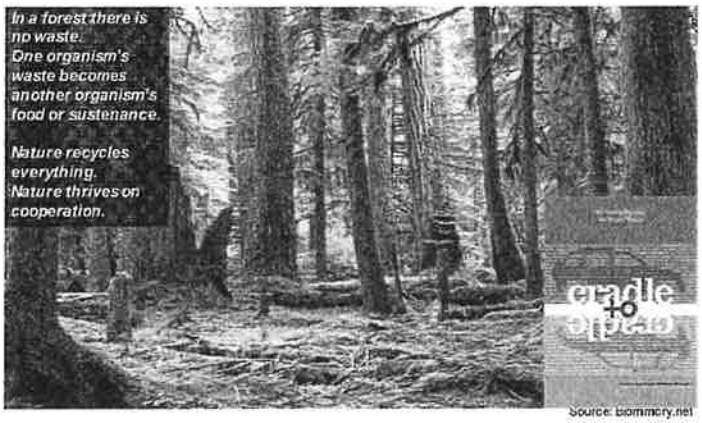
- *Responsible use of the worlds resources*
- *Working with the climate (do we need air conditioning, what's wrong with being warm???)*
- *Responsible material consumption.*

2. How we live well with each other

- *Sharing spaces and places*
- *Community initiatives and tolerance*
- *Working together*

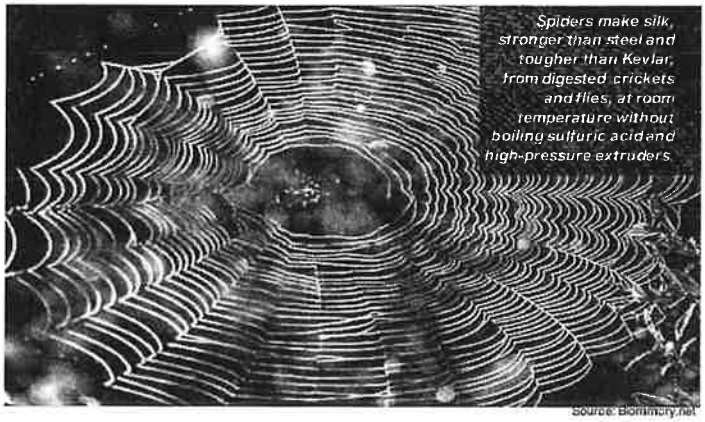
Learning from Mother Earth

Recycling & photosynthesis



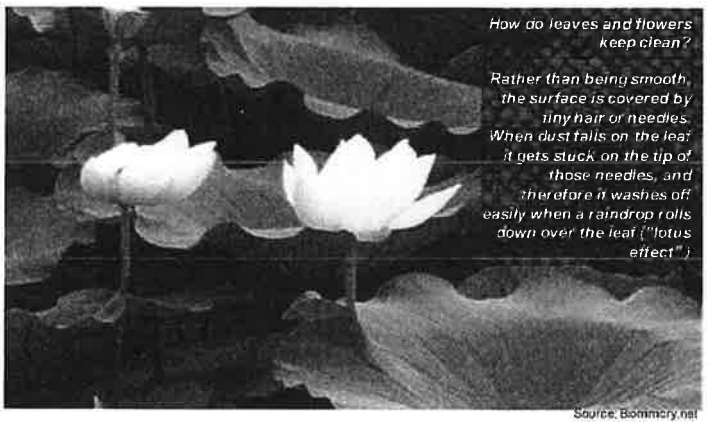
Learning from Mother Earth

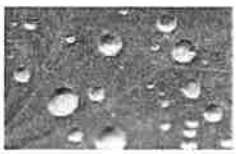
Harmful emissions



Learning from Mother Earth

Water Efficiency





https://www.videezy.com

Lotusan self cleaning paint



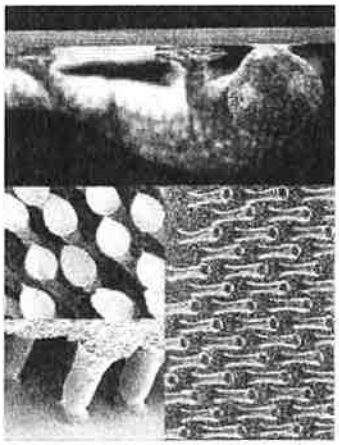
Source : pezi.o

Learning from Mother Earth

Gecko Tape



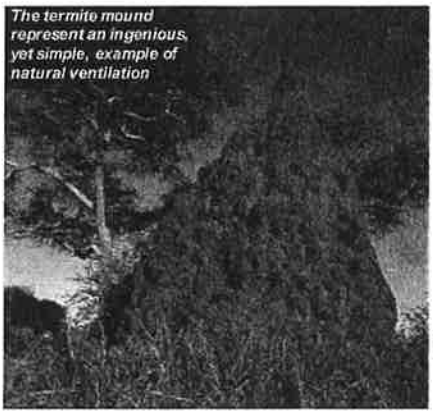


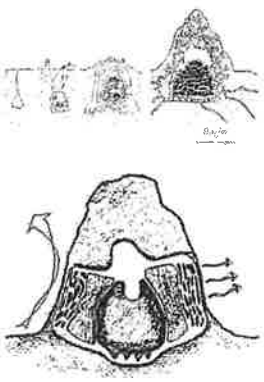


www.corvallisadvocate.com

Learning from Mother Earth

Natural ventilation





Source: Biomimicry.net

Architecture is the Key .....

Projects that heal the earth & us

Omega Institute (New York)

Water Reclamation Capacity

52,000

Maximum Design Flow

38,000

Monitored Maximum Flow

gallons per day (GPD)

Building Sq. Ft.

6,250

Site Acreage

4.5

Electricity Demand

132.77

Kwh/sf (average)

Electricity Usage

-1.43

Net Zero Energy – the building is designed to generate more electricity than it uses

Rainwater Use for Toilet Flushing

40

gallons: Average Daily Demand

1,800

gallon cistern stores enough water for 45 days

Embodied CO2

-1,387

Generation Capacity (Electricity)

2,830

sq.ft of photovoltaic panels, 211 panels in 2 arrays

134.2

Energy (85 W/sf peak DC output)

Construction Waste Recycling and Diversion from landfill

99% 99% 99% 99% 100% 100%

of wood waste was re-routed for reuse or recycling instead of landfill

of glass, metal, plastic, and mixed paper/packaging waste was recycled







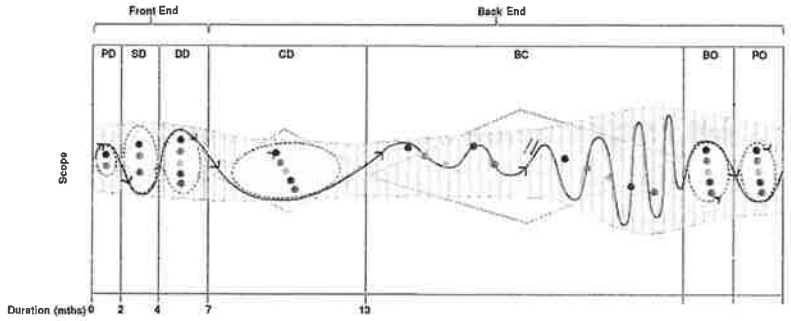
Source : https://eomega.org

快速增长中。。。
we might need to pause a little...

exploring a agenda beyond architecture

Does the process matter?
What else can Architects and Engineers do to make a difference

The Solution

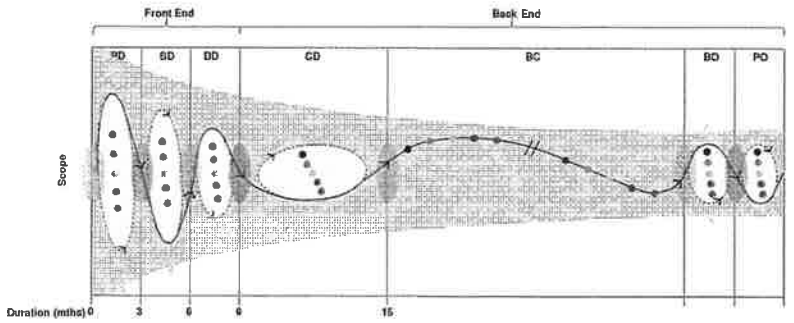


◇	Project Constraints	PD	Pre-design	BC	Bidding, Construction, Commissioning
	Exploratory Design Process	SD	Schematic Design	BO	Building Operation (start up)
	All Team Workshop	DD	Design Development	PO	Post-Occupancy (long term)
	Focused Team Workshops (water, energy, materials, etc)	CD	Construction Documents		
	Iterative Process				
	Additional Iterations as necessary				

Source Fryen Lim & Owen Wee , Surbana IGBC 2012

Passive design & natural ventilation 2017

The Solution



◇	Project Constraints	PD	Pre-design	BC	Bidding, Construction, Commissioning
	Exploratory Design Process	SD	Schematic Design	BO	Building Operation (start up)
	All Team Workshop	DD	Design Development	PO	Post-Occupancy (long term)
	Focused Team Workshops (water, energy, materials, etc)	CD	Construction Documents		
	Iterative Process				
	Additional Iterations as necessary				

Source Fryen Lim & Owen Wee , Surbana IGBC 2012

exploring a agenda beyond architecture

•Carbon Management	碳处理
•Indoor Occupant well being	室内环境与居住者的康乐
•Pollution (including heat)	环境污染（包括热气污染）
•Land Rejuvenation	土地还原
•Community Building	社区建筑
•Water management	治水
•Resource Management	资源管理
•Quality Management	质量管理
•Whole Life cycle	建筑生命周期
•Etc etc	

Service Provided : Faster , Cheaper , better

You can only choose 2

Faster and Cheaper , but will not be good

Faster and Better , but will not be Cheap

Cheap and Better , but will not be fast

X

END

Al Gore

Part 2 : SUSTAINABLE DEVELOPMENT (ARCHITECTURE)

Why sustainable development ?

Sustainable Framework

Climate and adaption

Thermal comfort

Sustainable Design

"Sustainable development is development that meets the needs of the present without compromising the ability of future generations to meet their own needs.

Brundtland Report , 1987

Sustainable Design

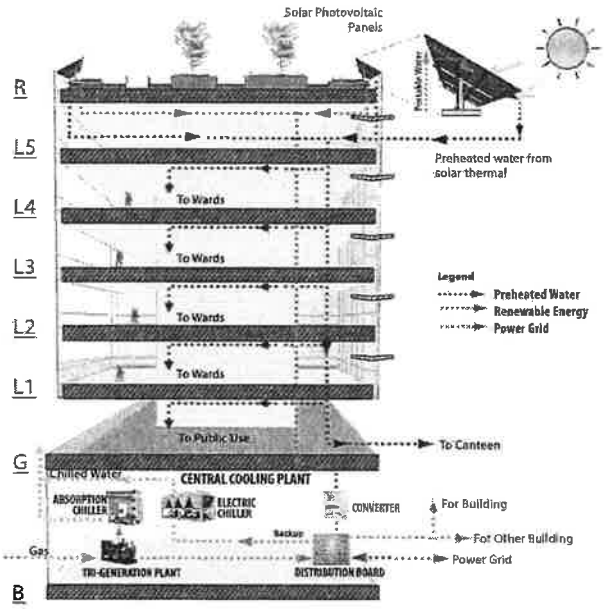


The Three trigger issues of Sustainability

Designing Sustainably

"Meeting thet needs of the present without compromising the ability of future generations to meet their own needs"

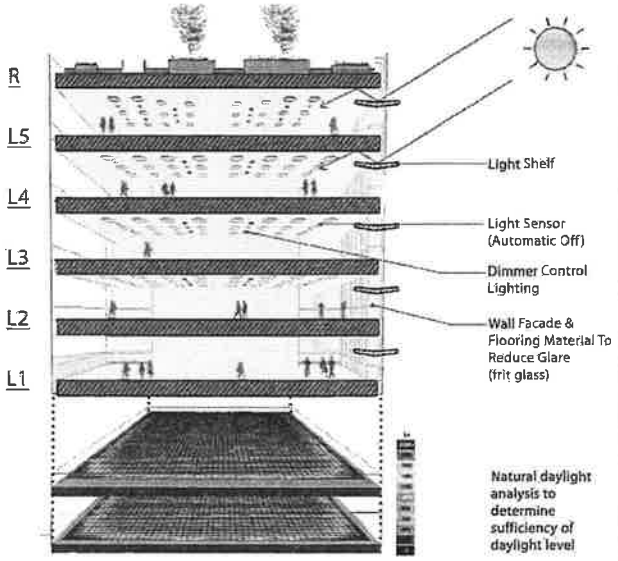
Passive Design & Natural Ventilation, GMP 2016



Sustainable Energy

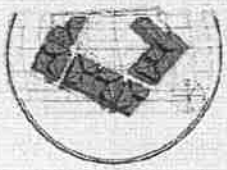
A modern building while requiring a **stable source of energy** will have to look at other sources of energy to reduce the reliance on fossil fuels. Energy output of fossil fuels will have to combine effective technology to ensure highest efficiency.

Passive Design & Natural Ventilation, GMP 2016

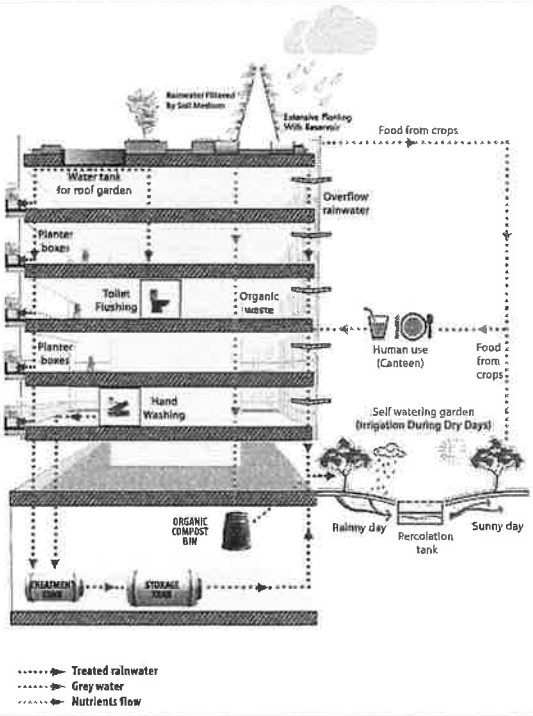


Maximizing Daylight

The key to daylight design is to bring in usable light while keep glare and heat away. Natural light improve human productivity and concentration of occupants as well as overall well being of occupants. Integrated Façade design with light shelves and shading devices together will pre-construction daylight analysis will allow a high percentage of daylight to all areas.

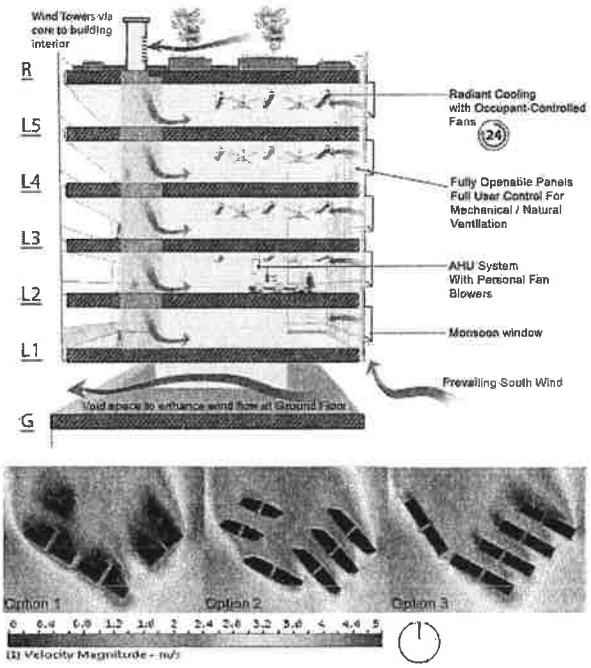


Passive Design & Natural Ventilation, GMP 2016



Sustainable Eco-system

The design aims to enhance and be part of the eco system within this area and aims to reduce, recycle and reuse resources. It also hopes to reproduce as well as rejuvenate by replacing non-producing ornamental plants to crops. The design of this eco-system includes opportunities to bring in the community (ie residents, schools) to learn about nature and the positive effect of biophilic design



Better Thermal Comfort

Acceptable Thermal Comfort is the combination of temperature, humidity and speed of ventilation. It is also the ability of the occupants to control these factors that makes final the environment comfortable. As such, operability and control is important for a climatic like Singapore with monsoon seasons and occasional haze.

What does passive design cost?

- Many passive design features cost nothing – they're about more effective use features that are common in buildings. Other passive design features do add initial cost but provide significant benefits to the owner but making the buildings warmer/cooler, healthier and drier, and by reducing ongoing costs such as energy bills .....

No cost or low extra cost features include:

- orientation to minimize heat radiation and provide daylighting
- building shade – compact building plans have less external wall area and therefore less potential heat gain, thin building are better for daylighting and cross-flow passive ventilation
- harnessing of natural ventilation – to cool internal spaces and remove contaminants
- carefully considered placement and seizes of windows and opening sashes
- proper zoning of rooms and occupant activities to take account of the sun's movement, daylighting and potential sources of noise
- using the thermal mass available in the building's structure – in roof, concrete floor slabs and walls – to moderate temperatures and provide cooling
- using correctly sized roof overhangs and other forms of shading to cut out unwanted light and heat
- good thermal insulation and choice of materials

What does passive design cost?

It is expensive **NOT** to go green !!!

Asset protection
Reduce operating costs
Improve productivity and reduce health costs

What does passive design cost?

Asset protection

three-year study of some 1,300 properties, which Compared similar properties within a quarter-mile (400 meter) radius, LEED-certified buildings had 4.1% greater occupancy and $11.33 per sq.ft. (about 30%) higher rent and sold about 30% higher prices.

Source: -CoStar:-Commercial Real Estate Information in the U.S. and U.K. 2008

Table 3: Impact of Green Retrofitting on Valuation of Retail Properties

s/n	Gross Annual Rent	Operating Expenses (OE)	Property Tax	Yearly Cost Saving	% of Saving on OE	Net Income	Capital Value (CV)	Retrofit Cost	Cost as % of Capital Value	% increase in Capital Value
Ave	41,691,605	6,253,741	4,169,161	537,821	8.6%	31,606,524	636,130,480	2,638,567	0.40%	1.77%

Table 4: Impact of Green Retrofitting on Valuation of Office Properties

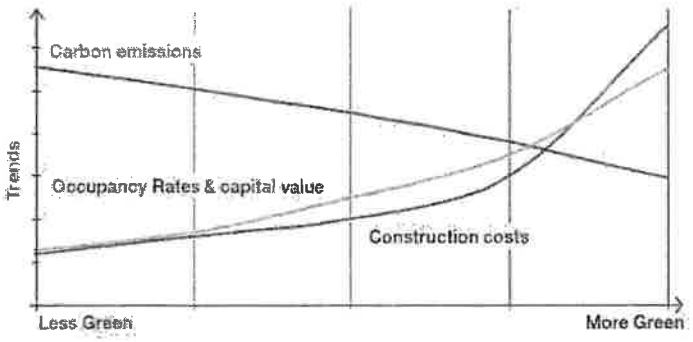
s/n	Gross Annual Rent	Operating Expenses (OE)	Property Tax	Yearly Cost Saving	% of Saving on OE	Net Income	Capital Value (CV)	Retrofit Cost	Cost as % of Capital Value	% increase in Capital Value
Ave	25,839,024	4,025,853	2,583,902	535,402	13.3%	20,554,671	413,293,420	3,197,673	0.77%	2.66%

Source: -BCA/NUS :- Valuation of Green Commercial Properties, Singapore ,2011

Passive Design & Natural Ventilation, GMP 2017

What does passive design cost?

Asset protection



Source: www.davislangdon.com
The cost & benefit of achieving Green buildings

What does passive design cost?

Reduce operating cost

Oil reached $140 a barrel

Source: China financial times , 2008/9/16

Indonesia rises petrol prices 44%

22/6/2013

Malaysia rises energy prices 15%

2010 – 2013

Reduce operating cost

In New South Wales, prices have climbed by 127 per cent or an average price rise of 10 per cent. Many consumers were hit with a double-digit percentage increase from July 1.

In Victoria, prices have risen by 120 per cent, with another likely from January 2017.

In Queensland prices have climbed by 151 per cent including an average increase of four per cent from last month.

In South Australia, prices have risen by 107 per cent. But South Australians endured the steepest rise in the country on July 1. And there are indications it could get much worse.

13 Aug 2016
Source : news.com.au

What does passive design cost?

Improve productivity and reduce health costs

Productivity improves 5%-10%

Medical absences 30% - 40%

Source:-Jerry Yudelson,2009

Passive Design & Natural Ventilation, GMP 2017

What does passive design cost?

Protect Environment



Sustainable Design : a necessary framework

-A mandatory frame integrated with the economy , culture and social needs that is responsible to the environment

-Efficient implementation throughout the life time of the building

-Effective knowledge sharing and skill workforce

-Behavioral adaption

-**Climate adaption** coupled with cutting edge technological design

Source : Nottingham University : Dr Sergio A.

Climate Adaption : Climate

3 main things about the climate :-

Temperature & Humidity
Solar
Wind

Climate Adaption : Climate

- In the **Koppen-Geiger Climate Classification**, the following climate types are distinguished:
  - **A** for Tropical Humid Climates.
  - **B** for Hot Dry Climates.
  - **C** for Mild Mid-Latitude Climates.
  - **D** for Cold Mid-latitude Climates.
  - **E** for Polar Climates.
  - **H** for Highland Climates.
- Based on **monthly temperature variations**, each major classification can have the following sub-types:
- **a** - warmest month >/=to 22°C (for C or D climates)
- **b** - warmest month < 22°C (for C or D climates)
- **c** - less than four months > 10°C (for C or D climates)
- **d** - same as 'c' but coldest month < -37°C (for D climates)
- **h** - hot and dry: all months > 0°C (for B climates)
- **k** - cool and dry: at least one month < 0°C (for B climates)



Source: http://squ1.org

Climate Adaption : Climate

Warm humid Climates :-

- Warm-humid climates cover the **largest area of Earth** (20% of land surface and 43% of ocean surface) and are typical of the **equatorial belt** within the tropical climate zone (Amazon Basin in Brazil, the Congo Basin in West Africa and Indonesia);
- There is **abundant rainfall** due to the active vertical uplift or convection of air that takes place here, and during certain periods, **thunderstorms** can occur **every day**;
- Because a substantial part of the Sun's **heat is used up in evaporation** and rain formation, temperatures rarely exceed 35°C; a daytime maximum of 32°C is more common;
- At night the **abundant cloud cover restricts heat loss**, and minimum temperatures fall no lower than about 22°C;
- **Seasons** are defined by **variation** of **rainfall** and **cloudiness**;
- Greatest rainfall occurs when the Sun at midday is overhead. **On the equator** this occurs twice a year in March and September, and consequently there are **two wet** and **two dry seasons**;
- **Away from the equator**, the two rainy seasons merge into one, and the climate becomes monsoonal (**a wet and a dry season**).



Source: http://www.ace.mmu.ac.uk

Climate Adaption : Climate

Warm humid Climates

- It is possible to **influence the wind**, but not the solar incidence, so **solar orientation** should be a dominant aspect in the design;
- **North** and **south walls** can have **large openings**;
- **Rooms** could be arranged **in one row** in order to allow both **inlet** and **outlet openings** for each room;
- **Wing walls** (or vegetation) may be used to create a positive (or negative) **pressure zones** and facilitate cross ventilation, driven by pressure differential other than simply by wind incidence;
- With **urban developments** and increasing densities in built-up areas, however, **ventilation effect may disappear**;
- The solution may be represented by a low-power, low-velocity slow moving **ceiling fan**, which can generate the required air movement velocity for physiological cooling;
- In such a dense situation (if other houses are also elevated), the benefit of the elevated house may also disappear, so a **concrete slab-on-ground floor** may (eventually) provide some **heat sink**;
- In **daytime rooms heavy construction** may be **acceptable**, but in **bedrooms** a **lightweight** construction should be preferred.

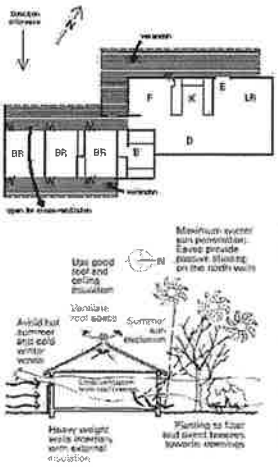




Source: Szokolay, 2004

Climate Adaption : Climate

Warm humid Climates :-





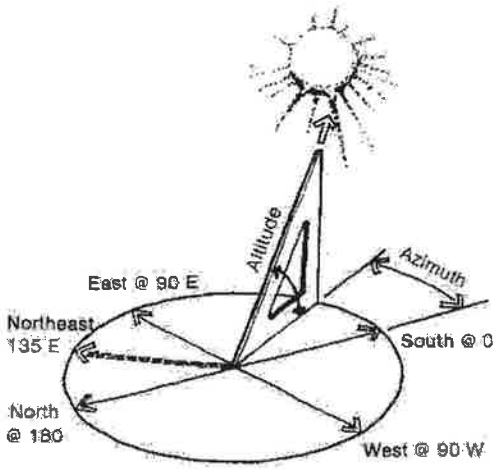
Climate Adaption : Climate





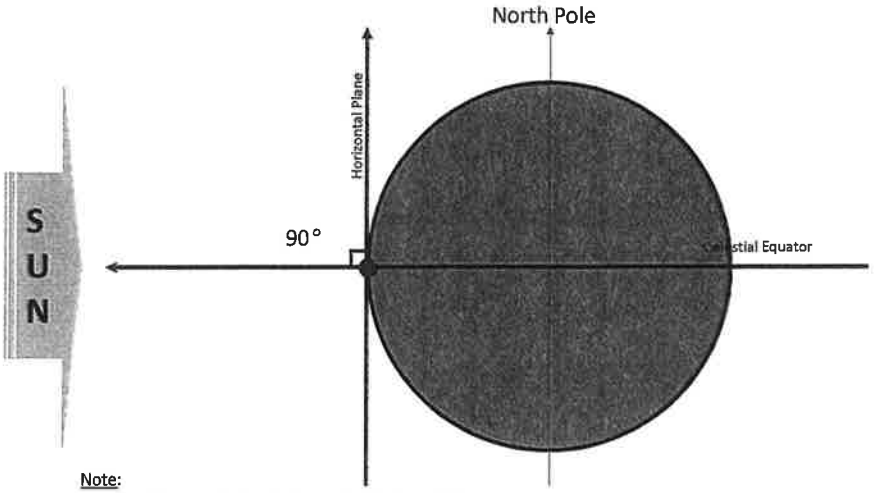
Passive people live in active houses , active people live in passive houses

Climate Adaption : Solar



Pic Source : Ben Lim University of Nottingham, 2009

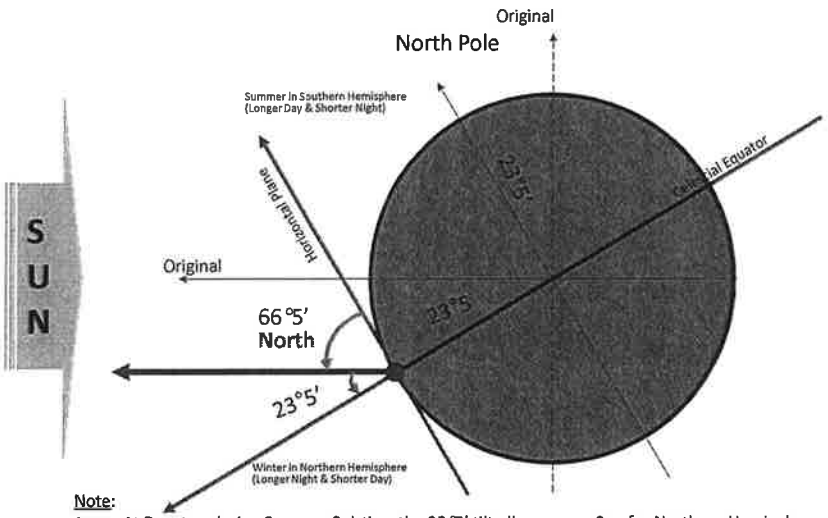
Climate Adaption : Solar



Note:
1. At Equator, during Equinox, the Sun is at 90° on top at noon time.
2. The Sun can be said to be 90° from either facing North of South (no difference in direction).

Equator During Equinox (Noon Time)

Climate Adaption : Solar

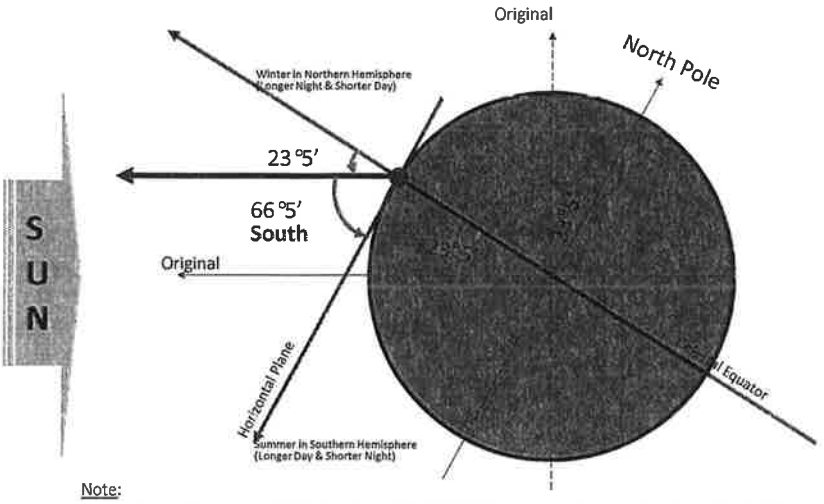


Note:
1. At Equator, during Summer Solstice, the 23°5' tilt allows **more** Sun for Northern Hemisphere.
2. The same person on the Equator will now see the Sun at 66°5' **N** from the Horizontal Plane.

Equator During Summer Solstice (Noon Time)

Passive Design & Natural Ventilation, GMP 2017

Climate Adaption : Solar



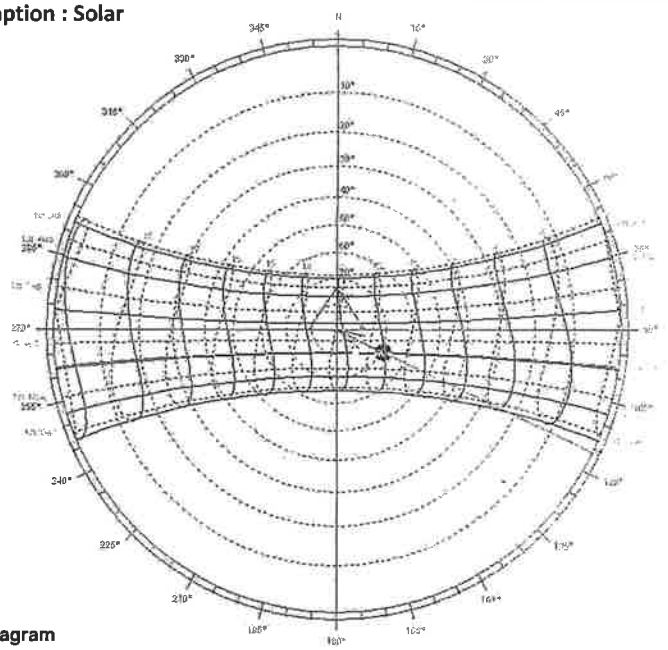
Note:

1. At Equator, during Winter Solstice, the 23°5' tilt allows less Sun for Northern Hemisphere.
2. The same person on the Equator will now see the Sun at 66°5' S from the Horizontal Plane (which is opposite direction, ie. facing South)

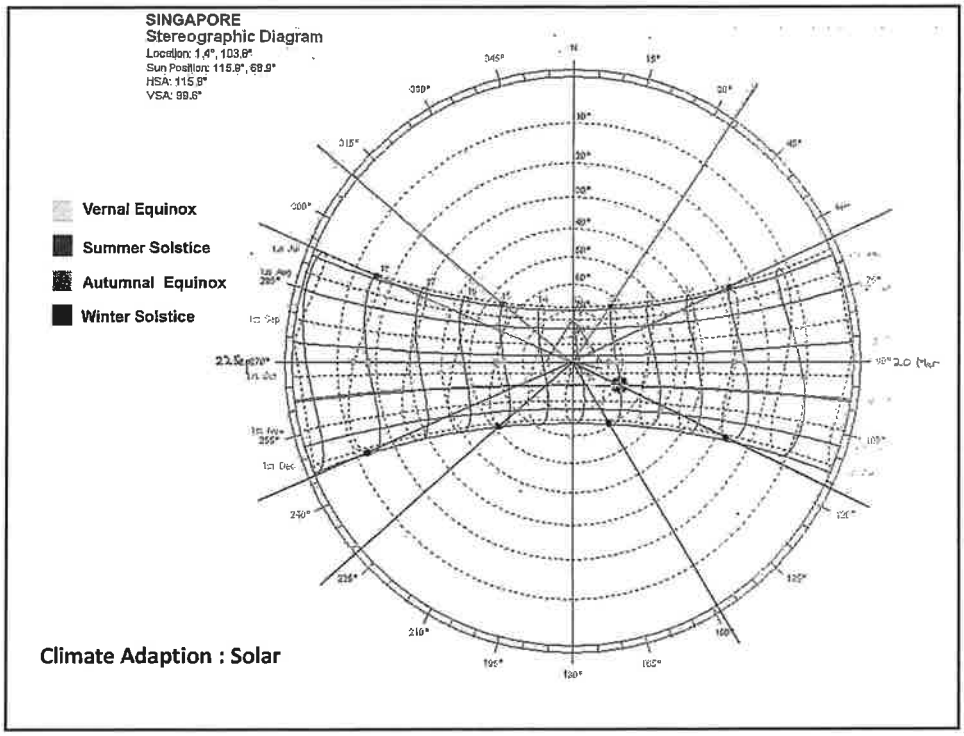
Equator During Winter Solstice (Noon Time)

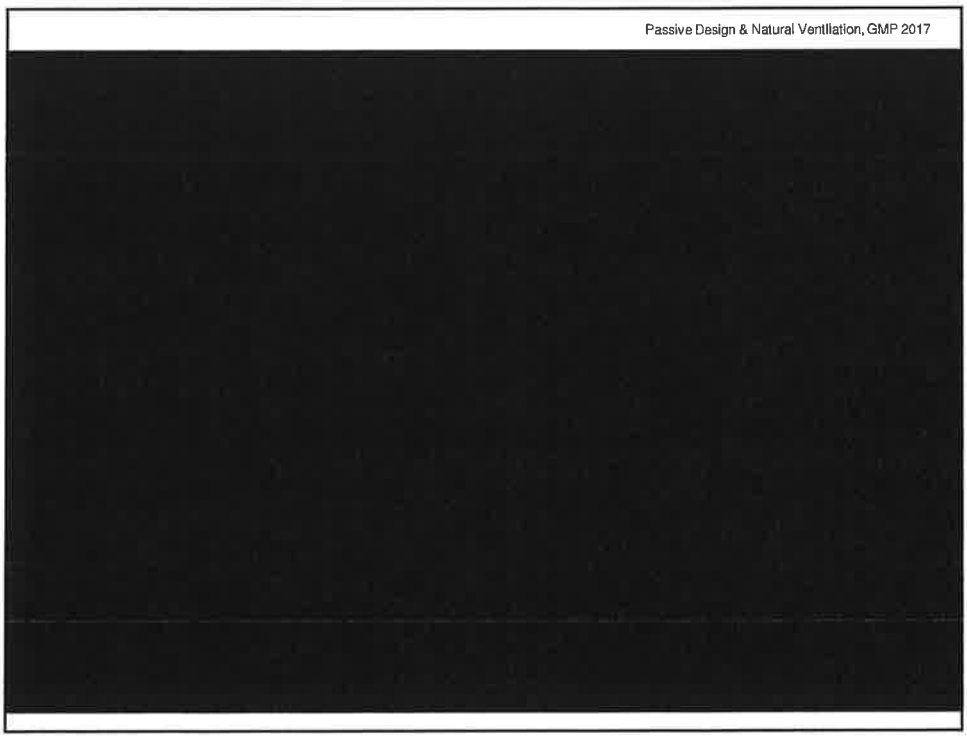
Passive Design & Natural Ventilation, GMP 2017

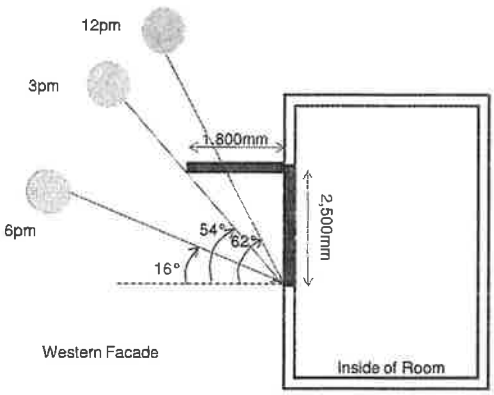
Climate Adaption : Solar

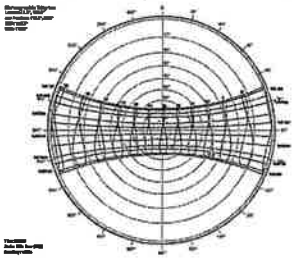


Stereographic Diagram


SINGAPORE
Stereographic Diagram
Location: 1.4°, 103.8°
Sun Position: 115.8°, 68.9°
HSA: 115.8°
VSA: 68.6°
Vernal Equinox
Summer Solstice
Autumnal Equinox
Winter Solstice
Climate Adaption : Solar


Passive Design & Natural Ventilation, GMP 2017





Sun's altitude in Singapore changes very slighting through the year. Taking and example of June Solstice, a sunshade of 1,800mm can shade the sun up to only 3pm.

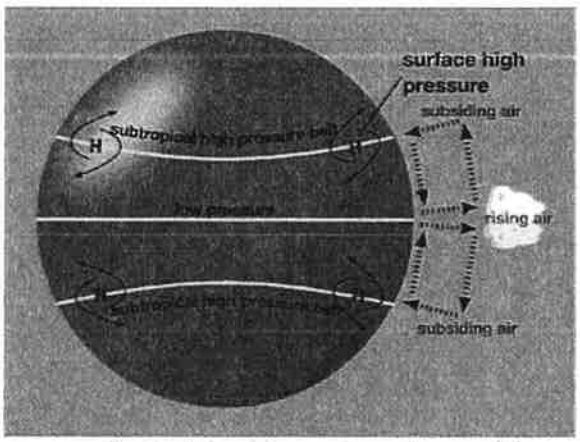
Horizontal shading is impractical on the western façade in Singapore.

Movable horizontal shading are more useful to keep the sun out as it can address the sun angle as it changes from Dec – June.

Shading angle for June Solstice

Time	Shading angle	Width of sunshade
12pm	62°	1,300mm
3pm	54°	1,800mm
6pm	16°	8,710mm

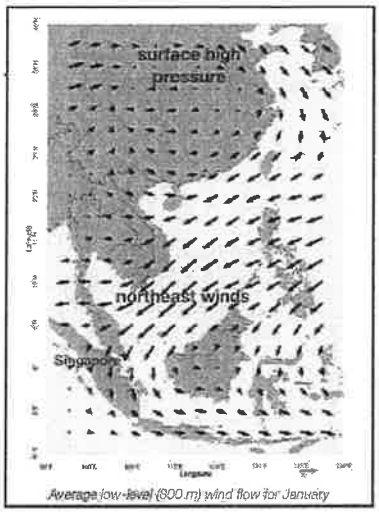
Climate Adaption : Wind



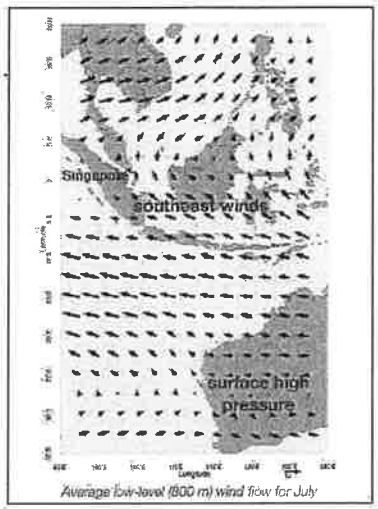
Average vertical circulations (Hadley cells) are shown on both sides of the equator. The downward arms are associated with the subtropical high pressure belts and fine weather.

Source : NEA , Weatherwise , 2009

Climate Adaption : Singapore Climate



Average low-level (900 m) wind flow for January



Average low-level (900 m) wind flow for July

Source : NEA , Weatherwise , 2009

Rainfall :- Monsoon seasons

Climate Adaption : Singapore Climate

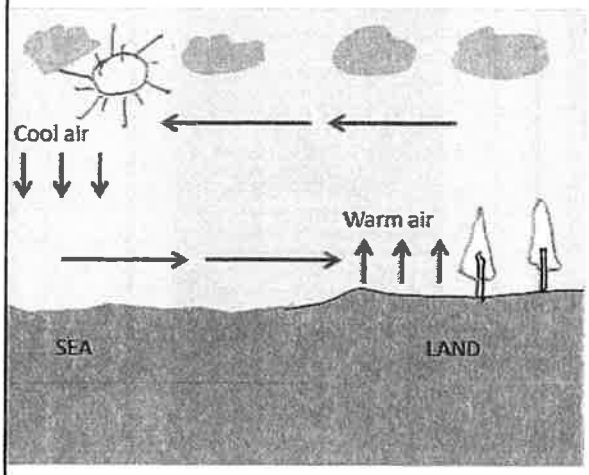
Rainfall

Month	Mean Total Raindays[a]	Occurrence of highest hourly mean[b] (hrs)	Monthly mean[c] (mm)
January	13.6	1500	255.8
February	8.1	1700	107.4
March	12.7	1500	171.0
April	14.0	1700	151.2
May	13.2	1300	163.9
June	12.7	1400	132.3
July	13.2	1300	150.0
August	13.7	1300	151.9
September	13.3	1400	157.3
October	14.7	1400	158.6
November	16.0	1500	262.4
December	16.5	1700	329.5

[a] Calculated by averaging the total number of raindays for each month for the 27 year period (1982 – 2008)
[b] Time of day for which the mean rainfall occurs for each month for the 27 year period (1982 – 2008) is the highest
[c] Calculated by averaging the monthly total rainfall for each month for the 27 year period (1982 – 2008)

Source : NEA , Weatherwise , 2009

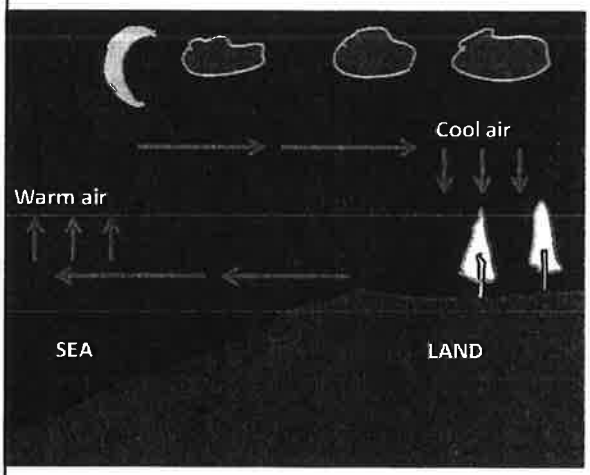
Climate Adaption : Singapore Climate



Sea wind (afternoon)

Regional wind movement due to temperature inertia of the sea and land

Climate Adaption : Singapore Climate



Land wind (night)

Regional wind movement due to temperature inertia of the sea and land

Climate Adaption : Singapore Climate

Surface Wind

Month	Direction	Mean speed (m/s)	Max gust speed (m/s)
January	N/NE	2.6	20.3
February	N/NE	2.8	17.8
March	N/NE	2.1	21.9
April	Variable	1.4	23.9
May	S/SE	1.5	18.1
June	S/SE	1.9	21.4
July	S/SE	2.3	23.9
August	S/SE	2.4	21.9
September	S/SE	1.9	21.4
October	Variable	1.4	20.3
November	Variable	1.3	21.1
December	N/NE	1.9	17.2

1 m/s = 1.94 knots = 3.6 km/h

Source : NEA , Weatherwise , 2009

Climate Adaption : Singapore Climate

Period	Prevailing Winds	Weather Features
Northeast Monsoon Season (December – early March)	Northerly to northeasterly winds 6 – 10 km/h	◆ Monsoon Surges cause widespread continuous moderate to heavy rain, at times with 25 – 35 km/h winds in the first half of the season. ◆ Rapid development of afternoon and early evening showers. ◆ Windy and relatively dry in the later part of the season.
Inter-monsoon Period (Late March – May)	Light and variable, interacting with land and sea breezes	◆ Thunderstorms, at times severe, occur in the afternoon and early evening. ◆ Hot afternoons are common (maximum temperature above 32°C).
Southwest Monsoon Season (June – September)	Southerly to southwesterly winds 6 – 10 km/h	◆ Occasional "Sumatra Squalls" with wind gusts of 40 - 80 km/h occuring between the predawn hours and midday. ◆ Short duration showers/thunderstorms in the afternoon are common.
Inter-monsoon Period (October – November)	Light and variable, interacting with land and sea breezes	◆ Thunderstorms, at times severe, occur in the afternoon and early evening. ◆ Generally wetter than the inter-monsoon Period earlier in the year.

Source : NEA , Weatherwise , 2009

Climate Adaption : Singapore Climate

Month	General Weather Features
October	October is the beginning of the second inter-monsoon Period of the year. It often marks the beginning of the rainy season which may last till January the following year. Monthly rainfall jumps more than 20 mm on the average from the September value. However, this is not Northeast Monsoon yet. Surface winds remain mostly light with afternoon thundery showers contributing to most of the rainfall. In some years, when *dry weather in the region persists* into October, episodes of hazy conditions may occur.
November	November is the second wettest month of the year. Winds are light initially and may turn temporarily to northwesterly or northeasterly. Heavy thunderstorms in the afternoon may extend into the evening. Though less common, monsoon rain may sometimes happen in November.
December	December is the wettest month of the year. Heavy thunderstorms occurring in the afternoon are common. In addition, surges of northeasterly monsoon winds may bring about windy conditions with widespread heavy rain. The rain may be continuous, lasting from a few days to more than a week.

Rainfall :- Monsoon seasons

Source : NEA , Weatherwise , 2009

Passive Design & Natural Ventilation, GMP 2017

Summary of Singapore climatic conditions

- Singapore's climate is general very humid and hot with monthly temperature range of 26° C to 28° C , climate condition remain quite constant through the year
- humidity of 80% to 88% (Climate Consultant, Singapore).
- The average wind speed is about 2m/s, however, during monsoon or rainy days gust wind speed can go up to 17m/s to 23 m/s (National Environment Agency, Singapore, 2009).
- The wind speed in the Central Business District, however, is very different due micro climate within the CBD area (Hien, 2009,pg 6).
- Wind comes from the North during Nov to March and South Wind during Jun to Sept.
- Singapore has high level of radiation/illumination and with high amount of cloud cover results in high level of diffuse radiation. (Climate Consultant, Singapore)
- Constant cooling and humidity control is required to bring about in door thermal comfort. It is classified as "A" Tropical Humid Climate, Hot, Monsoonal rain climate (Koppen-Geiger classification)

Thermal Comfort

Thermal Comfort : Factors of Thermal comfort in buildings

1) **Environmental Factors**

a) Dry bulb temperature

b) Mean Radiant Temperature

c) Relative Humidity

d) Air movement

2) **Physiological**

a) Metabolic Rate

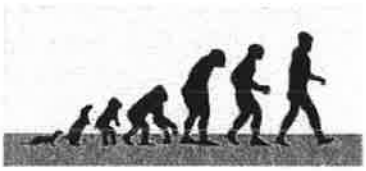
b) Clothing Level

Thermal comfort is very subjective and depends on personal preferences and acclimatisation

Thermal Comfort : Factors of Thermal comfort in buildings

1. Thermal Neutrality
2. Adaptive model
3. Predicted Mean Vote

Human are the most adaptive species



And the most demanding

1. Thermal Neutrality

neutrality, thermal: the indoor thermal index value corresponding with a mean vote of neutral on the thermal sensation scale.

Thermal Neutrality (***Tn***) refers to the air temperature at which, on average, a large sample of people would feel neither hot nor cold. This temperature is affected by both the average annual climate and seasonal fluctuations within it. Experiments have found that ***Tn*** correlates well with the outdoor average temperature

$T_n = 17.6 + 0.31 * T_{out}$

Source : Ashrae 55

Thermal Comfort : Factors of Thermal comfort in buildings

1. Find the **neutrality temperature** (Tn= 17.6 + 0.31 x $T_{o,av}$) for the **coldest** and/or **hottest** month (or every month);
2. Take the **comfort limits as ± 2.5°C**;
3. **Mark these on the 50%RH** curve to define the '**sides**' of the comfort zone as the **corresponding SET lines**;
4. The **humidity limits** (top and bottom) will be **12** and **4 g/kg** respectively (1.9 and 0.6 kPa vapour pressure);

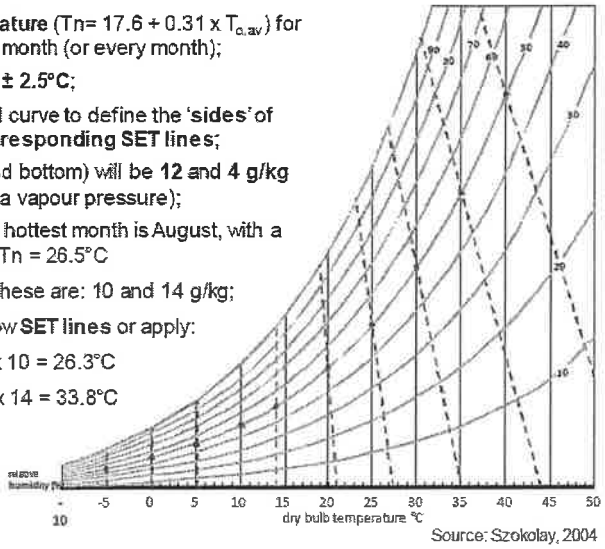
- In the case of **Singapore**, the hottest month is August, with a mean temperature of 28,5°C; Tn = 26.5°C

  T_L = 24°C; T_U = 29°C; AH for these are: 10 and 14 g/kg;
- For the **side boundaries** follow **SET lines** or apply:

  for T_L = 24 + 0.023 (24 – 14) x 10 = 26.3°C

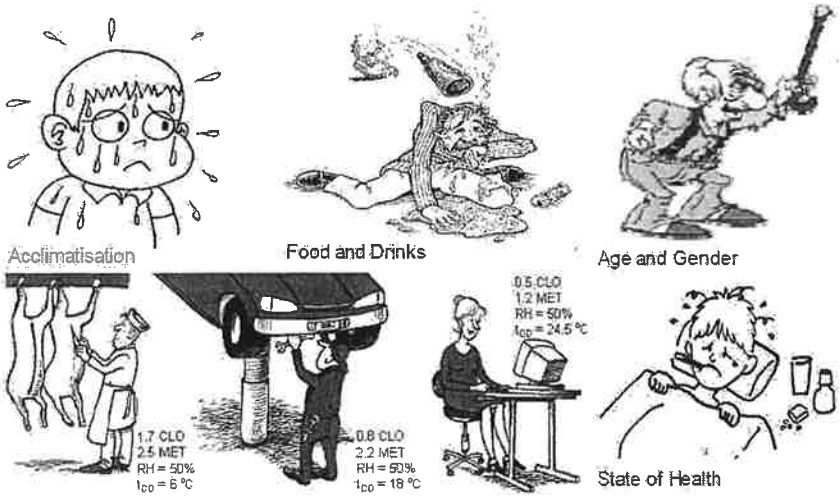
  for T_U = 29 + 0.023 (29 – 14) x 14 = 33.8°C

Source :Nottingham University , Sergio A



Source: Szokolay, 2004

Thermal Comfort : Factors of Thermal comfort in buildings



1. Adpative

adaptive model: a model that relates indoor design temperatures or acceptable temperature ranges to outdoor meteorological or climatological parameters.

Adaptive model for thermal comfort:

Adaptive comfort models supplement the human behavioral factors into the thermal comfort model. They assume that, if changes occur in the thermal environment to produce discomfort, then people will generally change their behavior and act in a way that will restore their comfort. Such actions could include putting on clothing, reducing activity levels or even opening a window.

The main effect of such models is to increase the range of conditions that designers can consider as comfortable, especially in naturally ventilated buildings where the occupants have a greater degree of control over their thermal environment.

Naturally ventilated building:

$T_c = 11.9 + 0.534 * T_{out}$

Heated or cooled building:

$T_c = 23.9 + 0.295(T_{out}-22)\ \exp([-(T_{out}-22)/33.941]^2)$

The above results hold true for hot and cold climatic conditions and are often used for design of buildings even in tropical climates. Some countries like India and China have developed their own adaptive comfort standards based on empirical studies conducted in their respective climate zones.

Source : Ashrae 55

Thermal Comfort : Factors of Thermal comfort in buildings

Table 1 - The effect of adaptive behaviors on optimum comfort temperatures. Taken from BRE

BEHAVIOR	EFFECT	OFFSET
Jumper/Jacket on or off	Changes Clo by ± 0.35	± 2.2K
Tight fit/Loose fit clothing	Changes Clo by ± 0.26	± 1.7K
Collar and tie on or off	Changes Clo by ± 0.13	± 0.8K
Office chair type	Changes Clo by ± 0.05	± 0.3K
Seated or walking around	Varies Met by ± 0.4	± 3.4K
Stress level	Varies Met by ± 0.3	± 2.6K
Vigour of activity	Varies Met by ± 0.1	± 0.9K
Different postures	Varies Met by ± 10%	± 0.9K
Consume cold drink	Varies Met by -0.12	+ 0.9K
Consume hot drink/food	Varies Met by +0.12	- 0.9K
Operate desk fan	Varies Vel by +2.0m/s	+ 2.8K
Operate ceiling fan	Varies Vel by +1.0m/s	+ 2.2K
Open window	Varies Vel by +0.5m/s	+ 1.1K

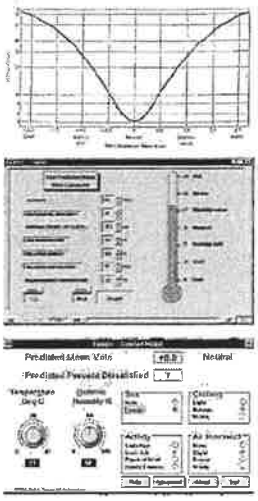
Adaptive Comfort

Adaptive comfort models add a little more human behaviour to the mix. They assume that, if changes occur in the thermal environment to produce discomfort, then people will generally change their behaviour and act in a way that will restore their comfort. Such actions could include taking off clothing, reducing activity levels or even opening a window. The main effect of such models is to increase the range of conditions that designers can consider as comfortable, especially in naturally ventilated buildings where the occupants have a greater degree of control over their thermal environment.

In order to consider adaptive comfort, the space must have operable windows, no mechanical cooling system, and the occupants must be near sedentary and have a metabolic rate between 1.0 and 1.3 met. Also, occupants have the option of adding or removing clothing to adapt to the thermal conditions.

Thermal Comfort : Factors of Thermal comfort in buildings

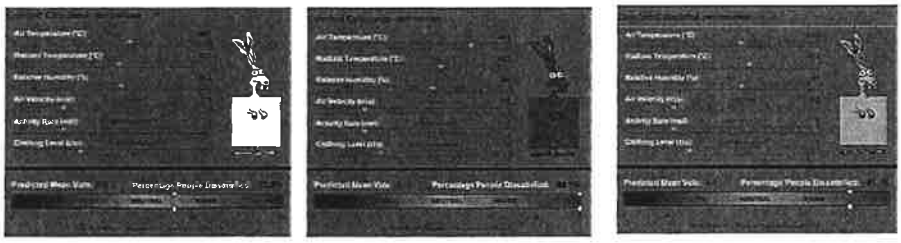
- The ***Predicted Mean Vote*** (PMV) refers to **a thermal scale** that runs from **Cold** (-3) to **Hot** (+3), originally developed by Fanger and later adopted by ISO (**Standard 7730**, ISO 1984);
- The original data was collected by subjecting a **large number of people** to different conditions within a climate chamber and having them select a position describing their comfort sensation;
- A **mathematical model** of the relationship between all the environmental and physiological factors considered has then been derived from the data;
- From the PMV, the ***Predicted Percentage of Dissatisfied*** (PPD) people can be determined;
- As PMV moves away from neutral (PMV = 0), PPD increases;
- The maximum number of people dissatisfied with their comfort conditions is **100%** and, as **you can never please all of the people** all of the time, the **minimum** number even in what would be considered perfectly comfortable conditions is **5%**;
- The **PMV equation** for thermal comfort is a **steady-state model** consisting of an empirical equation for predicting the average vote of a large number of people on a 7 point scale (-3 to +3).



Source :Nottingham University , Sergio A
Image : Http://squ1.org

Passive Design & Natural Ventilation, GMP 2017

Thermal Comfort : Factors of Thermal comfort in buildings

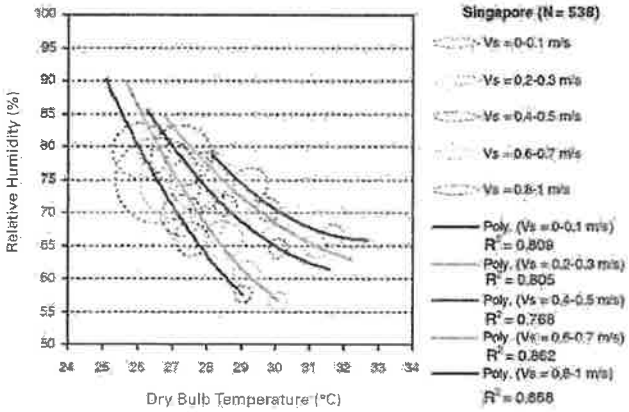


PMV = 0.6 PMV = 3 PMV = 2.1

The recommended acceptable PMV range for thermal comfort from ASHRAE 55 is between **-0.5 and +0.5** for an interior space

http://sustainabilityworkshop.autodesk.com/buildings/human-thermal-comfort

Thermal Comfort in Singapore



[Figure 1] Thermal Comfort Chart for Singapore.

Source: Givoni, B., J. Khedari, N H Wang and M. Noguchi. (2006). "Thermal sensation responses in hot, humid climates: effects of humidity". Building Research and Information, 34, no. 5 (Special on Indoor Environmental Quality).

Thermal Comfort in Singapore

Recommended air speed for achieving thermal comfort		
Wind for indoor air comfort	~ 1 m/s	Over occupant body sitting in doors
Wind for outdoor air comfort	~ 1.5 m/s	When in Shade, sitting or walking
Wind Gust	< 5 m/s	When walking

Recommended air speed for achieving thermal comfort under various air temperature	
Air Temperature °C	Recommended wind velocity (m/s)
28	**0.3**
29	**0.5**
31	**0.8**
32	**1.2**
33	**1.5**
34	**1.8**

Source : Department of Architecture, CUHK. (2006) " final report for feasibility study for establishment of air ventilation assessment system

THE END

Part 3 : Principles of Natural Ventilation

Measuring Wind
Factors affecting wind movement

Factors affect Natural ventilation



Air-conditioning liberated Design

Factors affect Natural ventilation

High density Cities







-Hot and humid

-Little air movement

-activities at the Pedestrian level

-Cars exhaust

-Dark/hot Materials

Factors affect Natural ventilation



Aircon turns neighbour's home into an oven

By ALVIN CHIANG

TECHNICIAN Patrick Tan has every reason to be hot and bothered when he is at home these days.

An education centre below his second-floor flat at Toa Payoh North has installed four hi[gh] air-conditioning temperatures outside two of his bedroom windows. And the units, which emit hot air, turn his flat into an oven.

The temperature inside it hovers between 32 and 33 deg C night and day.

The maximum outdoor temperature recorded by the Meteorological Service for the past two days has been 34 deg C.

The 42-year-old said his 74-year-old mother, who died last year, used to sleep in the living room every night to escape the unbearable heat.

"Besides my father's bedroom, the toilet and living room are also affected."

Closing the bedroom windows to keep out the heat is the least of his evils, he said, as this makes the room stuffy. According to Mr Tan, the problem began about three years ago, after an upgrading project. The compressors were then put right under the bedroom of his 83-year-old father, Mr Tan Chiaw Beng.

He said the family cannot afford to relocate from the home they have had for more than 30 years, or to install air-conditioning given the current gloomy economic outlook.

The Straits Times spoke to two of Mr Tan's neighbours on his floor and both said they do not have such a problem, despite having compressors outside their bedrooms too.

Last month, a Housing Board official checked on the situation in response to Mr Tan's complaint.

When contacted yesterday, an HDB spokesman said the board is aware of the situation and has told the Bedok-Tua Payoh North Education Centre to relocate the compressors. She said the HDB would "monitor the situation closely".

Call NEWSLINK 737-9700

Next to emitting by these four compressors up to Mr Tan Chiaw Beng's second-floor bedroom.



Source : Straits times

Factors affect Natural ventilation

High density Cities







-Hot and humid

-Little air movement

-activities at the Pedestrian level

-Cars exhaust

-Dark/hot Materials

Factors affect Natural ventilation

High density Cities – Urban structure



Tight sites , narrow streets and alleys

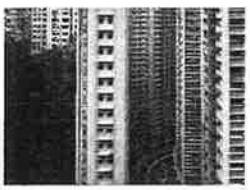
Lots of resistant to wind flow ,



Source: www.Opengreenmap.org

Ratio of street width and building height is small

streets are like valley that trap heat and pollutants



www.featureshool.com

Control of building height result in all building in the same zone is of the same general height

-Little air flow at the pedestrian level

Factors affect Natural ventilation

High density Cities – Urban structure

 www.shutterstock.com	Building with minimum setback	This forms small, tall and long alleys resulting in less than favorable wind conditions
 Skyscraper dictionary	Large podium and non-permeable (very solid)buildings	Poor wind movement into the building and pedestrian levels
 Reddit.com	Rows of building	Blockage of the wind for rows that are behind.
	Horizontal Signage	Obstruct wind

Source : www.Wanderloveworld.com

Principles of Natural Ventilation : Why Natural ventilate

-Reduce energy usage / carbon emissions

-Reduced capital, maintenance and running costs

-Mitigates sick building syndrome , reducing concentration of pollutants in spaces

-Improve comfort (wider range of acceptable temperatures) for users

-Improve productivity

-Improve user control

-Reduces plant space requirements and improve GFA efficiency

Accelerating passive building mass cooling

Principles of Natural Ventilation

-For air to move in and out of buildings , there must be a pressure different in the 2 spaces.

-The resistance to flow of air through the building will affect the actual air flow (speed & volume)

-Generally , Natural ventilation in buildings are caused by pressure difference and infiltration

Pressure Difference is caused by

1) Wind driven
2) Difference in air density (hot and cold air exchanges)
3) Combination of the 2 above.

Factors influencing wind around buildings

-Location

-Topography

-Roughness of site

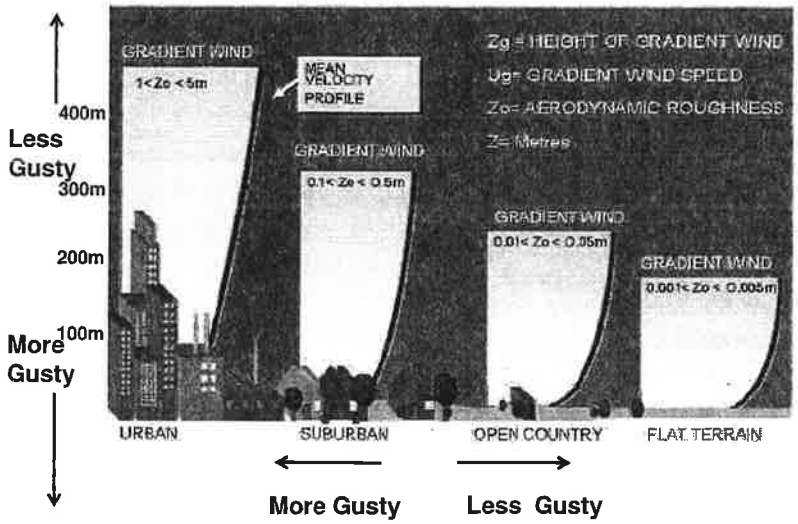
-Building Density and Configuration

-Building Height and Form

-Orientation

Source : philip oldfield University of Nottingham, 2009

Wind and its surrounding

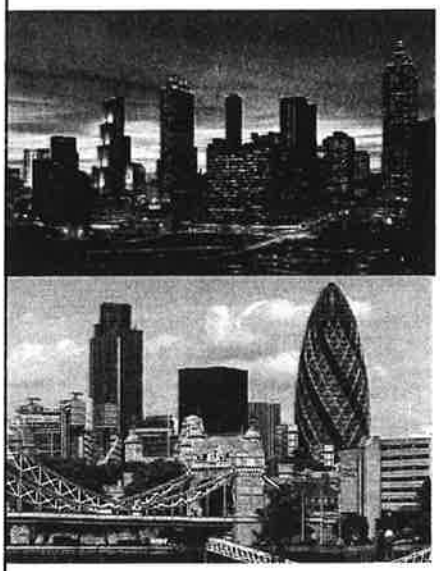


Measuring Wind - Beaufort Scale

Force	Wind (Knots)	WMO Classification	Appearance of Wind Effects	
			On the Water	On Land
0	< 1	Calm	Sea surface smooth and mirror-like	Calm, smoke rises vertically
1	1-3	Light Air	Scaly ripples, no foam crests	Smoke drift indicates wind direction, still wind vanes
2	4-6	Light Breeze	Small wavelets, crests glassy, no breaking	Wind felt on face, leaves rustle, vanes begin to move
3	7-10	Gentle Breeze	Large wavelets, crests begin to break, scattered whitecaps	Leaves and small twigs constantly moving, light flags extended
4	11-16	Moderate Breeze	Small waves 1-4 ft. becoming longer, numerous whitecaps	Dust, leaves, and loose paper lifted, small tree branches move
5	17-21	Fresh Breeze	Moderate waves 4-8 ft taking longer form, many whitecaps, some spray	Small trees in leaf begin to sway
6	22-27	Strong Breeze	Larger waves 8-13 ft, whitecaps common, more spray	Larger tree branches moving, whistling in wires
7	28-33	Near Gale	Sea heaps up, waves 13-19 ft, white foam streaks off breakers	Whole trees moving, resistance felt walking against wind
8	34-40	Gale	Moderately high (18-25 ft) waves of greater length, edges of crests begin to break into spindrift, foam blown in streaks	Twigs breaking off trees, generally impedes progress
9	41-47	Strong Gale	High waves (23-32 ft), sea begins to roll, dense streaks of foam, spray may reduce visibility	Slight structural damage occurs, slate blows off roofs
10	48-55	Storm	Very high waves (29-41 ft) with overhanging crests, sea white with densely blown foam, heavy rolling, lowered visibility	Seldom experienced on land, trees broken or uprooted, "considerable structural damage"
11	56-63	Violent Storm	Exceptionally high (37-52 ft) waves, foam patches cover sea, visibility more reduced	
12	64+	Hurricane	Air filled with foam, waves over 45 ft, sea completely white with driving spray, visibility greatly reduced	



Developed in 1805 by Sir Francis Beaufort, U.K. Royal Navy



Source
www.wallconvert.com
www.standard.co.uk

Roughness and Wind Shear

High above ground level, at a height of about 1 kilometre, the wind is hardly influenced by the surface of the earth at all. In the lower layers of the atmosphere, however, wind speeds are affected by the friction against the surface of the earth. In the wind industry one distinguishes between the roughness of the terrain, the influence from obstacles , and the influence from the terrain contours, which is also called the orography of the area. We shall be dealing with orography, when we investigate so called speed up effects, i.e. tunnel effects and hill effects , later.

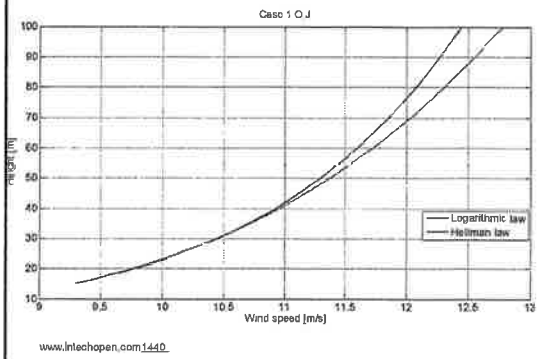


http://www.tour-beijing.com/china_silk_road_tours/bayanbulak_travel.php

Roughness

In general, the more pronounced the roughness of the earth's surface, the more the wind will be slowed down.
Forests and large cities obviously slow the wind down considerably, while concrete runways in airports will only slow the wind down a little. Water surfaces are even smoother than concrete runways, and will have even less influence on the wind, while long grass and shrubs and bushes will slow the wind down considerably.

Roughness class



In the wind estimation , we usually refer to roughness classes or roughness lengths. A high roughness class of 3 to 4 refers to a landscape with many trees and buildings, while a sea surface is in roughness class of "0"

Concrete runways in airports and flat open landscapes are in the roughness class of "0.5".

The roughness length is the distance above the ground level where the wind speed is in theory zero.

Roughness Class and Roughness Length Table

Roughness Class RC	Roughness Length, Z_0 [m]	Energy Index [percent]	Landscape
0	0.0002	100	Water surface.
0.5	0.0024	73	Completely open terrain with a smooth surface, such as concrete runways in airports, mowed grass.
1	0.03	52	Open agricultural area without fences and hedgerows and very scattered buildings. Only softly rounded hills
1.5	0.055	45	Agricultural land with some houses and 8 meter tall sheltering hedgerows within a distance of about 1,250 meters.
2	0.1	39	Agricultural land with some houses and 8 meter tall sheltering hedgerows within a distance of about 500 meters.
2.5	0.2	31	Agricultural land with many houses, shrubs and plants, or 8 meters tall sheltering hedgerows within a distance of about 250 meters.
3	0.4	24	Villages, small towns, agricultural land with many or tall sheltering hedgerows, forests and very rough and uneven terrain.
3.5	0.8	18	Larger cities with tall buildings.
4	1.6	13	Very large cities with tall buildings and sky scrapers.

Wind shear formula

V = Vref ln (z/z) / ln (Zref / Zo)

V	= Wind speed at height z above ground level
Vref	= reference speed (a known speed at a certain height Zref)
ln	= Natural logarithm function
Z	= height above ground level for the desired velocity , v
Zo	= roughness length in the current wind direction (found in the reference manual table)
Zref	= reference height, ie the height where we know the exact wind speed Vref

Assume we know that wind is blowing at 7.7 m/s at 20m height .
We wish to know the wind at 60 m. If roughness length is 0.1m then V ref = 7.7

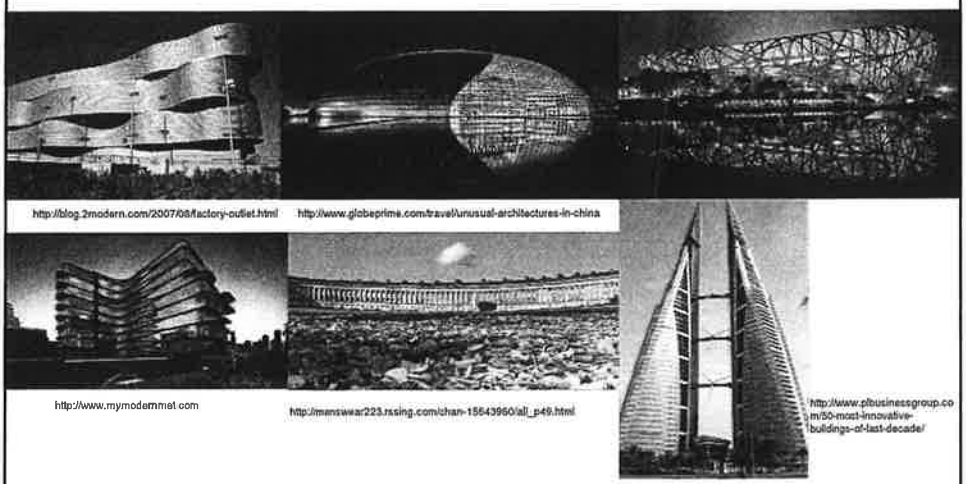
Z	= 60
Zo	=0.1
Zref	= 20 hence,

V = 7.7 ln (60/0.1) / ln (20/0.1) = 9.297 m/s

Speed of the wind at 60 m is 9.297 m/s

Source : Danish wind industry association

Factors that affect wind : Building form



http://blog.2modern.com/2007/08/factory-outlet.html http://www.globeprime.com/travel/unusual-architectures-in-china

http://www.mymodernmet.com http://menswear223.rssing.com/chan-15643960/all_p49.html

http://www.plbusinessgroup.com/50-most-innovative-buildings-of-last-decade/

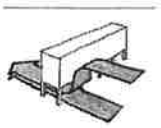
Factors that affect wind : Building form



Downwash



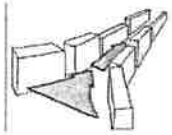
Corner effect



passageways



Low & high bldgs



Channel effects



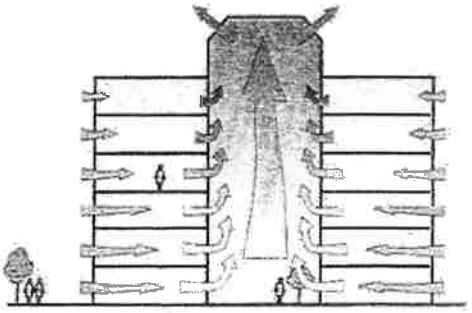
Building recesses

Source : philip oldfield University of Nottingham, 2009

Factors that affect wind : Building form



Cross ventilation



Stack ventilation

Factors that affect wind : Wind Sensitivity for building



1) Drafts
2) Sound
3) Rain
4) Humidity in wind
5) Heat

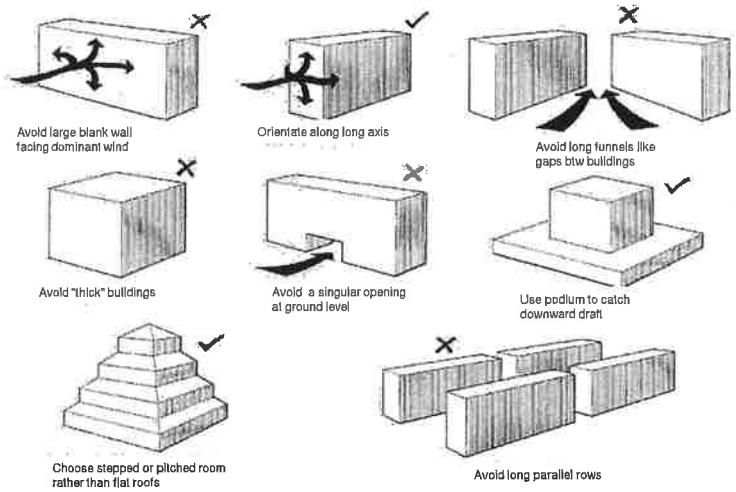
Wind environment is the wind flow experience by people and the influence it has on their activities

It is well understood that the structures influences the wind conditions at pedestrain level

Increasing importance is given to the wind environment for the pedestrain safety and comfort issues.

Source:Image
http://historymaniacmegan.com/2015/02/20/

Factors that affect wind : Wind Sensitivity for building



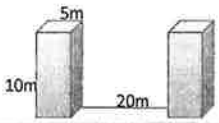
Choose form and arrangement of building to avoid downdrafts and shelter external spaces.

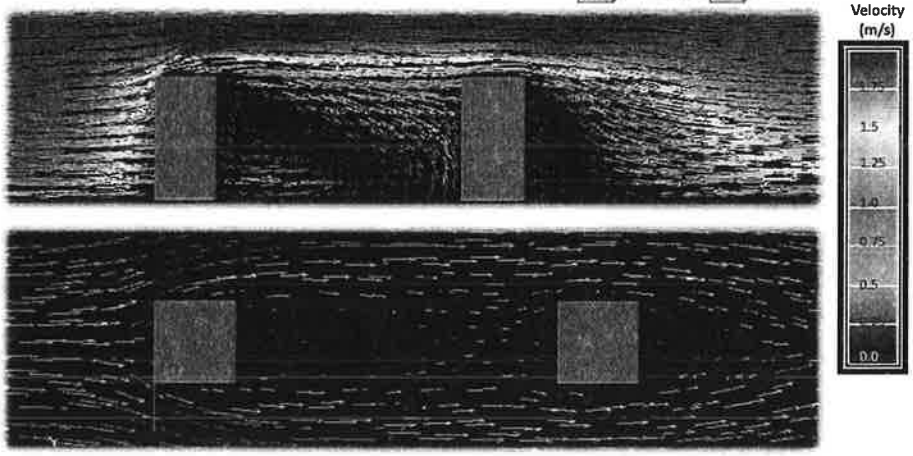
Source : philip oldfield University of Nottingham, 2009

Factors that affect wind : Building form

Stagnant

When buildings are close to each other, the building in front will block the wind and create an stagnant effect behind which will cause turbulence and poor ventilation.



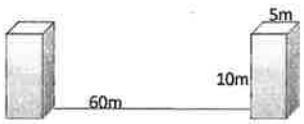


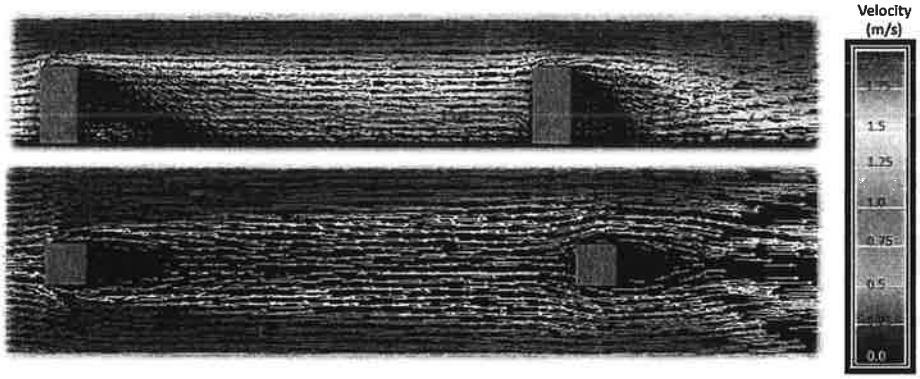
Source : Surbana ESD , 2010

Factors that affect wind : Building form

Stagnant

When buildings are located in a greater distance, the building in front will have minimum impact to the building behind in terms of ventilation.

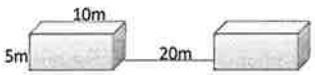


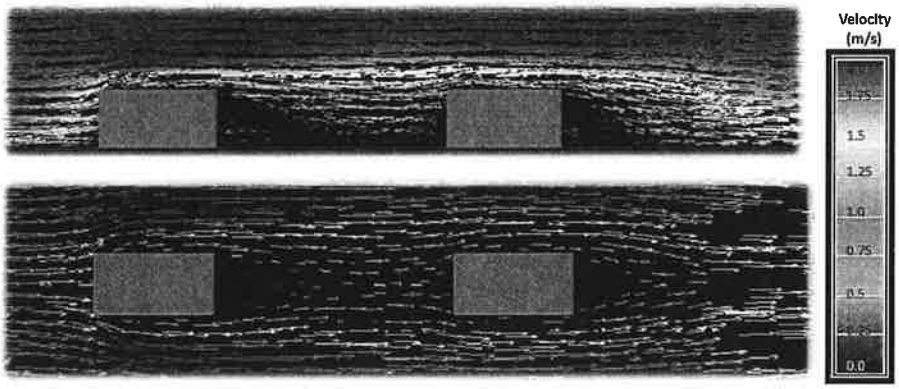


Source : Surbana ESD , 2010

Factors that affect wind : Building form

Stagnant
when buildings are close to each other, the building in front will block the wind and create an stagnant effect behind which will cause turbulence and poor ventilation.

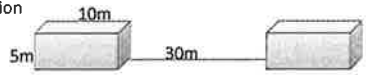


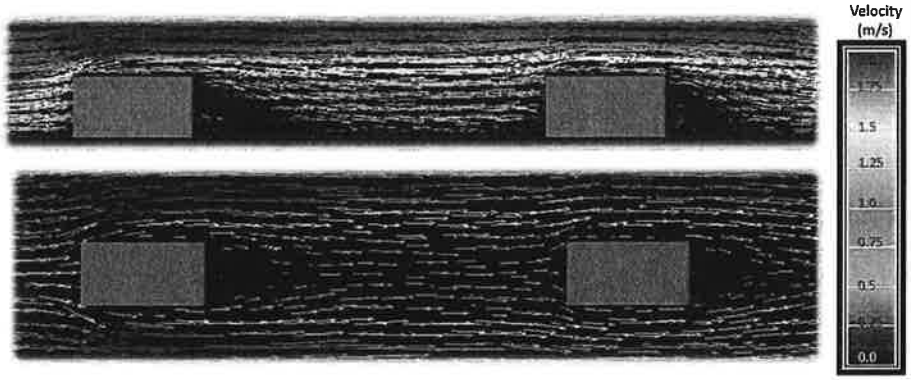


Source : Surbana ESD

Factors that affect wind : Building form

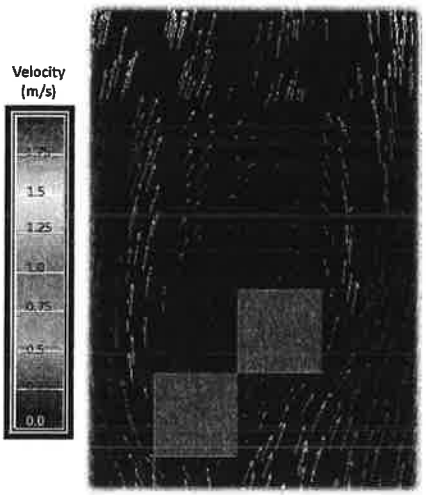
Stagnant
When buildings are located in a certain distance, the building in front will block part of the wind and affect the ventilation on the lower part of the building behind.





Source : Surbana ESD

Factors that affect wind : Building form

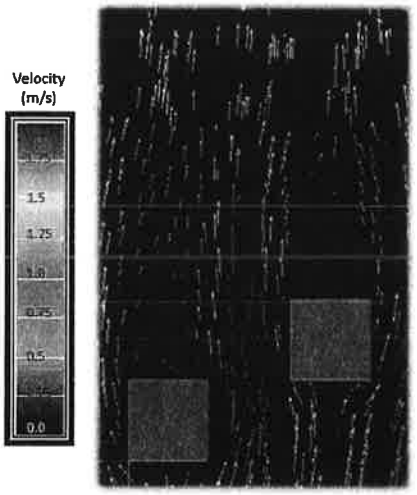


Stagnant effect

When buildings are located next to each other where there is no gap in between, there will be no ventilation created between the buildings.

Source : Surbana ESD

Factors that affect wind : Building form

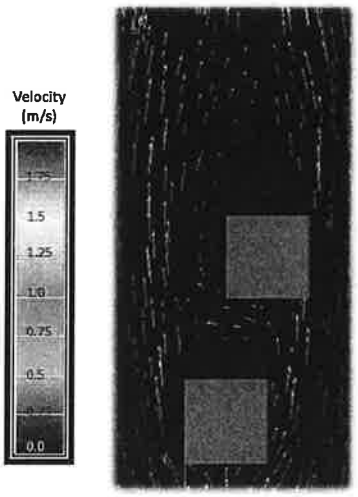


Canyon effect

When there is a gap between the buildings, canyon effect is created and channeled high speed wind in between the buildings.

Source : Surbana ESD

Factors that affect wind : Building form

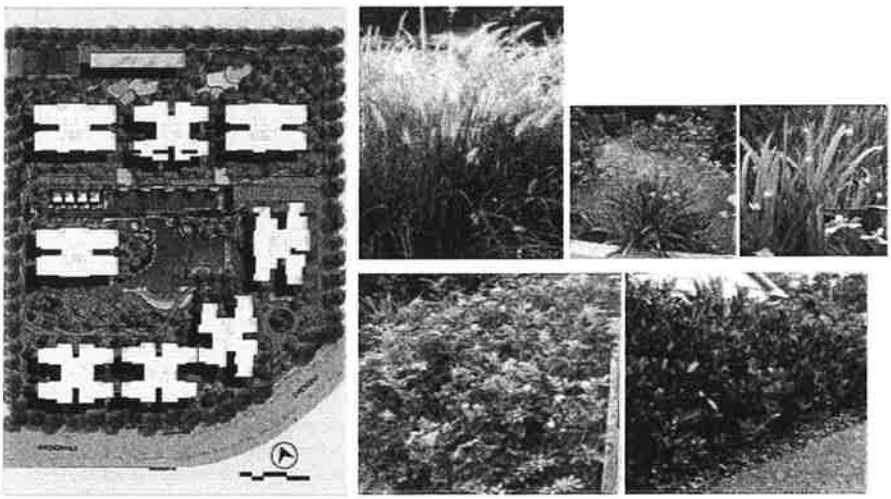


Scooping effect

When buildings created a gap in between and behind each other, wind will be scooped into the gaps.

Source : Surbana ESD

Vegetation influence air movement

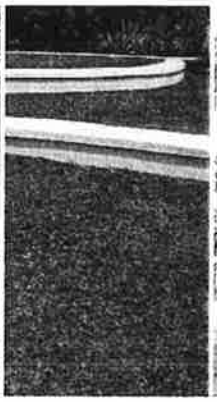


Vegetation influence air movement

Which one of the following plants have a Leaf Area Index of 4.5 ?



Scientific Name : Ixora ' Siam Ribbon'
Leaf Area Index (for green plot ratio)
4.5 (shrub & groundcover)



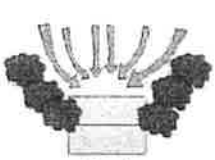
Scientific Name : Axonopus Compressus (Pearl Grass)
Leaf Area Index (for green plot ratio)
2.0 (turfgrass)



Scientific Name : Salacca magnifca
Leaf Area Index (for green plot ratio)
4.0 (Palm)

https://florafaunaweb.nparks.gov.sg
Source of images : national park board

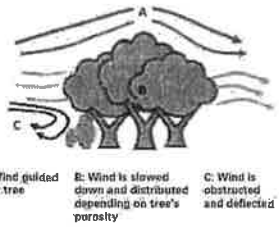
Vegetation influence air movement



Plan view



Plan view



A: Wind guided over tree

B: Wind is slowed down and distributed depending on tree's porosity

C: Wind is obstructed and deflected

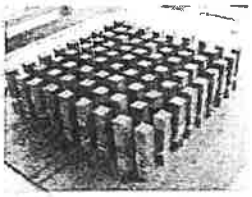
Angle	Wall	No trees				Trees				Effect Trees	
		WT	CFD	diff.	R^2	WT	CFD	diff.	R^2	WT	CFD
90°	A	19.7	23.0	17%	0.78	32.7	35.0	7%	0.77	66%	52%
90°	B	5.34	8.03	50%	0.95	2.69	7.10	169%	0.83	-50%	-12%
45°	A	18.4	14.3	-22%	0.81	31.0	21.6	-30%	0.75	69%	51%
45°	B	3.70	2.41	-35%	0.80	5.27	11.6	120%	0.27	42%	380%
0°	A/B	7.10	7.22	2%	0.88	9.73	12.2	26%	0.88	37%	70%

*Summary of the wall **maximal** normalized concentrations near the buildings inside the idealized street canyon W/H = 1.0, presented CFD results vs. the wind tunnel (WT) results (CODACS). For each case the relative difference between the simulation and the experimental value is included.*

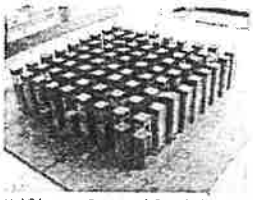
Comparative study by Atmosys

Image Source : BCA, Platinum building series : Building Planning and massing

Factors that affect wind : Permeability



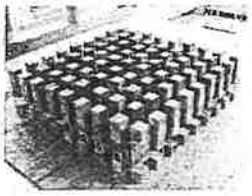
11% regular with plain sky



11% random with plain sky



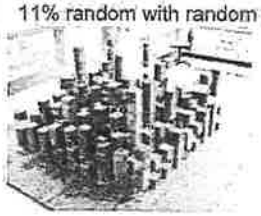
11% random with random sky



20% regular with plain sky



20% random with plain sky



20% random with random sky

At the same level ½ voids and full sty voids

At the random levels ½ voids and full sty voids

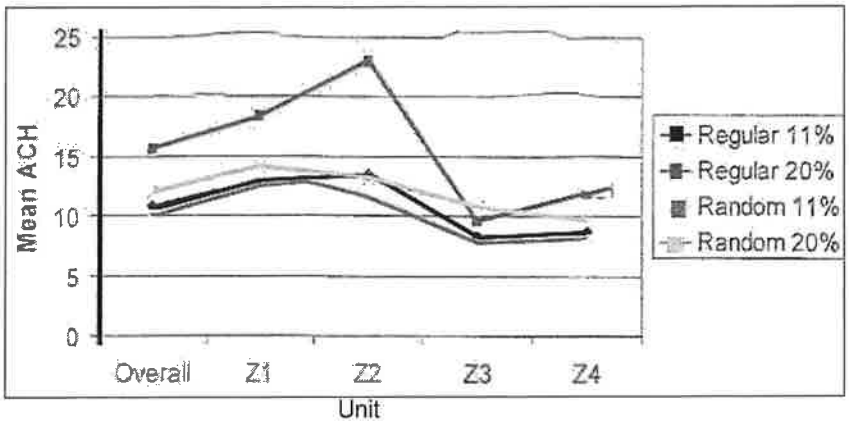
At the random levels ½ voids and full sty voids but different building height

Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Passive Design & Natural Ventilation, GMP 2017

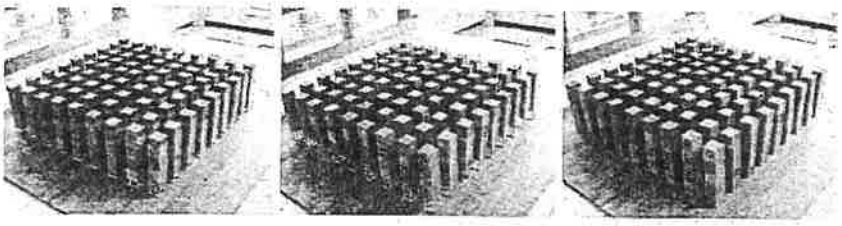
Factors that affect wind : Permeability

Plain Skyline under percentage of permeability and permeability type treatment



Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Factors that affect wind : Void decks

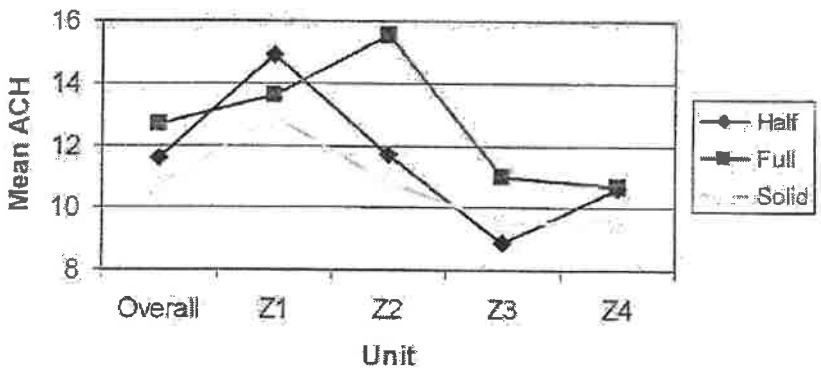


Half Voids Full Voids No Void

Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Factors that affect wind : Void Decks

Plain Skyline under different type of voids (air change)

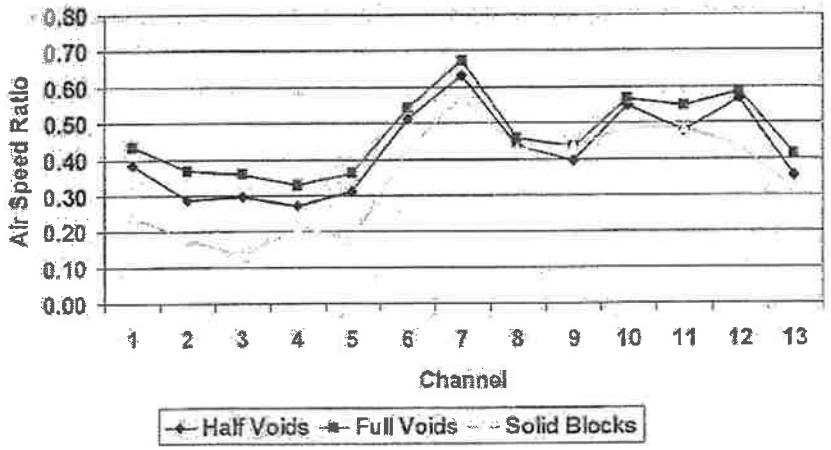


Full voids give best air change at pedestrian level

Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Factors that affect wind : Void Deck

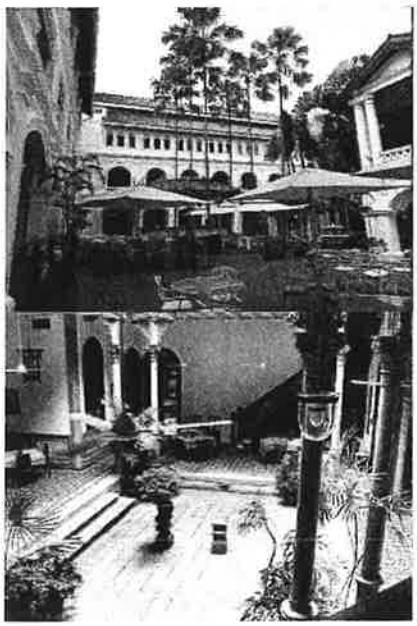
Air speed ratio for different types of voids



Source : Prof Wong Nyuk Hien, NUS , Studies on Urban design and their implied performance on ventilation , joint research Chinese Hong Kong University and NUS

Full voids give best air speed at pedestrian level

Courtyard strategy



There has been a history of courtyard buildings in Chinese Shop house and Colonial building in the Tropics

Studies[1] done showed there is a positive effect on passive cooling in hot and humid tropical climate.

Note 1 : Feriadi, H., Wong, N. H., Tham, K. W., Sekhar, C., Cheong (2000) Natural Ventilation Characteristics of Courtyard Buildings in Singapore,

Courtyard strategy

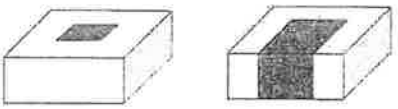
A. CLOSE COURTYARD

- The classic or standard subtype which defined on all sided of the courtyard by occupied zones. It is most prevalent type and can be and shape in plan (square, rectangle, triangular and circular). The only source of natural ventilation and daylight is the skylight opening or roof clerestory.

B. OPEN-SIDED COURTYARD

- A courtyard with one, two or three sides partially or completely glazed or covered by ventilation elements. By placing courtyard near to one or two perimeter walls, the occupied zone may have L, U shape building plan. Natural ventilation and daylighting can penetrate inside spaces through sidewall of the courtyard.





Source : Feriadi, H., Wong
Image Source :file-nut.com

Courtyard strategy

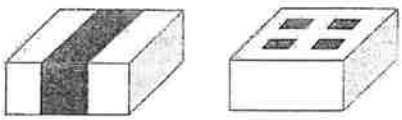
C. LINEAR COURTYARD

- Occupied zones on opposite sides of the courtyard with circulation connection across an elongated rectangle. Ends may be glazed or defined by building elements which allow the fresh air to penetrate into the building.

D. MULTIPLE LATERAL COURTYARD

- With more than one courtyard within a building, each is laterally disposed and spatially organizing a section of the building. Multiple courtyards are used for a building that has deep building plan and big occupied zone. The potential utilizing daylight and natural ventilation in this large building can still be achieved.





Source : Feriadi, H., Wong
Image Source : toberlinandbeyond.weebly.com1100

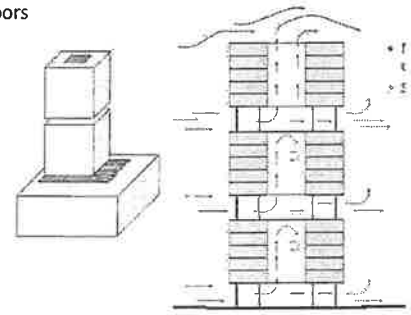
Courtyard strategy

E. **PARTIAL COURTAYRD**

- Any atrium that spatially organizes only a part of the building, can be a tower base form with the courtyard in the base. It can be a vertically stacked form with multiple courtyards in a high-rise building, each one relating only to a set number of floors or it can be adapted form spatially relating an old and new building. Multiple vertical courtyards can be combined with the sky garden concept, which incorporate natural landscape and ventilation on some floors in a high-rise building.

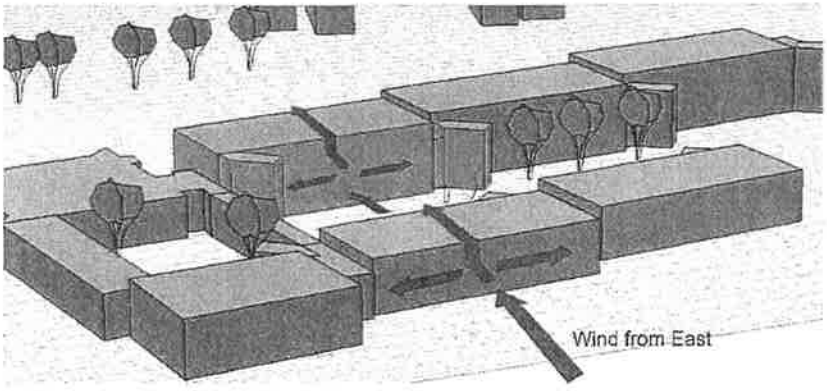


Image source http://demolition.com/



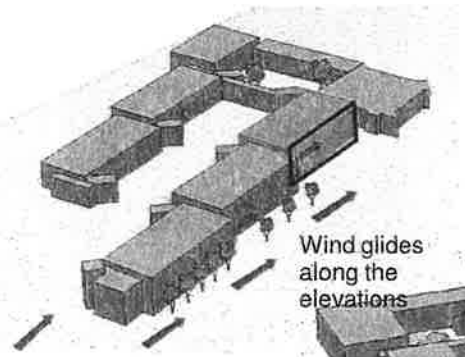
Source : Feriadi, H., Wong

Design Strategies – Potential for ventilation



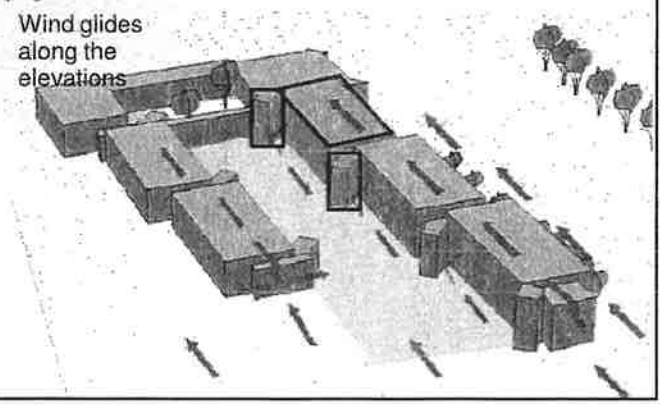
- The existing building prevents the wind from moving through and into the building

Design Strategies – North Wind

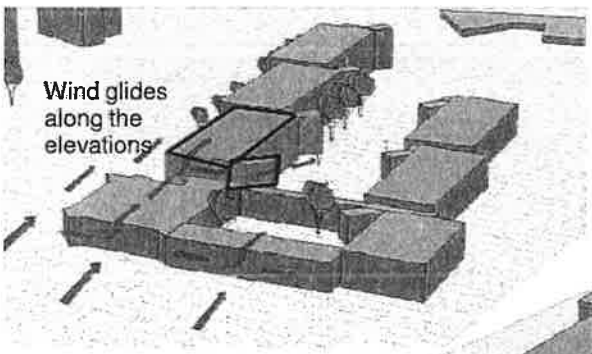


Wind Movement study

- Wind from north
- Surrounding buildings create an opening to catch wind.
- Opportunity to catch wind on west elevation, staircase and roof



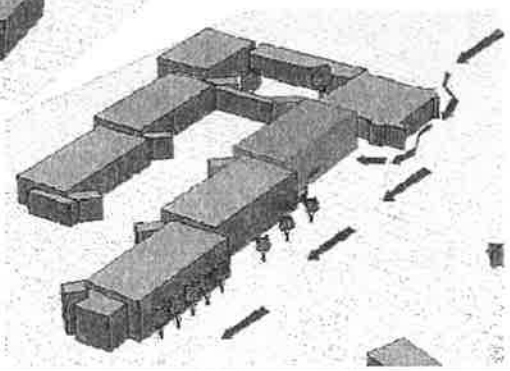
Design Strategies – South Wind



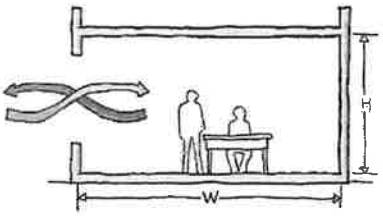


Wind from the south is blocked by the surrounding buildings of similar height

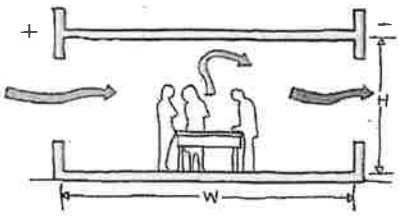
Opportunity for wind scooping is from the West elevation and roof of our building



Factors that affect wind : Internal Layout



Single sided ventilation – width of room (w) maximum 2.5 x height (h), limit to 5m. .



Cross ventilation (openings on opposite or 2 sides) maximum room width (w) = 5 x height (h), limit to 12-14m maximum

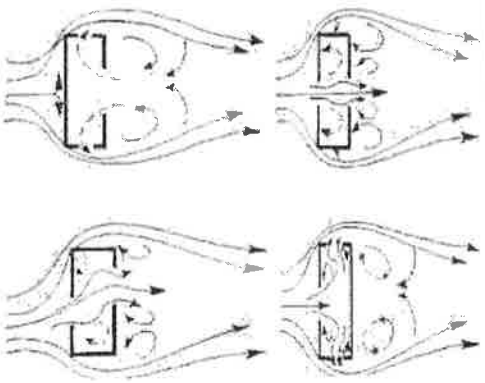
Due to the low wind speeds in Singapore, cross ventilation with windows facing towards the north and south directions are preferred.

Source, SGBC 2013

Factors that affect wind : Internal Layout

- Place horizontally two or more external permeable openings , one should consider the following rules :

1. Place opening on both sides (windward and leeward); windows or vent placed only leeward would determine an insufficient ventilation
2. In case wind is blowing perpendicularly to the façades where openings are located, one should avoid placing them on facing the other to increase air circulation

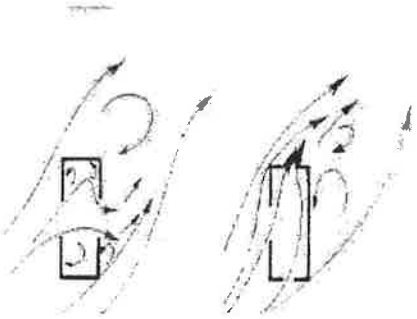


Source : Nottingham University : Dr Sergio A.

Factors that affect wind : Internal Layout

- Place horizontally two or more external permeable openings , one should consider the following rules :
3. Distrubition of opening on two – three adjacent walls may enhance an effective ventilation provided the windows (or vent) are sufficient spaced one to another

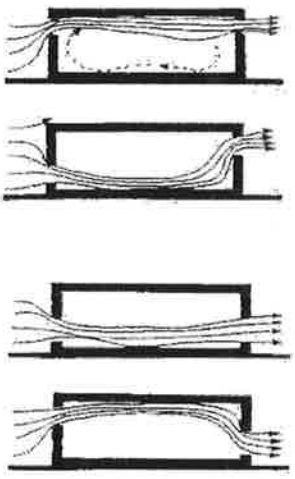
4. A wind blowing diagonally to the façade rather than perpendicularly improves cross ventilation both in the case of directly facing windows than with windows placed onto adjacent walls



Source : Nottingham University : Dr Sergio A.

Factors that affect wind : Internal Layout

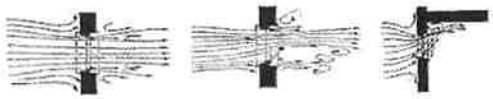
- The **vertical positioning** of openings influences cross ventilation, modifying both the **direction** – when it is determined mainly by the wind – and the **rate of air flow** – when is fostered by vertical circulation due to stack effect.
- In case of cross ventilation, the **mutual position of openings** (inlet and outlet) is of **fundamental importance** in order to optimise the specific result that a ventilation strategy should try to achieve.
- If the main aim is **cooling of thermal mass** (walls, floor, ceiling), as for a night cooling strategy, the **inlet** opening (not necessarily also the outlet) should be placed quite **close to the surfaces** that have to be cooled down.
- If the main aim is the lowering of the mean internal temperature (**passive cooling**), or the optimisation of indoor air quality, opening should be placed at a **height consistent** with the activities that take place in the space.
- In any case the **choice of the type of opening** is fundamental in order to achieve the best efficiency of the natural ventilation system.



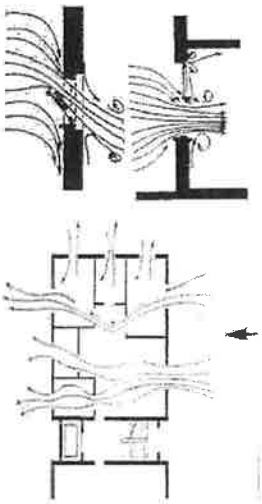
Source : Nottingham University : Dr Sergio A.

Factors that affect wind : Internal Layout

- In order to achieve an efficient ventilation of internal spaces, **spaces have to be distributed in an accurate way.**
- One should be aware of their **destination** (functions), the **patterns of occupation** (night/day), the overall **orientation** and exposition to **winds**, position of openings and their **typology**.



- In the planning disposition it should be considered the potential for cross ventilation, trying to **avoid vertical partitions** placed **perpendicularly to air circulation.**
- **Finishes** have to be placed in order **not to interfere with air movements** and with the speed of the flow rate.
- In residential buildings, **living rooms** and **studying areas** should be placed **windward**, while **kitchens** and **toilets** should be placed **leeward** (to avoid odour propagation).
- Bedrooms can be placed on both sides but need to be protected.

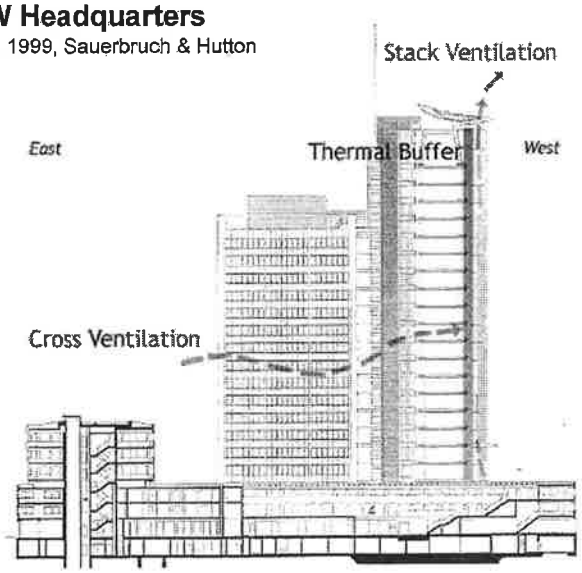


Source : Nottingham University : Dr Sergio A.

Factors that affect wind : Internal Layout

GSW Headquarters

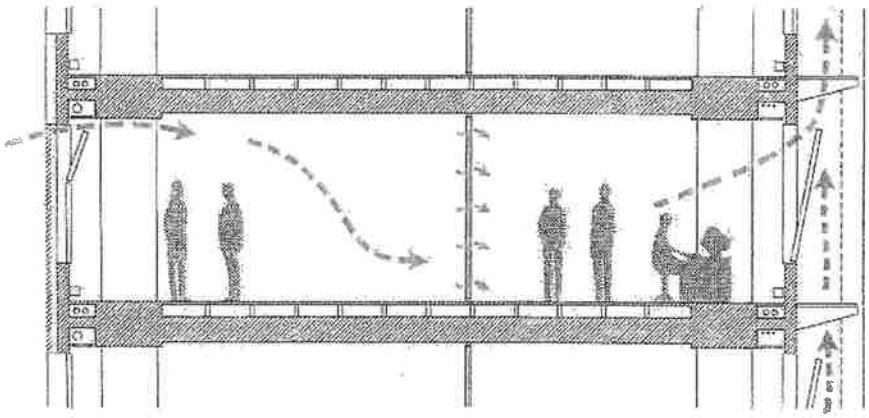
Berlin, 1999, Sauerbruch & Hutton



Factors that affect wind : Internal Layout

GSW Headquarters

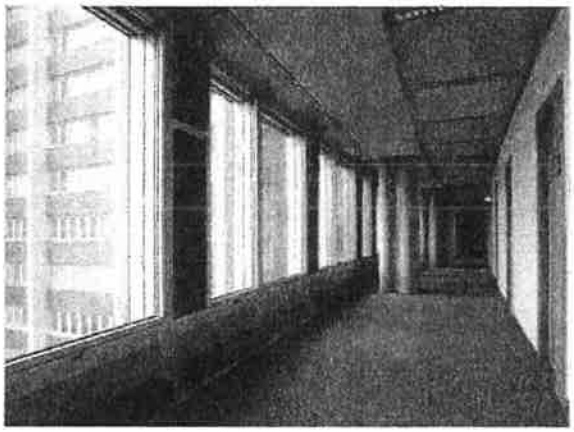
Berlin, 1999, Sauerbruch & Hutton

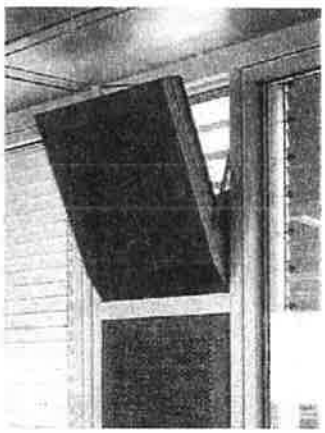


querlüftung - kombi/ost

cross ventilation - combi/east

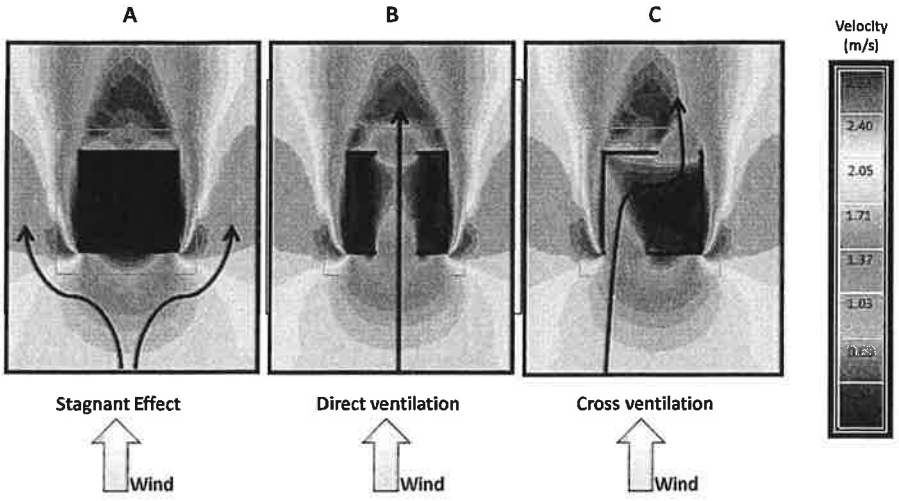
Factors that affect wind : Internal Layout





Eastern facade showing fresh air intake

Factors that affect wind : Internal Layout



Source : Surbana ESD

Factors that affect wind : Internal Layout

NATURAL VENTILATION ANALYSIS

- Assessing the effectiveness of natural ventilation in the context of specific design proposals for hostel units.

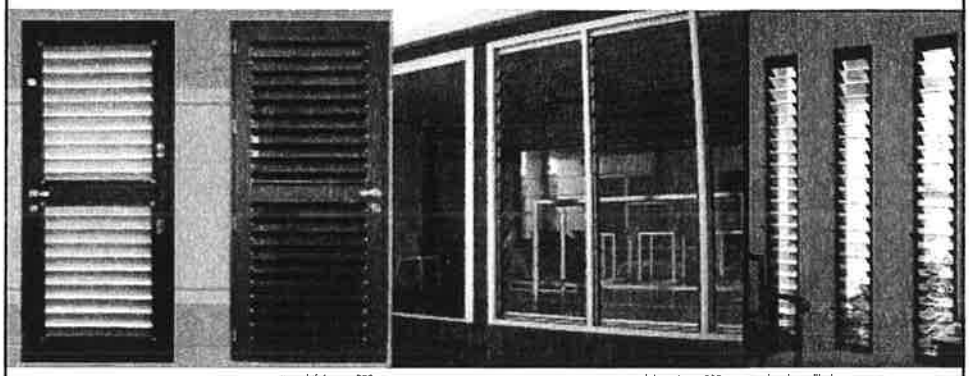
○ Louvers 百叶窗



Avg speed: 0.56 m/s 平均风速：0.56m/s	Avg speed: 0.63 m/s 平均风速：0.63m/s	Avg speed: 0.69 m/s 平均风速：0.69m/s	Avg speed: 0.7 m/s 平均风速：0.7m/s	Avg speed: 0.78 m/s 平均风速：0.78m/s
Case 1 Two louvers located at each side	Case 2: 案 Two louvers at one side	Case 3 One louver at right side	Case 4: : One louver at left side	Case 5: Four lovers evenly distributed

Source : Surbana ESD

Factors that affect wind : Internal Layout



www.lufata.com960 www.pinterest.com236 gzmingqi.en.alibaba.com

Factors that affect wind : Internal Layout



[Figure 21] Monsoon windows allow wind movement into the unit without allowing rain to penetrate into the unit.

Photo courtesy of Tim Griffith

Sketch © WOHA All Rights Reserved

Optimizing Wind in the Tropics

-Urban Level

1)Min heat generation - eg district air-conditioning systems

2) Widen Streets or green buffers

3) Varying Height Zoning

4) Allow sea breeze/prevailing winds to be unobstructed

5) Reduce Dark materials, increase cool materials and greenery

6) Control projection from building / signboards etc

7) Increase plaza spaces

Optimizing Wind in the Tropics

Project Level

-Make good use of cross ventilation by control the depth of the building

-Position of the openings are to be carefully considered

-Courtyard ,mid building openings and 1st storey void can be considered in a city environment which is usually of a higher density (ie increase building permeability)

-Daylighting , indoor air quality and acoustics need to be considered and optimized

-Building form and positions , landscape integration , skygardens (to create differential pressure and reduce unwanted wind noise)

-Staggering buildings

-Avoid large podium blocks below

Optimizing Wind in the Tropics

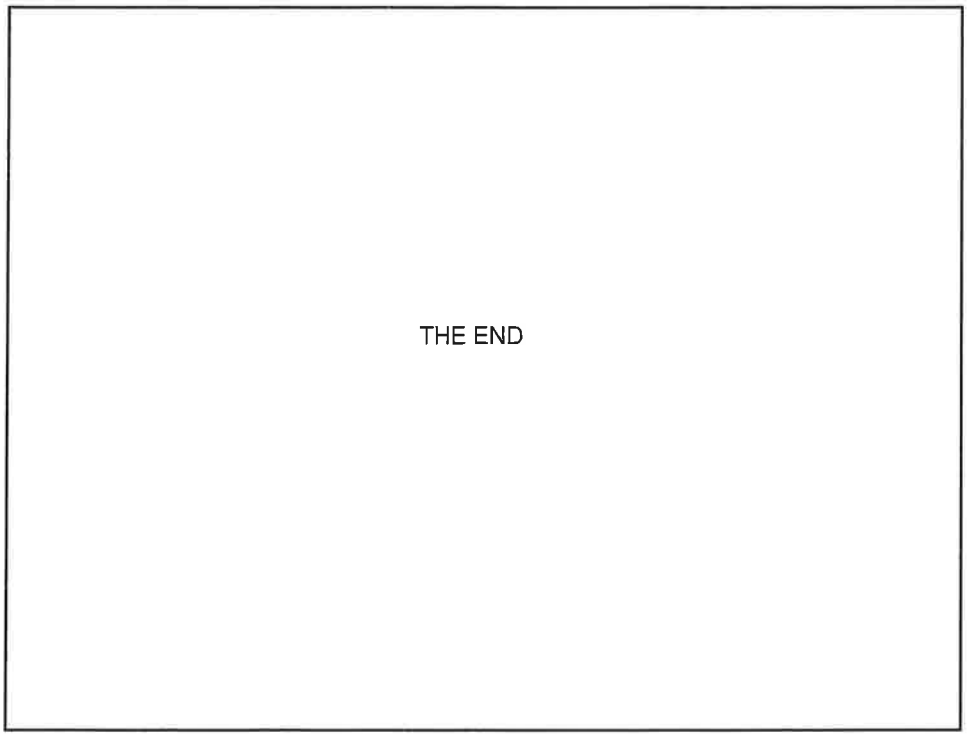
Project Level

-Increase the use of Cool materials

-Position areas (plant rooms, condensers) that generate heat from prevailing winds



Green parks, cycling paths for Bidadari, Punggol Matilda and Tampines North: HDB, Singapore News & Top Stories - The Straits Times


THE END

Part 4 : CASE STUDIES

We can't solve problems by using the same kind of thinking that we used when we created them !!

Albert Einstein



我们不可能以同样创造问题的思考模式了来解决问题

愛因斯坦

Purpose of Natural Ventilation

-Reduce energy usage / carbon emissions

-Reduced capital, maintenance and running costs

-Mitigates sick building syndrome , reducing concentration of pollutants in spaces

-Improve comfort (wider range of acceptable temperatures) for users

-Improve productivity

-Improve user control

-Reduces plant space requirements and improve GFA efficiency

Accelerating passive building mass cooling

Purpose of Natural Ventilation

Some considerations for the use of natural ventilation :

- What is the wind for ?: Does it require tight control for the room air conditioning ? (such as: clean room, ICU etc.)
- Does it need fresh air ? (such as: kitchen, sport hall, factory etc) – Establish criteria
- How is the climatic condition ? (temperature diurnal, relative humidity, wind direction etc)
- How is the environmental conditions around the building ? (air pollution, noise etc)
- Does it require other equipment during the worst condition ? (no wind condition,haze etc)
- Are there any other considerations due to energy constrains ? (regarding the cost and availability of the electric power)
- Action and Design : How to improve the conditions so that it is comfortable.

Purpose of Natural Ventilation

Do not underestimate user's Role

- Occupants need much more understanding of the building they live in.
- Occupants are willing to take personal adjustment make themselves more comfortable. They don't shy away from putting clothes on the morning or changing them to suit the weather. We can ask them to be involved in operation of our buildings.
- Simple manual building control is more preferred rather than sophisticated equipment.



Case studies

Local wisdom : traditional Architecture

-Airflow around the house
-Permeable building material
-Semi outdoor spaces
-High roofs
-Landscaped



http://nosurplus.blogspot.sg/2015/04/running-out-of-time.html



http://cabby65.blogspot.sg/2015/04/remembering-bukit-panjang-village-in.html

Case studies

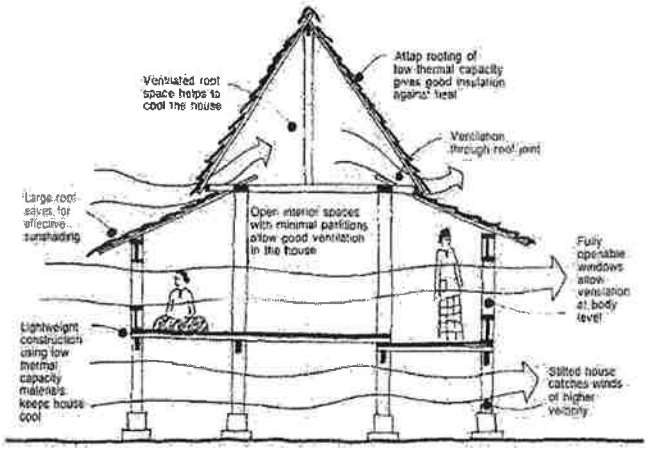
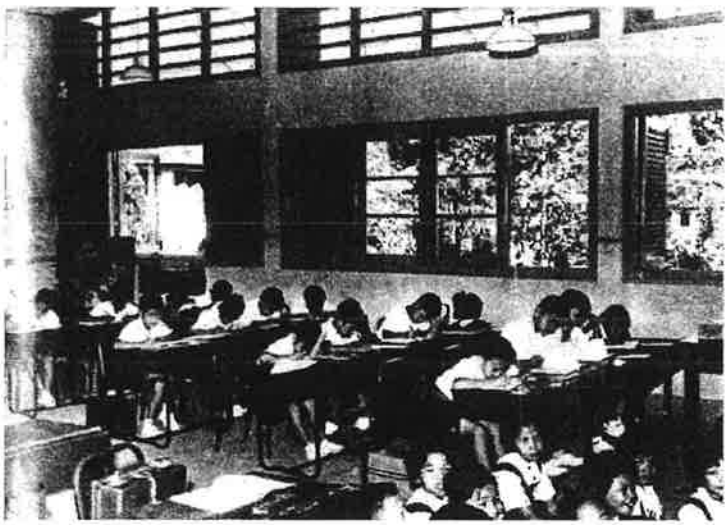


Fig-3.3 Climatic design of a Malay house
Source : Lim Jee Yuan; 1991, pg 80

Passive Design & Natural Ventilation, GMP 2017

Case studies



Case studies

Local wisdom : Colonial Architecture

Elaborate roof
High ceiling
Corridor/verandah (five foot way)
Terraces for outdoor activities





www.insightguides.com

Passive Design & Natural Ventilation, GMP 2017

Case studies

Local wisdom : Colonial Architecture

Elaborate roof
High ceiling
Corridor/verandah (five foot way)
Terraces for outdoor activities



www.telegraph.co.uk



singaporebrides.com



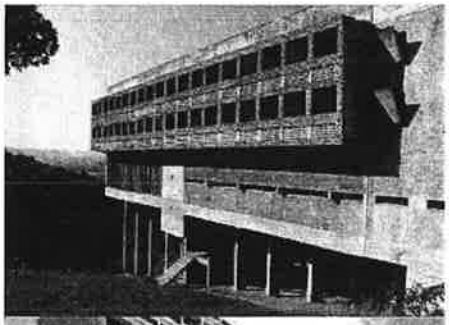
www.world-traveler.eu/432

Modern houses

Case studies

Adapting Modern Architecture to the Tropics

Sainte Marie de La Tourette



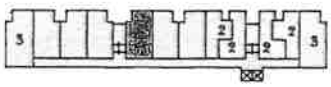
Le Corbusier's Berlin Unité



en.wikipedia.org2560

Case studies

Adapting Modern Architecture to the Tropics



Source:- "h'n'h" slab blocks for good cross ventilation
Source: Wong & Yeh, 1985 ,pg 73)



http://remembersingapore.org/singapore-en-bloc-flats/



https://seejy.wordpress.com/2012/10/03/my-grandmas-stories-a-roof-over-our-heads/

Case studies

HDB façade design over the years

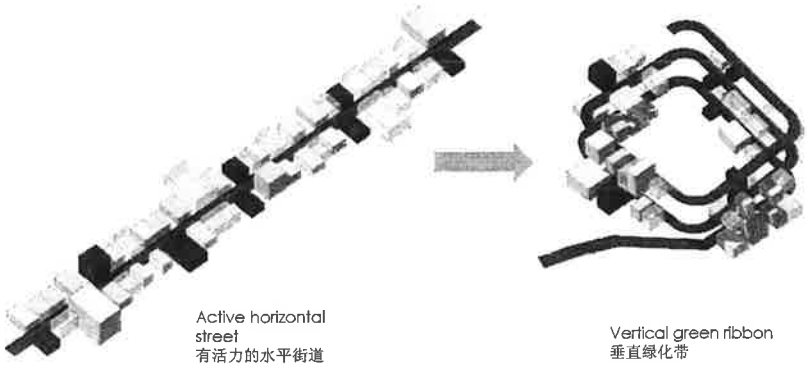
1965 to 1970s Source : HDB	Late 1970s - 1980s Photo by Owen wee	early 1990s Photo by owen wee	2006 Photo by: William Tan, 2007
			

DESIGNING FOR NATURAL VENTILATION

Case 1

Case studies

A NEW AMENITIES CONCEPT
配套设施概念



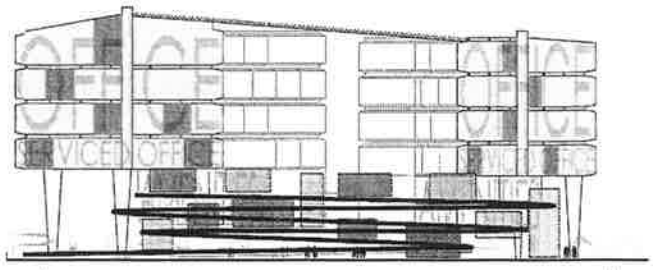
Active horizontal street
有活力的水平街道

Vertical green ribbon
垂直绿化带

Green Ribbon Gallery
垂直绿化带

Case studies





Recreational and entertainment amenities are consolidated at the lower levels of the building. These lower levels are integrated with the green spine, creating a "living" realm that embraces and interacts with the green spine at all times of the day. Upper levels are planned for commercial flexibility.

康乐和娱乐设施都整合在建筑物较低的楼层。这些楼层与地面的中央绿化带的景观在纵横方向上相互融合渗透，营造出全天候“活”在绿色的大自然中的境界。建筑物的上层则规划设计成灵活，多样的商业建筑。

Green Ribbon Gallery
垂直绿化带

Passive Design & Natural Ventilation, GMP 2017

Case studies

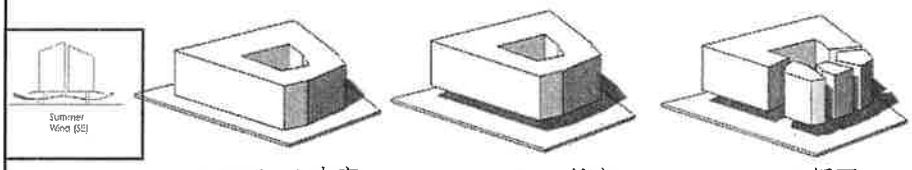


Void only 中空 Compare to Lift up 抬高 Compare to Split 拆开

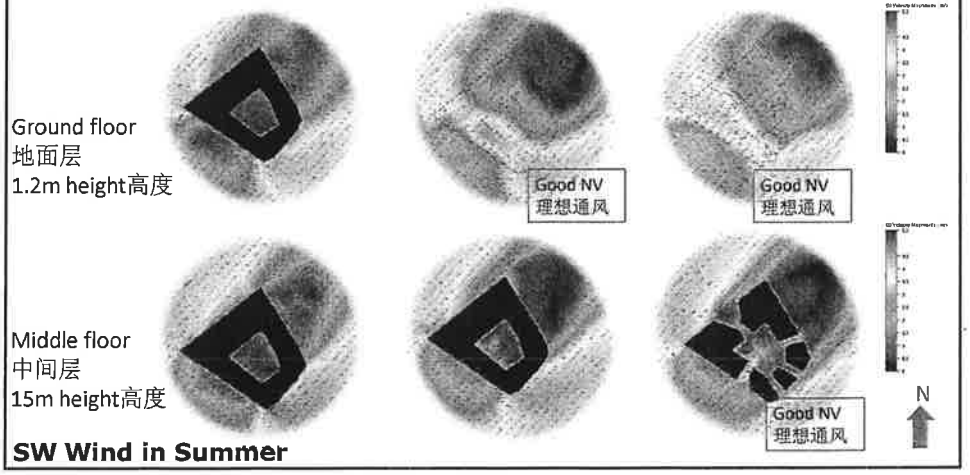
Building Form Analysis
建筑形体设计分析

Passive Design & Natural Ventilation, GMP 2017

Case studies



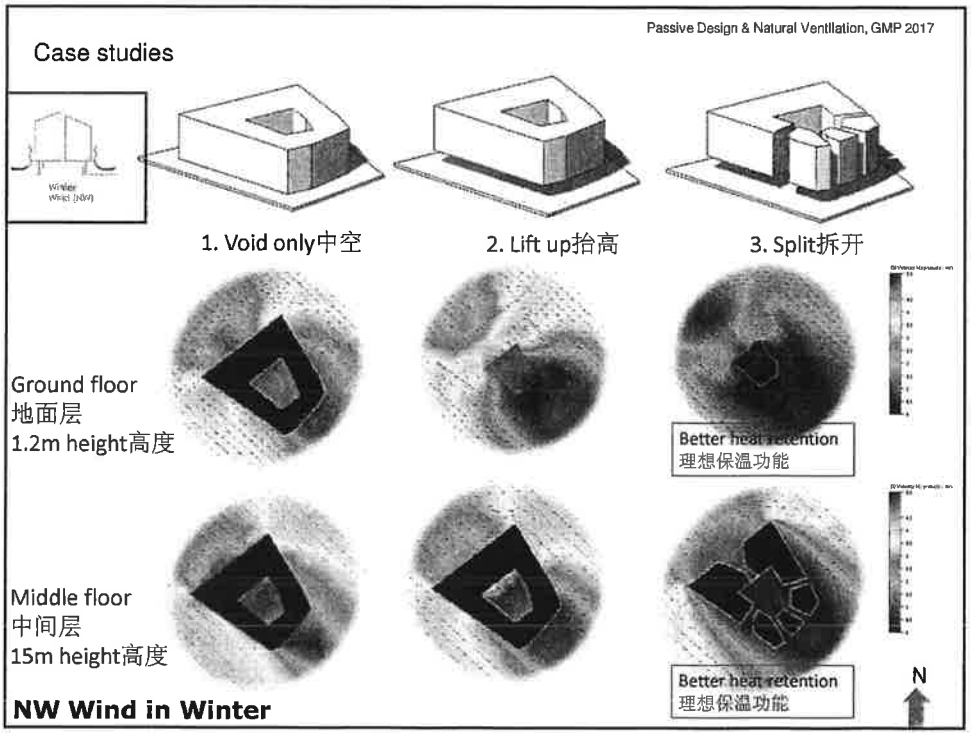
1. Void only中空 2. Lift up抬高 3. Split拆开

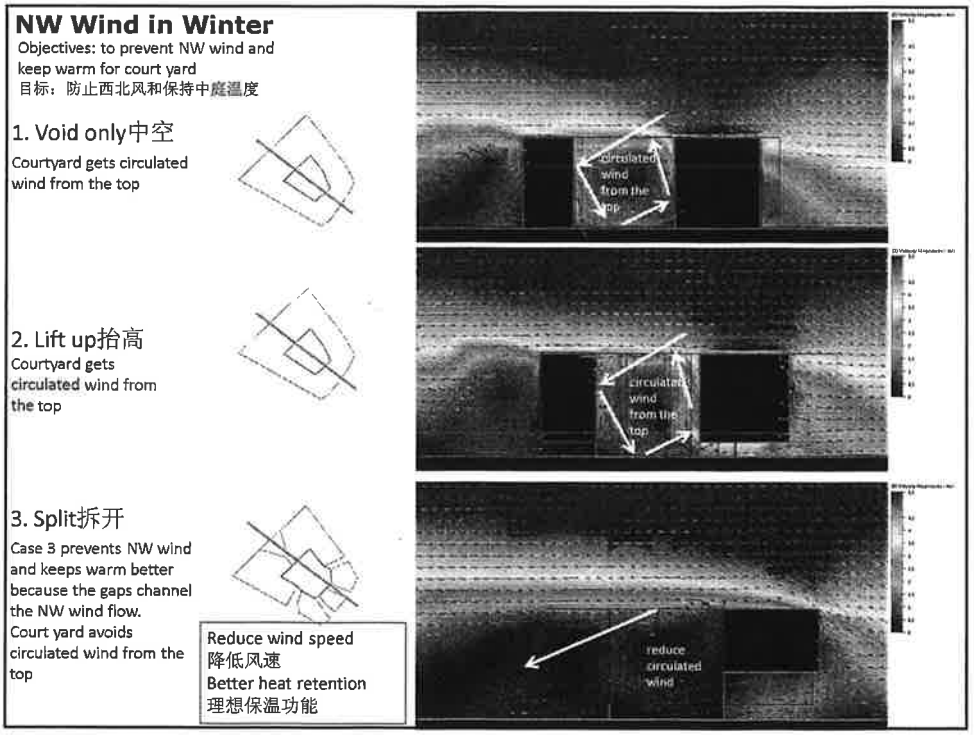


Ground floor
地面层
1.2m height高度

Middle floor
中间层
15m height高度

SW Wind in Summer


Passive Design & Natural Ventilation, GMP 2017
Case studies
1. Void only中空
2. Lift up抬高
3. Split拆开
Ground floor
地面层
1.2m height高度
Better heat retention
理想保温功能
Middle floor
中间层
15m height高度
Better heat retention
理想保温功能
N
NW Wind in Winter

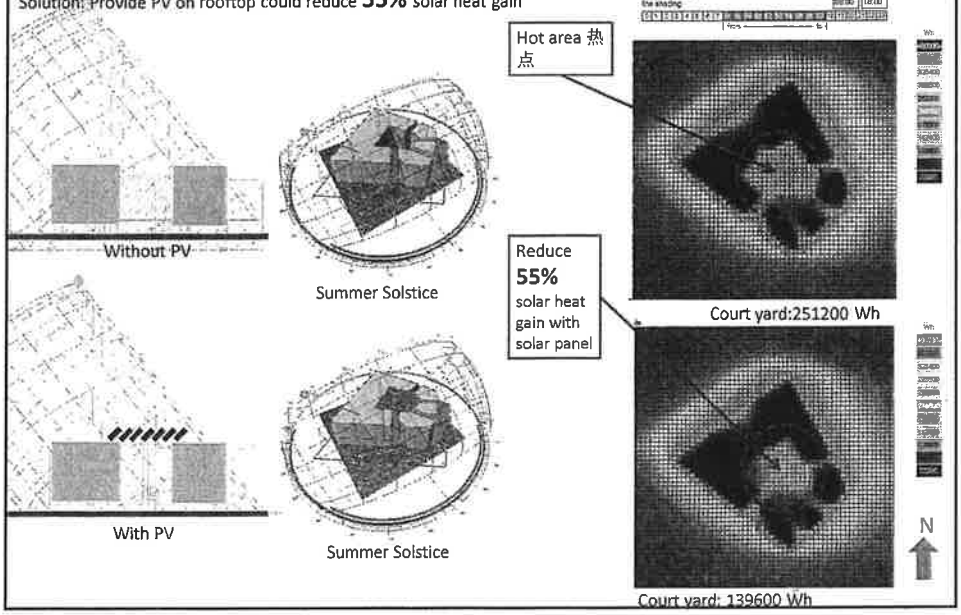

NW Wind in Winter
Objectives: to prevent NW wind and keep warm for court yard
目标：防止西北风和保持中庭温度
1. Void only中空
Courtyard gets circulated wind from the top
2. Lift up抬高
Courtyard gets circulated wind from the top
3. Split拆开
Case 3 prevents NW wind and keeps warm better because the gaps channel the NW wind flow.
Court yard avoids circulated wind from the top
Reduce wind speed
降低风速
Better heat retention
理想保温功能
circulated wind from the top
circulated wind from the top
reduce circulated wind

Solar Analysis in Summer夏季日照分析

Case studies

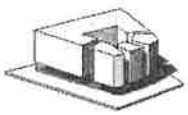
Objectives: to reduce heat gain for court yard

Solution: Provide PV on rooftop could reduce **55%** solar heat gain

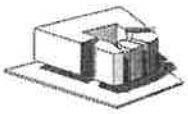


Case studies

Conclusion 结论



1. Lift up, open space at ground floor
   向上提起，一层作为开放空间



2. Split at SW and SE for better NV in Summer
   在西、南方拆开，为夏季提供更好的通风



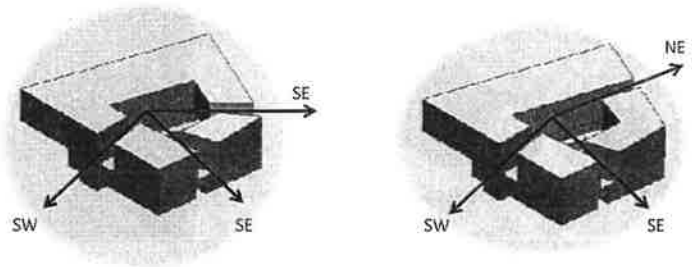
3. Close at NW to prevent cold wind in Winter
   冬季封闭防止冷风进入



4. Cover the void by open PV roof to reduce heat gain，whilst maintaining good natural ventilation
   光伏屋顶覆盖中庭，以减少热量的吸收并仍保持良好 的自然通风

Building Form Analysis
建筑形体设计分析

Case studies



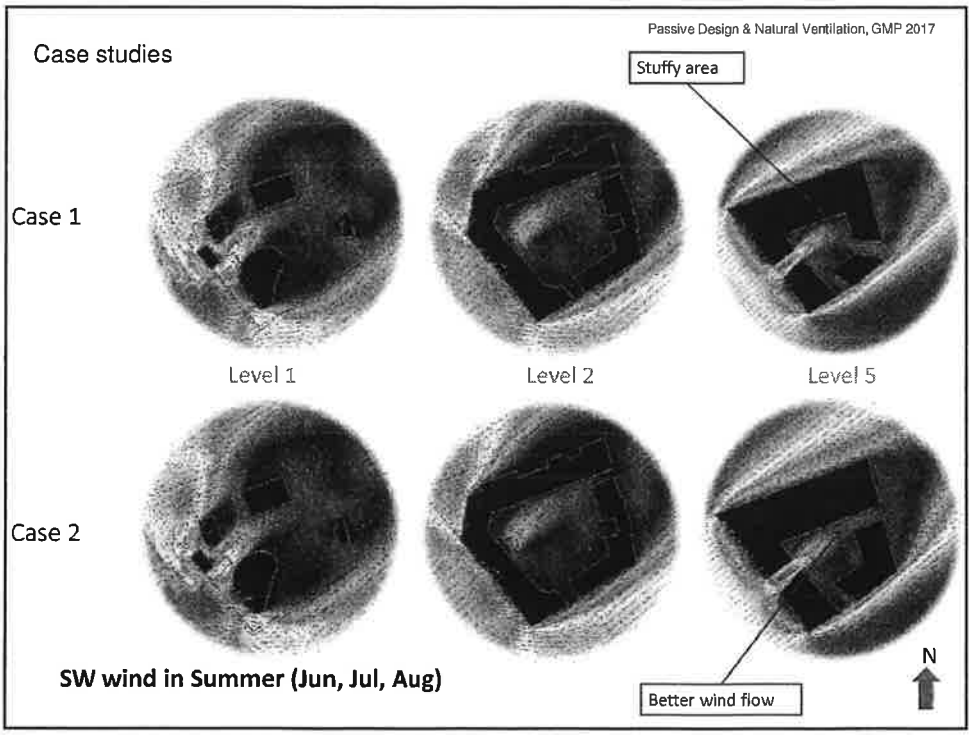
Case 1
方案一

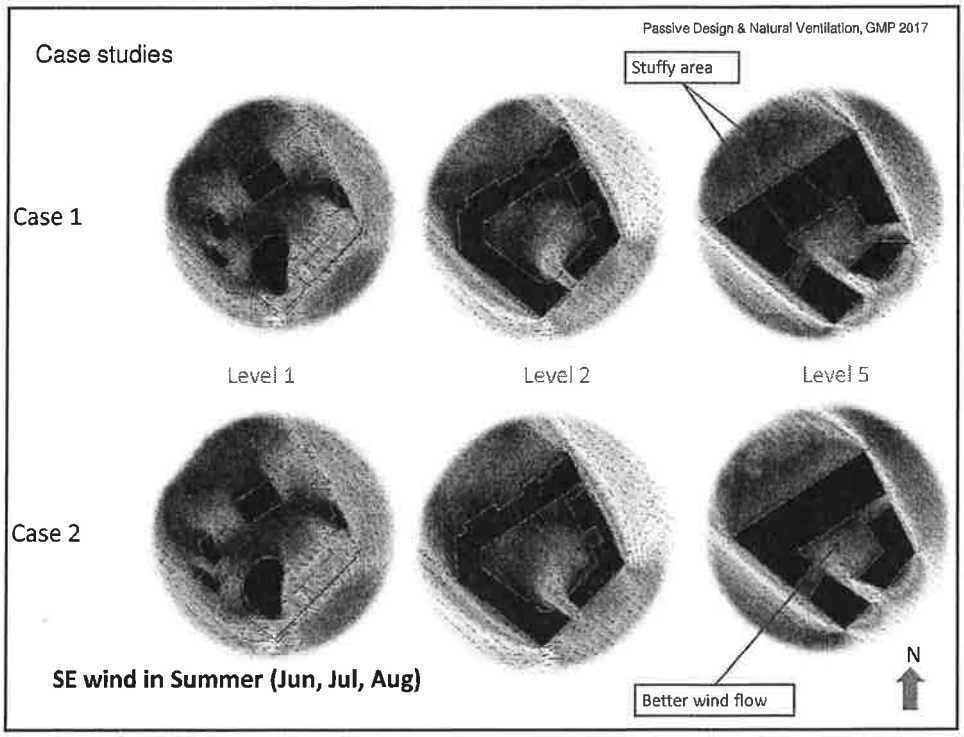
Compare to

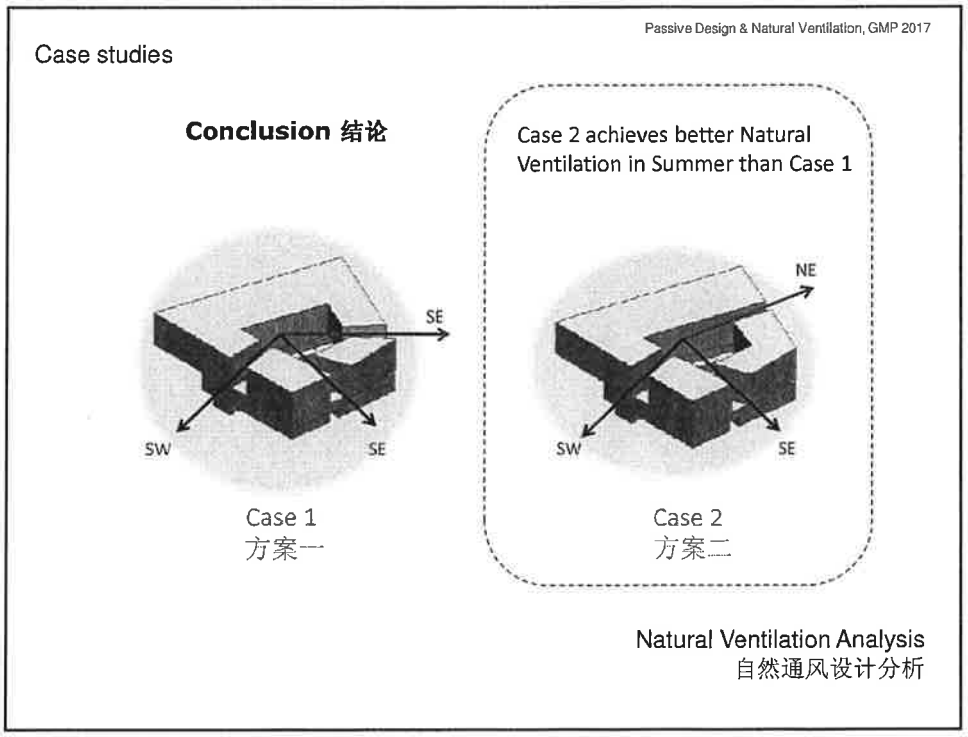
Case 2
方案二

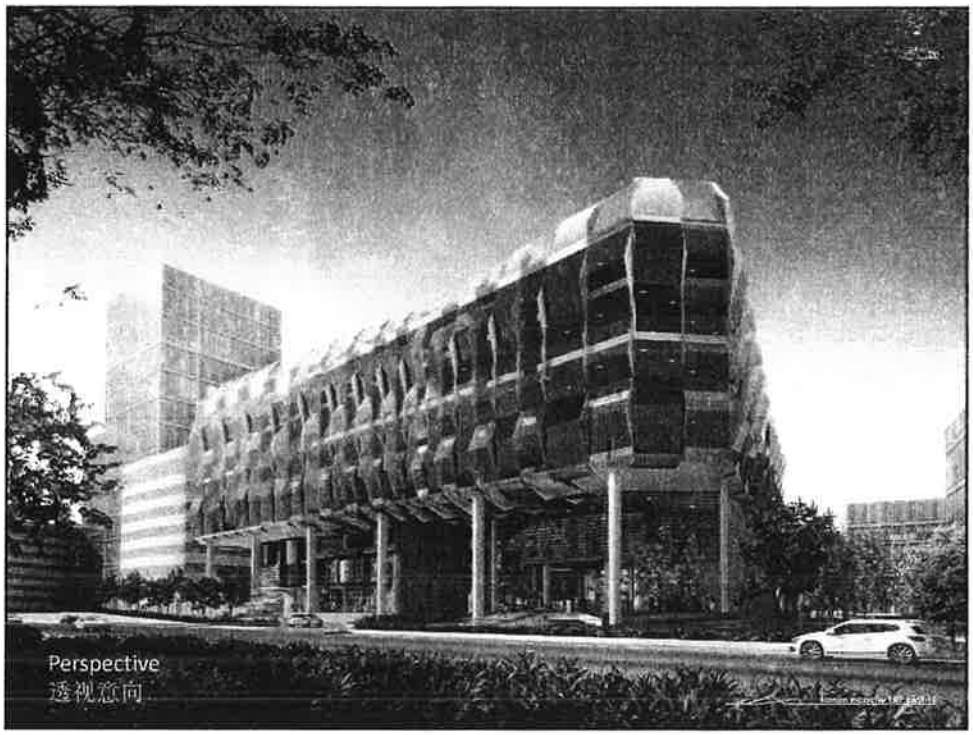
Natural Ventilation Analysis
自然通风设计分析

Case studies



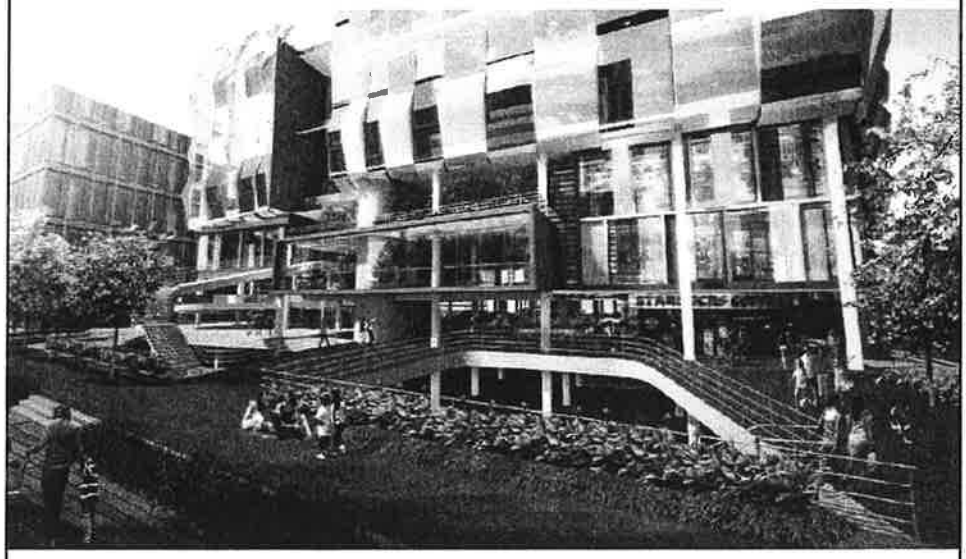

Passive Design & Natural Ventilation, GMP 2017
Case studies
Stuffy area
Case 1
Level 1
Level 2
Level 5
Case 2
SE wind in Summer (Jun, Jul, Aug)
Better wind flow
N


Passive Design & Natural Ventilation, GMP 2017
Case studies
Conclusion 结论
Case 2 achieves better Natural Ventilation in Summer than Case 1
SE
SW
SE
NE
SW
SE
Case 1
方案一
Case 2
方案二
Natural Ventilation Analysis
自然通风设计分析

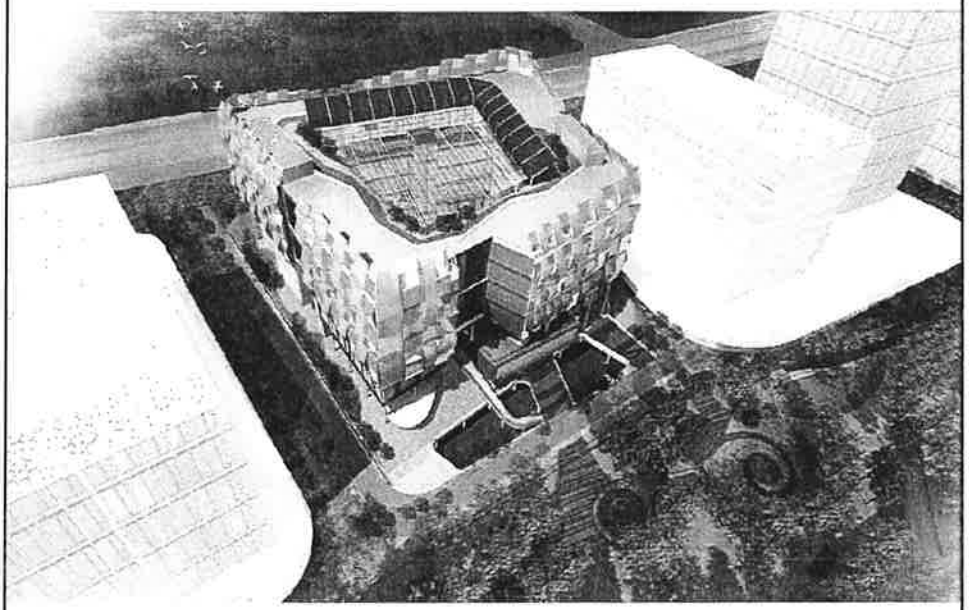

Perspective
透视意向


Perspective
透视意向

Passive Design & Natural Ventilation, GMP 2017



Passive Design & Natural Ventilation, GMP 2017



DESIGNING FOR NATURAL VENTILATION

Case 2

Case studies

Findings
The Wind Analysis Contour Diagram reveals **that the Level 1 receives adequate natural** ventilation when the prevailing wind is from the South.
Level 1 Workshop

Recommendation
Additional ventilating solutions may be **proposed to ensure that the thermal** comfort is still good when the prevailing wind direction is from north.



Velocity Contour with Prevailing Wind Direction **from NORTH** Wind Velocity at 2.0m/s



Velocity Contour with Prevailing Wind Direction **from SOUTH** Wind Velocity at 2.8m/s



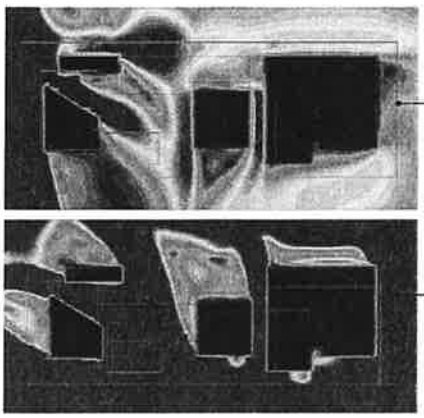
Wind Velocity taken at 1.2m above the floor

Case studies

Findings
The Wind Analysis Contour Diagram reveals that Level 2 receives adequate natural ventilation most of the time.

Recommendation
Additional ventilating solutions may not be necessary. On the other hand, rain protection may be required.

Level 2 Social Plaza



Velocity Contour with Prevailing Wind Direction from NORTH
Wind Velocity at 2.0m/s

Velocity Contour with Prevailing Wind Direction fromSOUTH
Wind Velocity at 2.8m/s



Wind Velocity taken at 1.2m above the floor

2nd floor open plaza natural ventilation is good

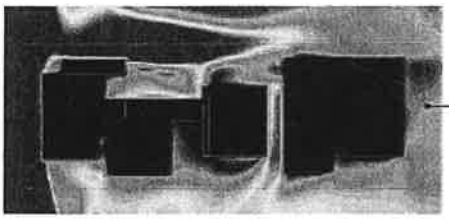
Passive Design & Natural Ventilation, GMP 2017

Case studies

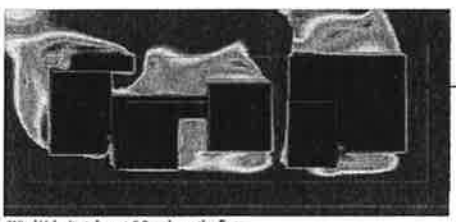
Findings
The Wind Analysis Contour Diagram reveals that generally Level 5 receives adequate natural ventilation.

Recommendation
Additional ventilating solutions may not be necessary.

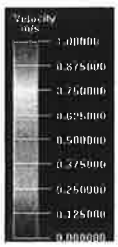
Level 5



Velocity Contour with Prevailing Wind Direction from NORTH
Wind Velocity at 2.0m/s



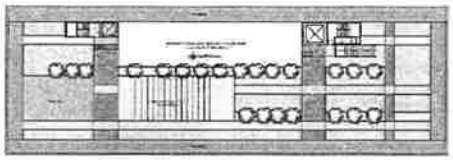
Velocity Contour with Prevailing Wind Direction from SOUTH
Wind Velocity at 2.8m/s



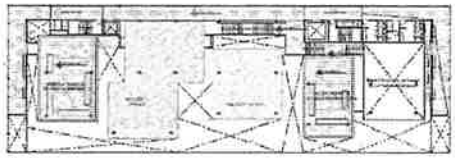
Wind Velocity taken at 1.2m above the floor

Case studies

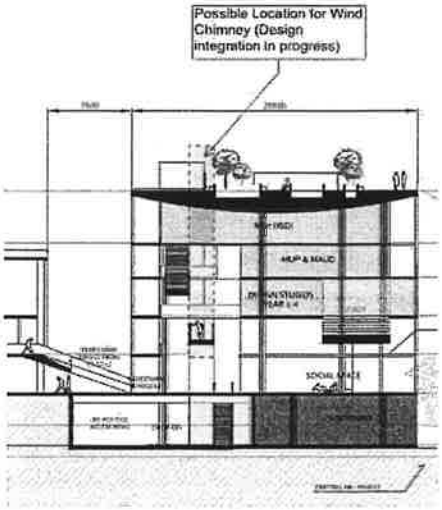
Proposal
Wind Chimney is proposed to help ventilate spaces that are unable to receive much natural ventilation.



ROOF

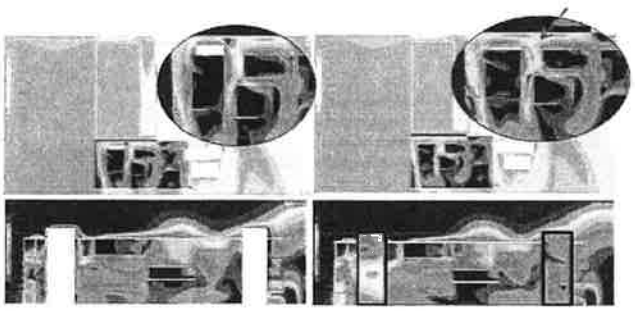


LEVEL 6



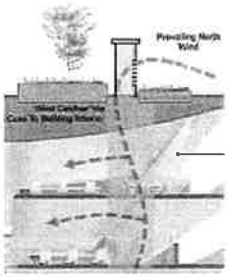
Case studies

Proposal
Wind Chimney is proposed to ensure that the thermal comfort is still good when the prevailing wind direction is from north.



Without wind Chimney

With wind Chimney



Diagrammatic Wind Flow

Introducing the wind chimney helps to enhance the ventilation in the atrium area

DESIGNING FOR NATURAL VENTILATION

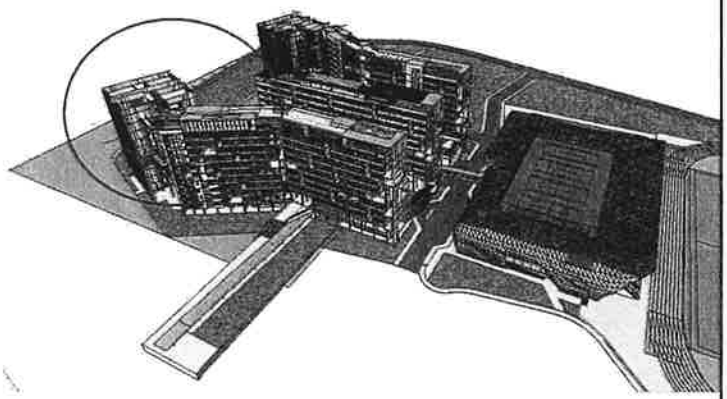
Case 3

Case studies

Building Layout Design

Air movement principle – wind scoop





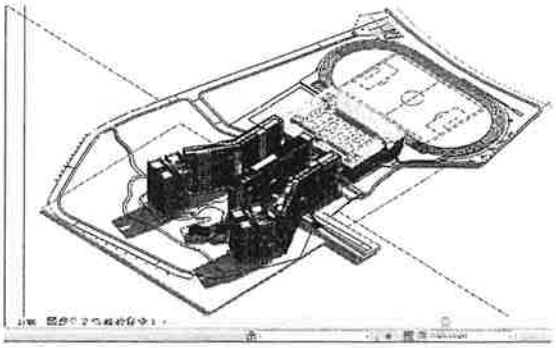
Case studies

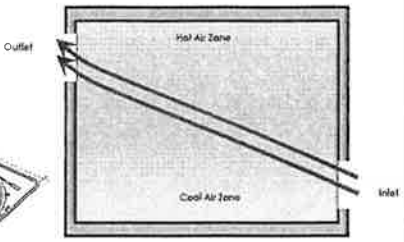
Thermal Comfort Design

Identify the air intake

Under the principle of hot air rises, project shall also consider shadow analysis so that the ventilation inside the room is comfortable.

Air movement principle – hot air rises





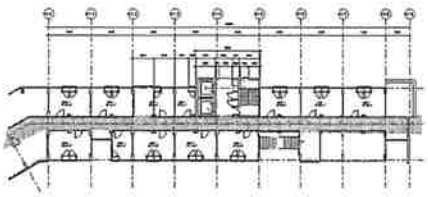
LOOK Architects

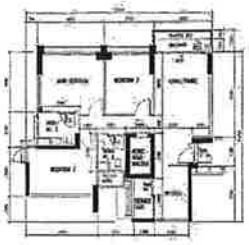
Case studies

The Challenge

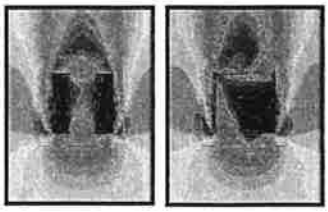
No cross ventilation for the hostel unit

Compare with a residential unit, the hostel does not have cross ventilation when considering the doors to be closed.





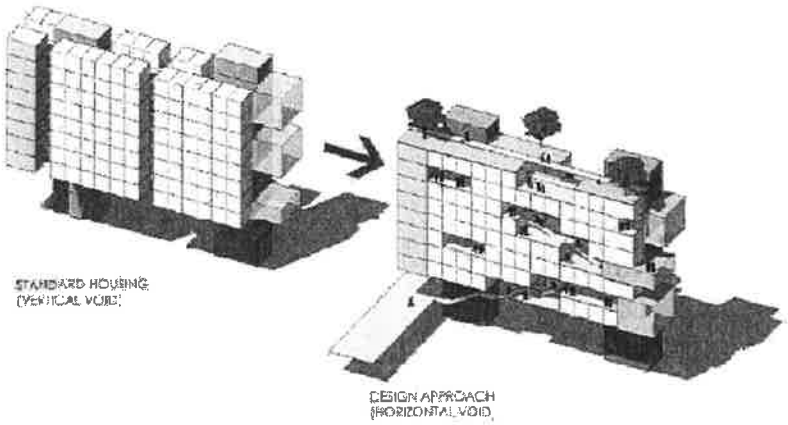




LOOK Architects

Case studies

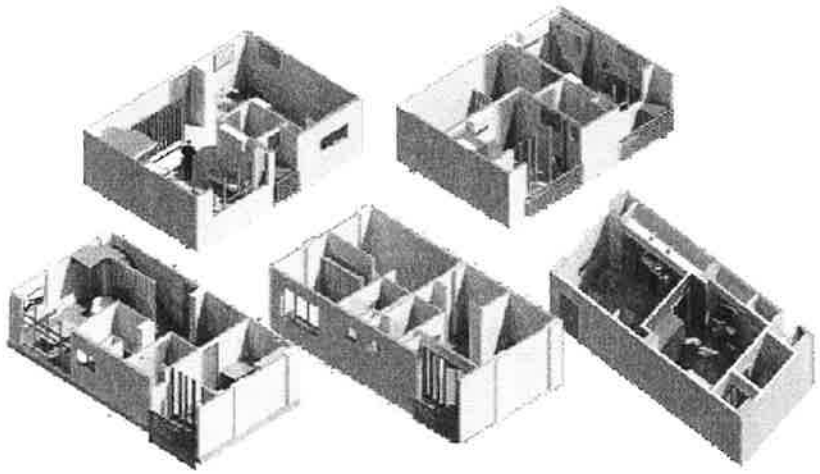
Create Openings



LOOK Architects

Case studies

Unit Design



LOOK Architects

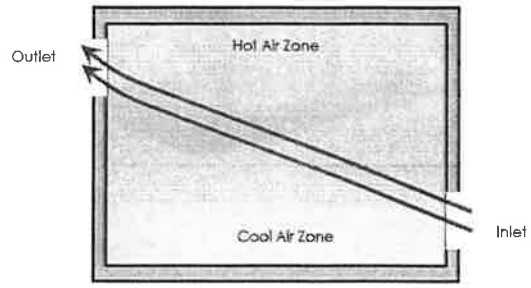
Case studies

Step One – Allow more air in

Air movement principle – wind scoop

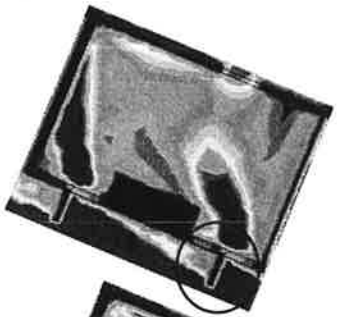


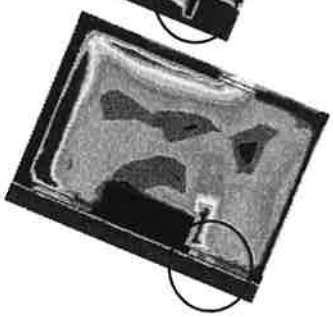
Air movement principle – hot air rises

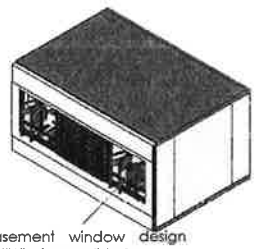


Case studies

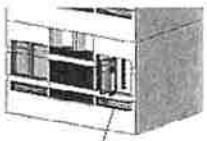
Step One – Allow more air in







Casement window design as "Wind scoop" to increase the wind speed intake into the units



Lower window opening allows wind enter from the lower level to create better ventilation at human level

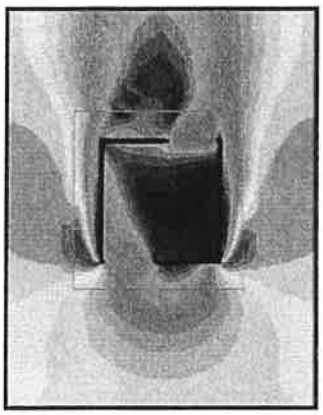


Sliding window

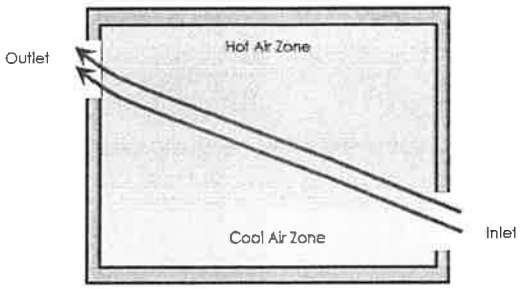
Case studies

Step Two – Allow more air out

Air movement principle – cross ventilation



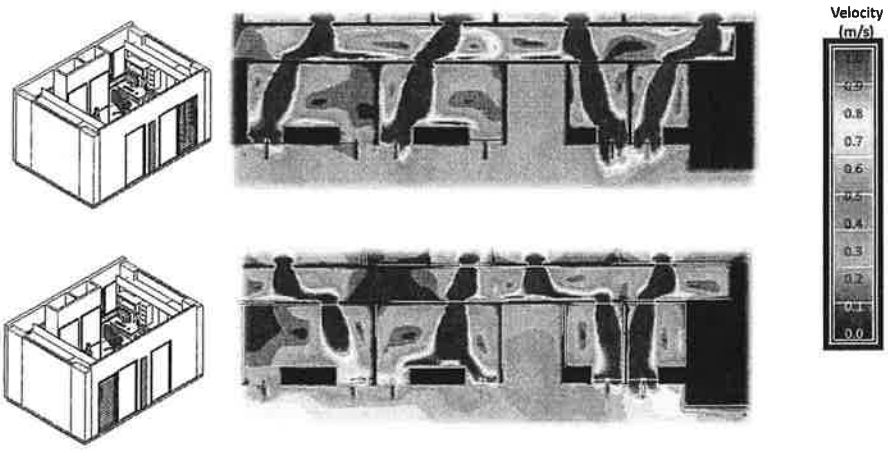
Air movement principle – hot air rises



Case studies

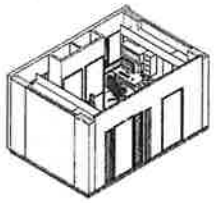
Step Two – Allow more air out

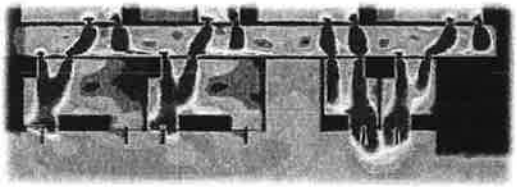
Casement window and 1 large louver on either side of UGD



Case studies

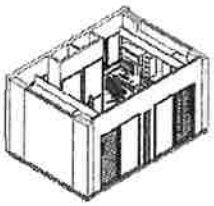
Step Two – Allow more air out







Casement window and 1 set of small double louvers on door

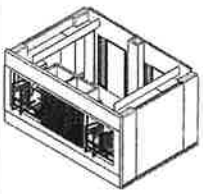


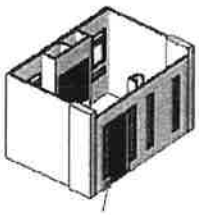


Casement window and One set of large louver by each door. Total 2 sets of louvers

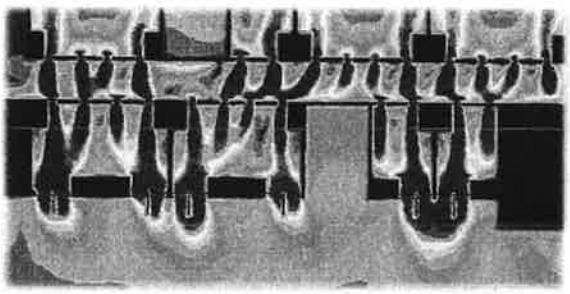
Case studies

Step Two – Allow more air out





Louver design to allow cross ventilation



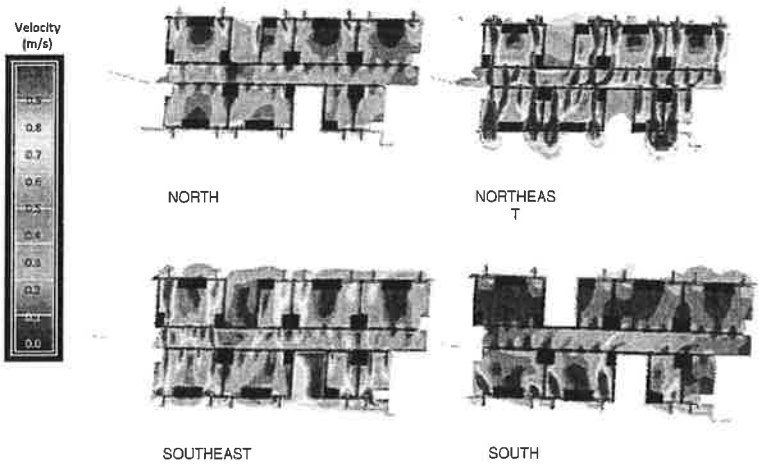


Casement window with 2 sets of double louvers

Case studies

Natural Ventilation Results

The Computer Fluid Dynamic (CFD) analysis shows the wind pattern and wind speed inside the units in 4 directions.

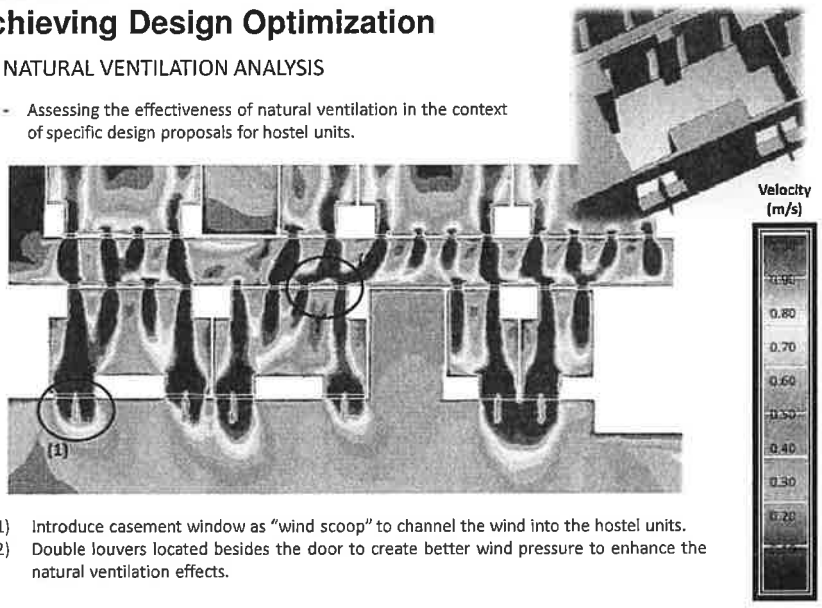


Case studies

Achieving Design Optimization

NATURAL VENTILATION ANALYSIS

- Assessing the effectiveness of natural ventilation in the context of specific design proposals for hostel units.



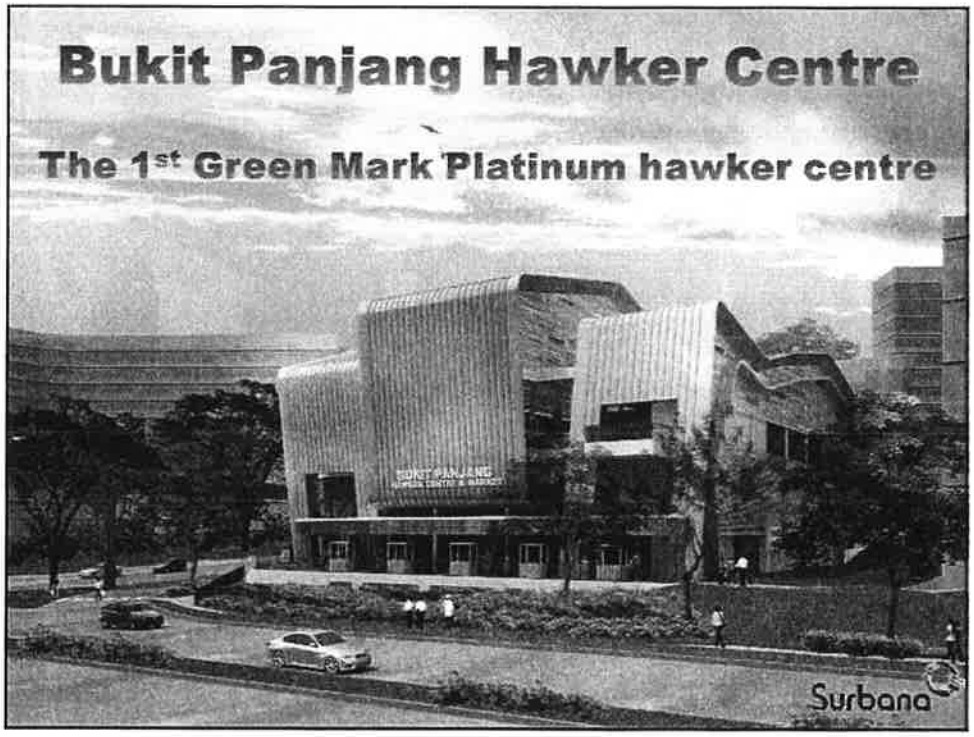
1) Introduce casement window as "wind scoop" to channel the wind into the hostel units.
2) Double louvers located besides the door to create better wind pressure to enhance the natural ventilation effects.

DESIGNING FOR NATURAL VENTILATION

Case 4

Bukit Panjang Hawker Centre

The 1st Green Mark Platinum hawker centre



Case studies

1st Storey Plan

28 nos. of Cooked Food Stalls

Comprising of :

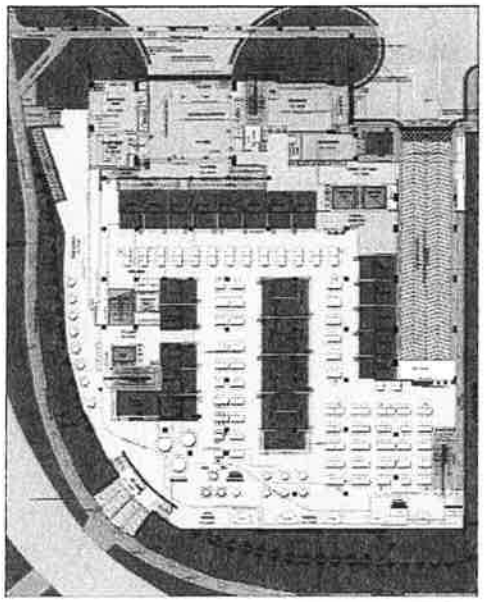
- 1 Drink Stall
- 1 Zi Char Stall
- 26 nos of cooked food Stalls

Seatings Capacity

- Tables 116 nos.
- Seating 489 nos.
- Handicapped seats 12 nos

Seating ratio to Stalls = 1:18.1

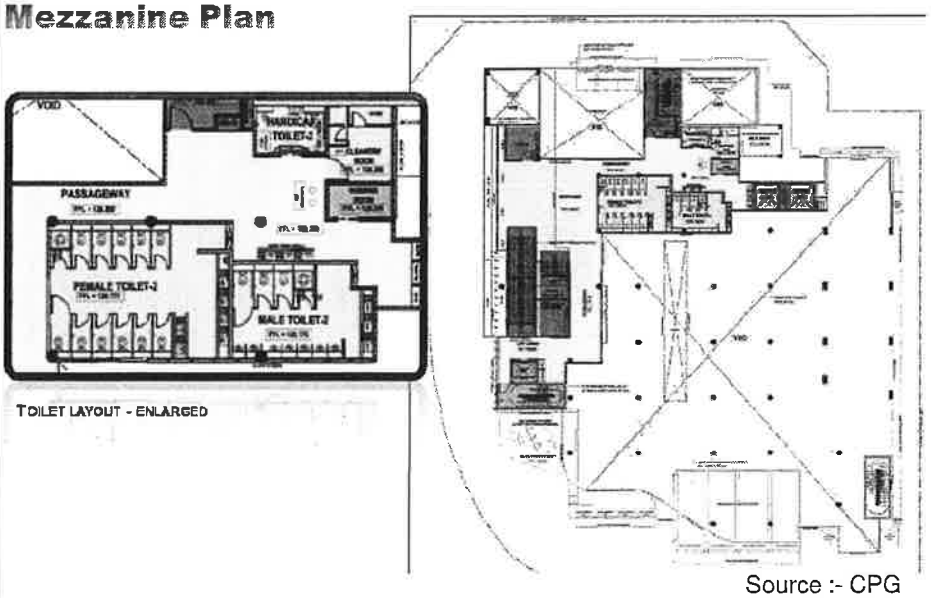
(Requirement = 1 :18)



Source :- CPG

Case studies

Mezzanine Plan



TOILET LAYOUT - ENLARGED

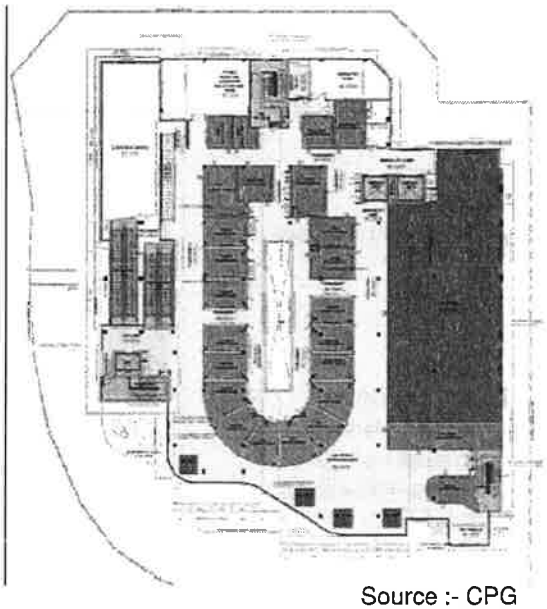
Source :- CPG

Case studies

2nd Storey Plan

Comprising of :

- 14 Market Slabs
- 11 Lock up Stalls
- 1 Fairprice
- 4 Kiosks



Source :- CPG

Case studies

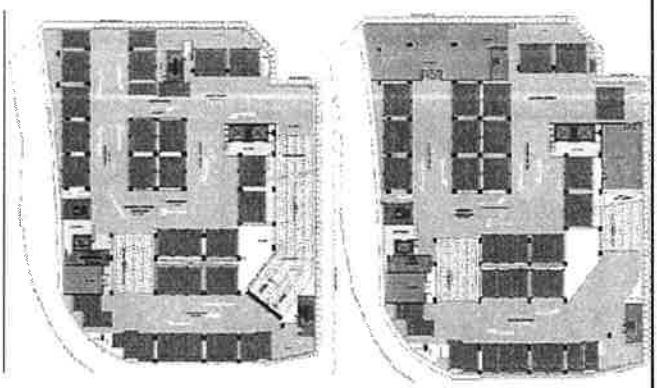
Basement 1 & 2

Basement 1 Parking Lots :

- 49 Car Parking Lots
- 1 Handicapped Lot
- 4 Motorcycle Lots

Basement 2 Parking Lots :

- 49 Car Parking Lots
- 1 Handicapped Lot
- 5 Motorcycle Lots



Total Parking Lots Tabulation :

- 98 Car Parking Lots
- 2 Handicapped Lots
- 9 Motorcycle Lots

Source :- CPG

Case studies

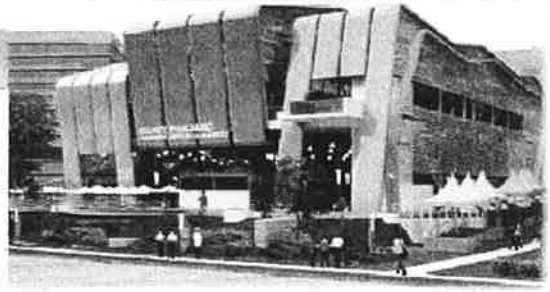




Bird-Eye Perspectives

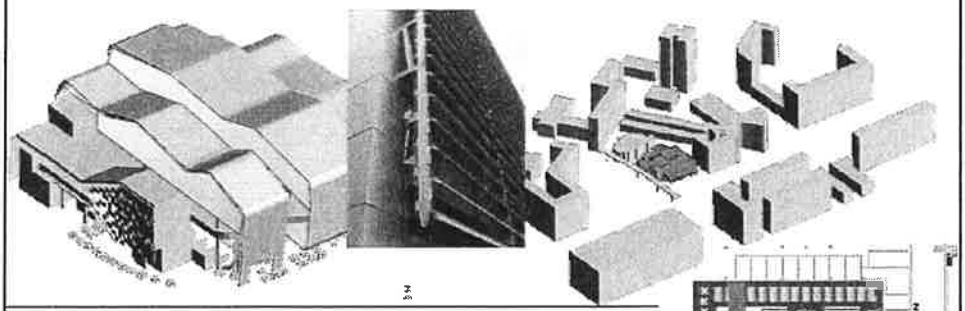
Isometric Perspectives





Source :- CPG

Case studies



Original

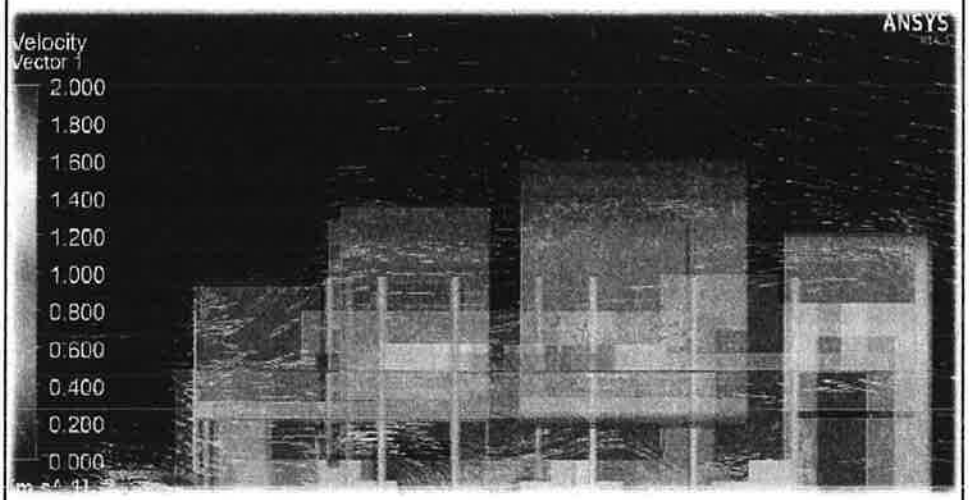
Unit type	Area (m^2)	North-Vavg (m/s)	South-Vavg (m/s)
1st Storey	852.521	0.237	0.352
2nd Storey	316.282	0.225	0.207

After adding operable lourves

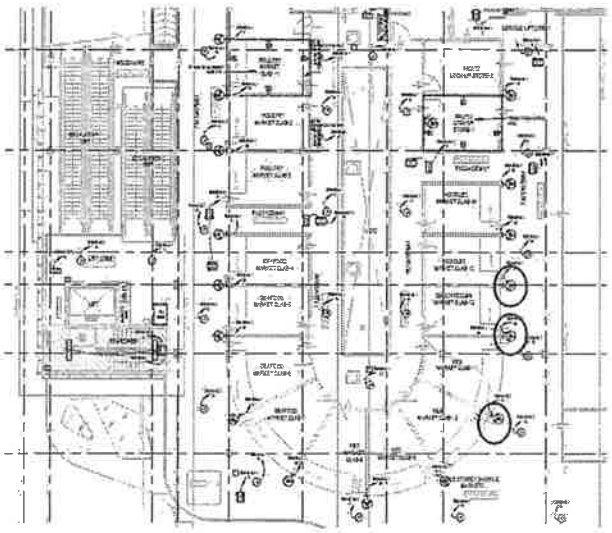
Unit type	Area (m^2)	North-Vavg (m/s)	South-Vavg (m/s)
1st Storey	852.521	0.615	0.706
2nd Storey	316.282	0.364	0.237

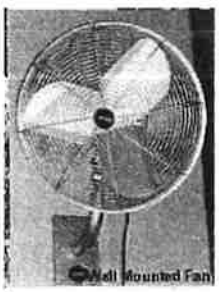
Case studies

Natural ventilation through adjustable louvers North



Case studies





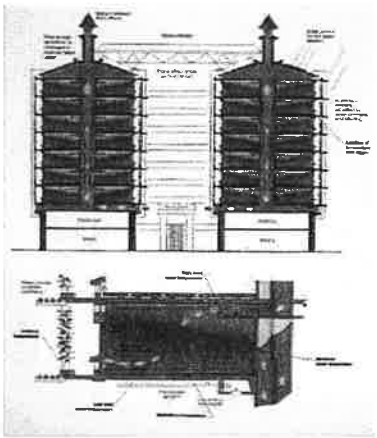
Assisted Ventilation

DESIGNING FOR NATURAL VENTILATION

Case 5

East Gate , Zimbabwe





https://biomimicron.wordpress.com/2012/12/11/biomythology-eastgate-gate-or-how-the-eastgate-centre-harare-is-not-like-a-termite-mound/



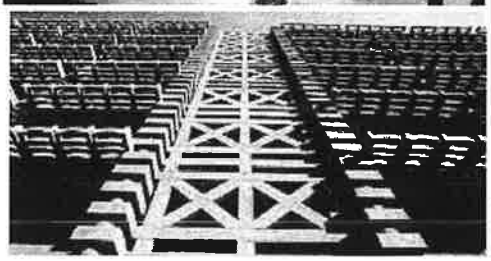
Architect Mick Pearce and engineering firm Arup borrowed the idea of a termite nest to build Eastgate Centre, a large office and shopping center in Zimbabwe that is cooled with the outside air.

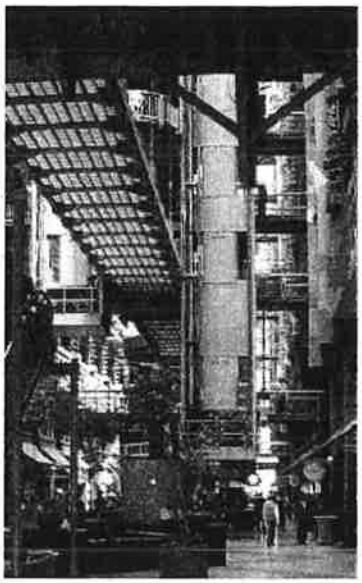
The system uses only 10 percent as much energy as conventional air-conditioning to drive fans that keep the air circulating.

Eastgate mall

East Gate , Zimbabwe







Source Pic :
https://doctorcrowd.wordpress.com/2012/10/20/aluminum-casts-of-ant-colonies/ Eastgate mall

DESIGNING FOR NATURAL VENTILATION

Case 5

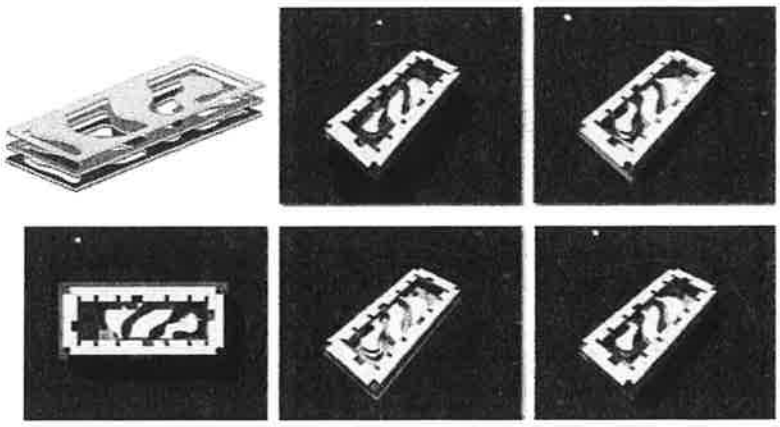
Case studies

Academy of Fashion
Jaipur

Manit Rastogi , Morphogenesis

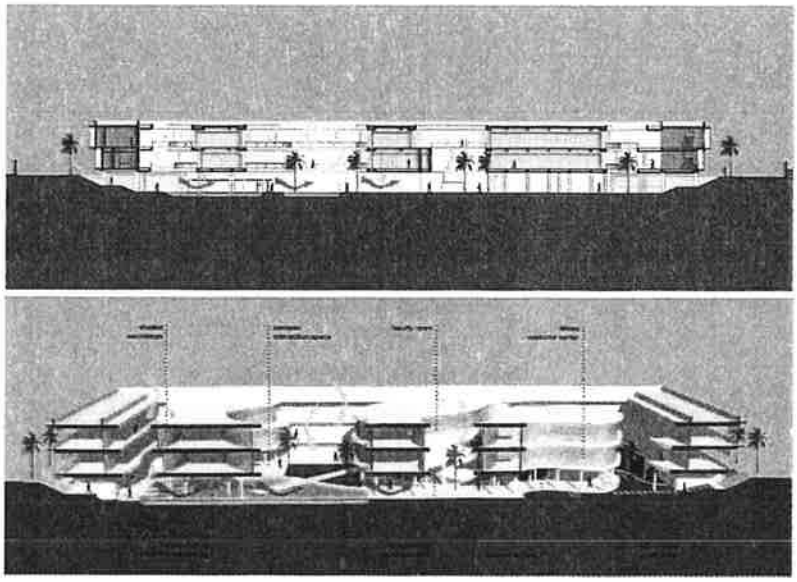
Case studies

Shadow Studies



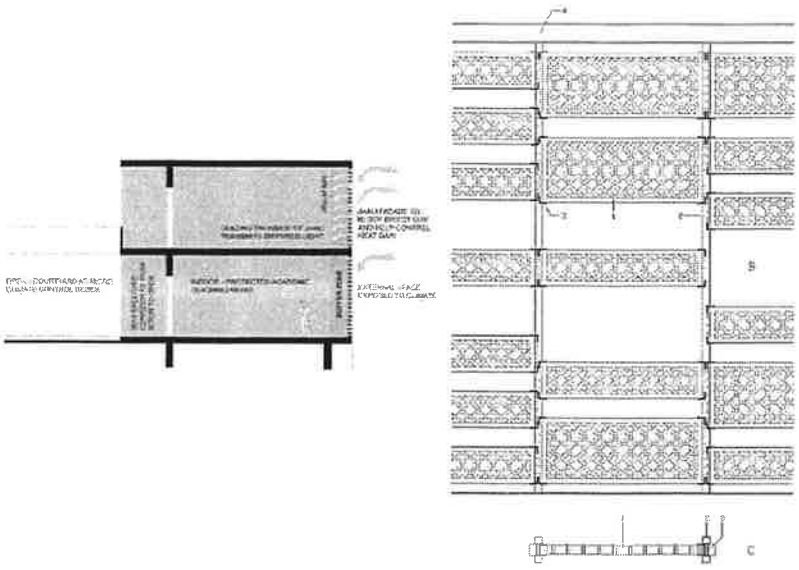
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



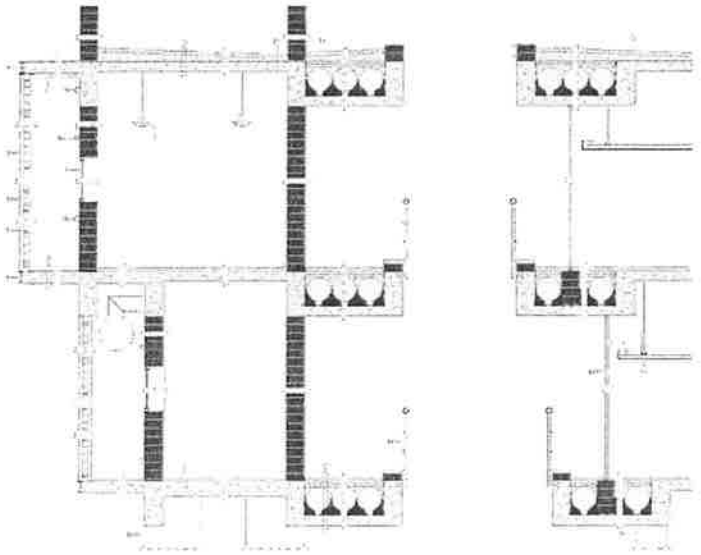
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



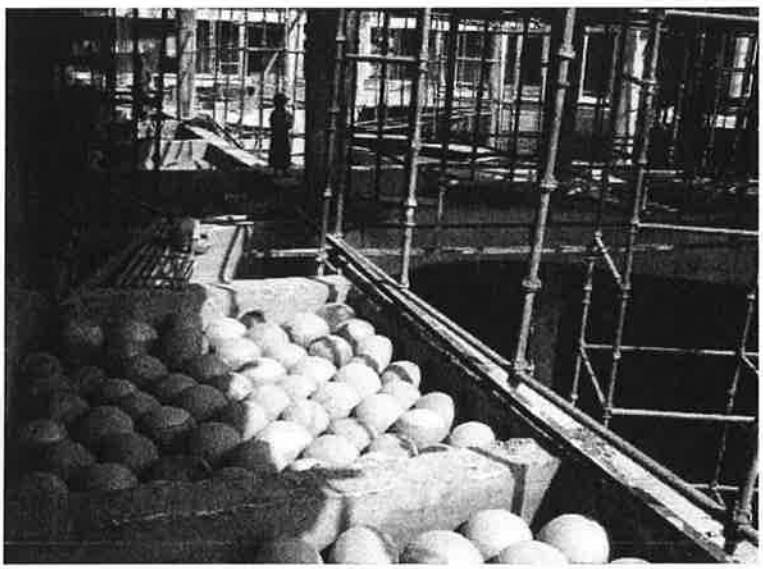
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



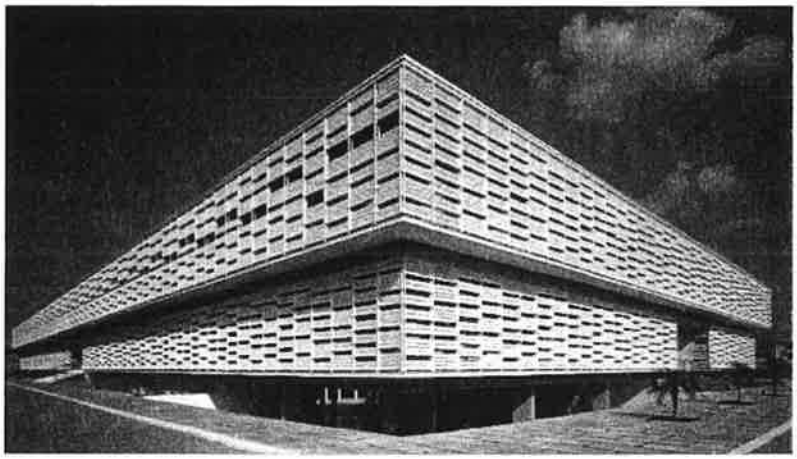
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



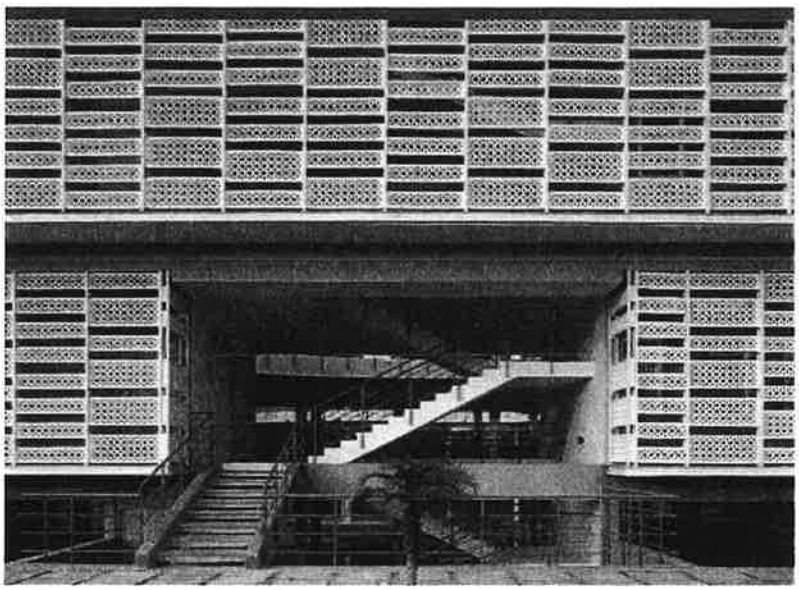
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



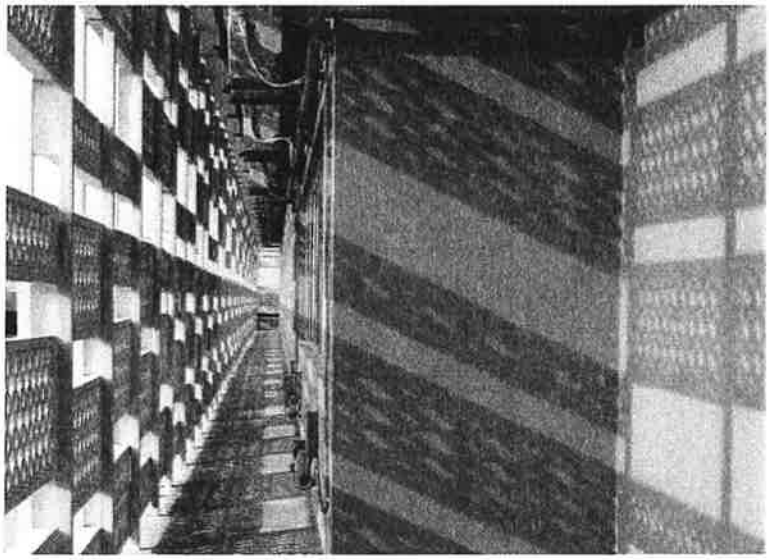
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



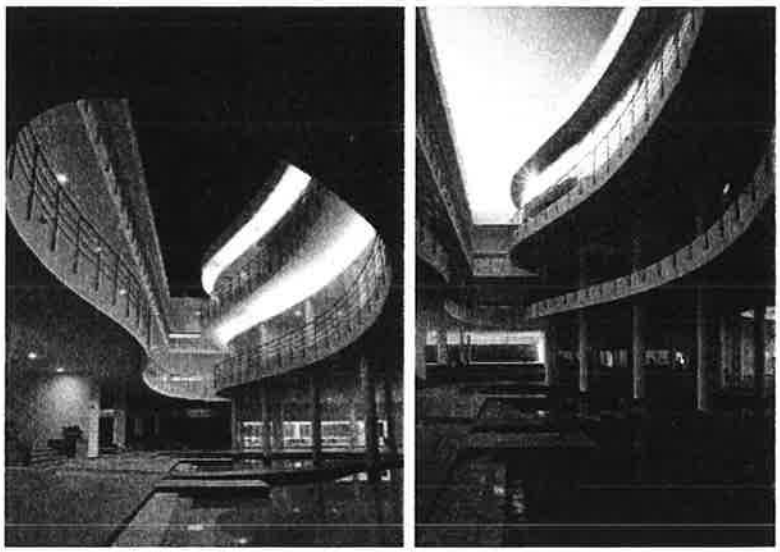
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Fashion School

Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

DESIGNING FOR NATURAL VENTILATION

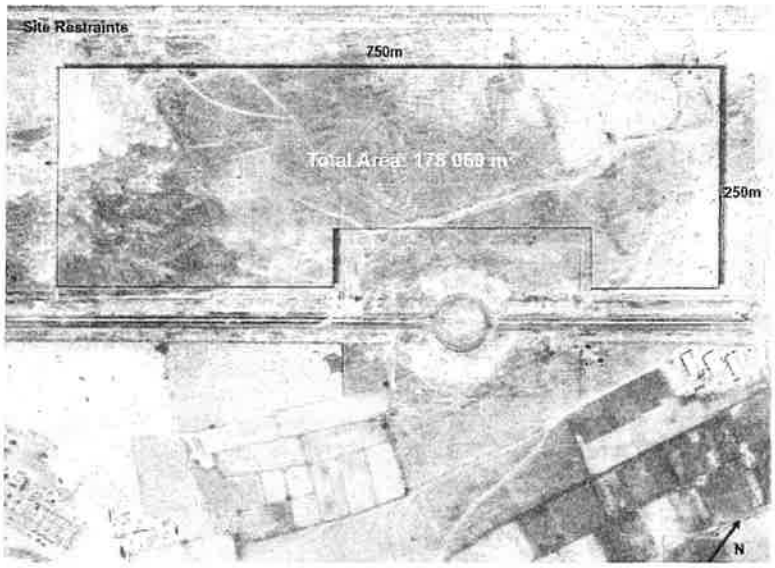
Case 6

Case studies

Educomp University
Greater Noida

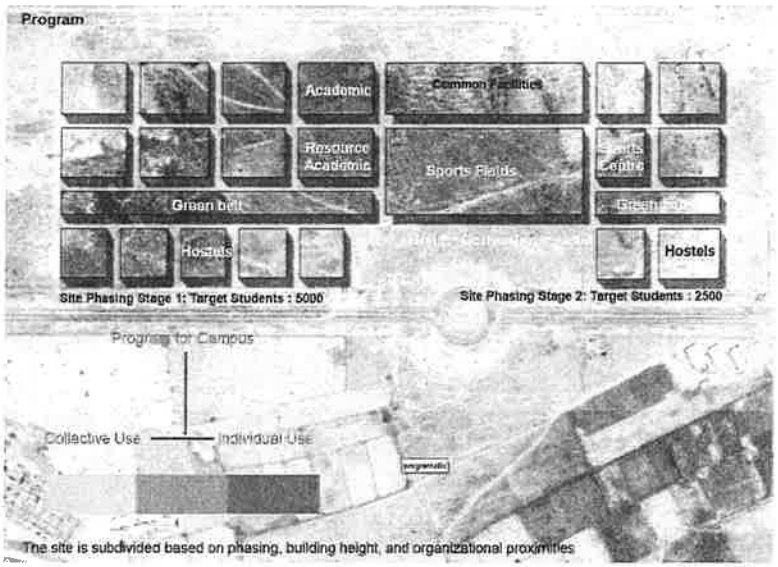
Manit Rastogi , Morphogenesis

Case studies



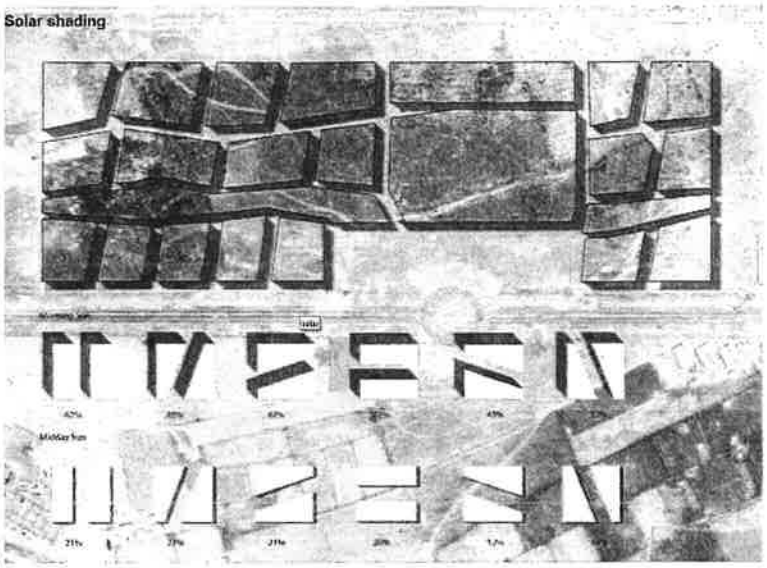
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



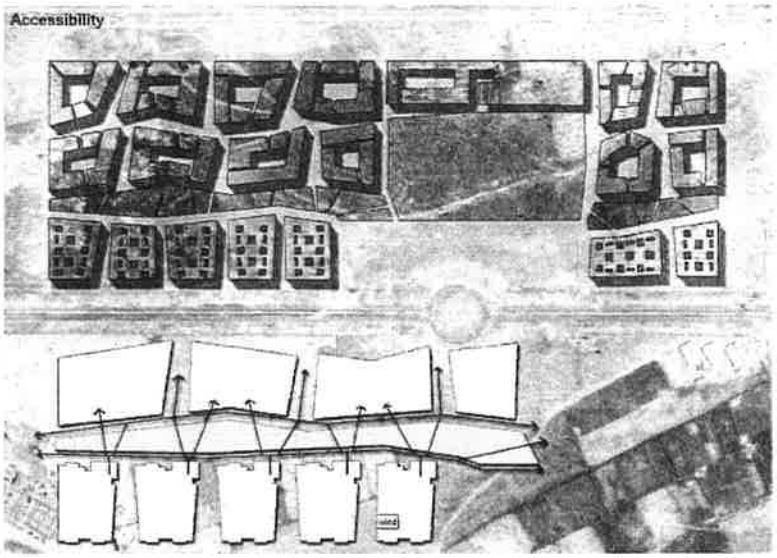
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



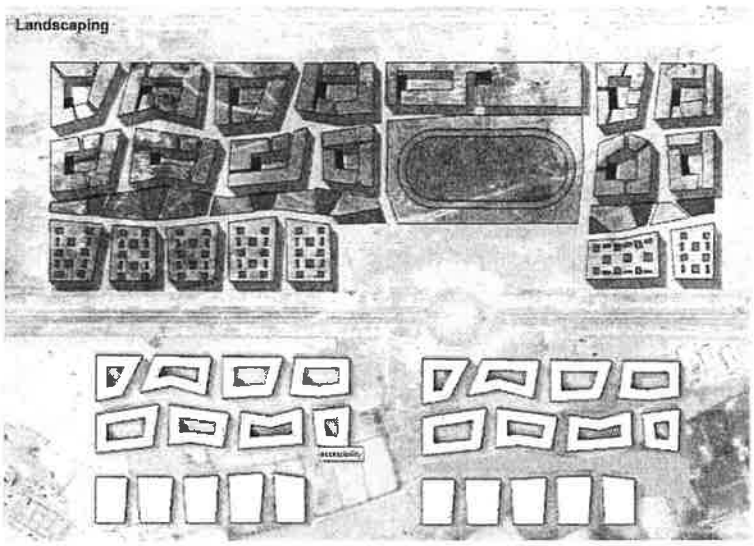
Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

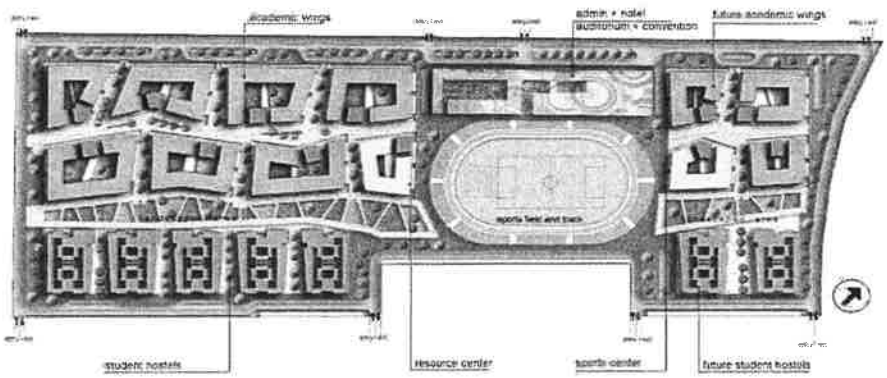
Case studies



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

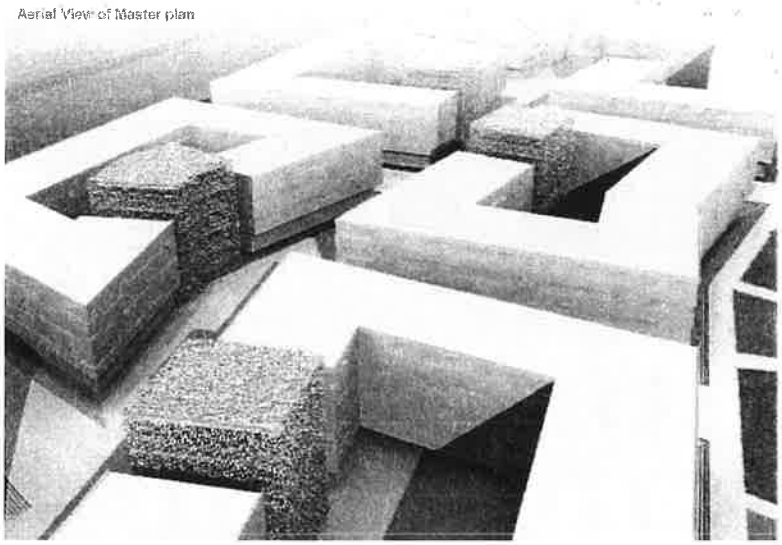
Master Plan



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

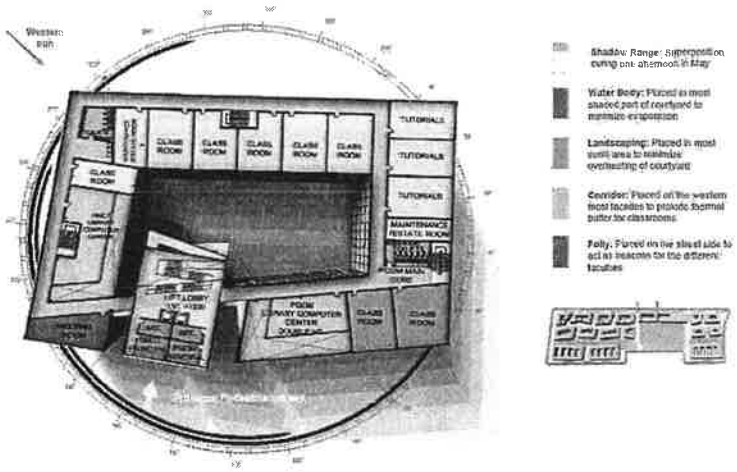
Aerial View of Master plan



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

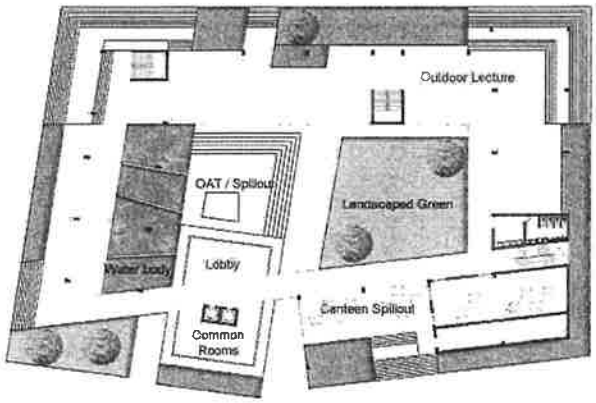
Functional layout of Block



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

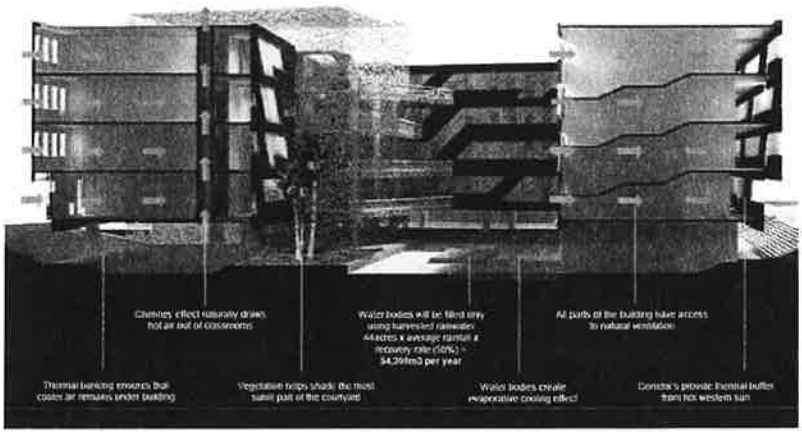
Academic Building 1: Plan at Underbelly Level -2500



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

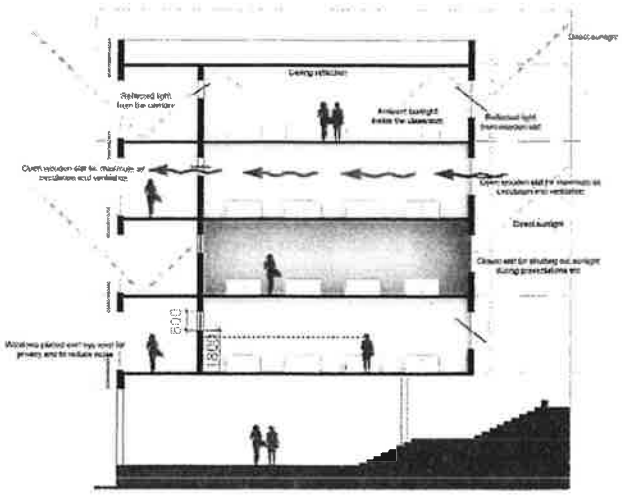
Functional section of Block/Underbelly



Source : Manit Rastogi , Morphogenesis , 2009 IGBC

Case studies

Classroom - section

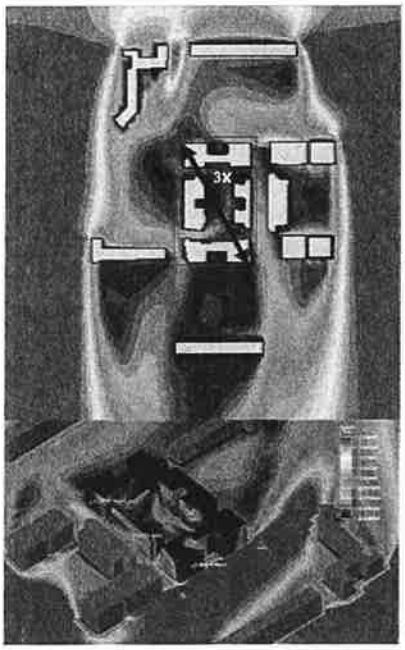


Source : Manit Rastogi , Morphogenesis , 2009 IGBC

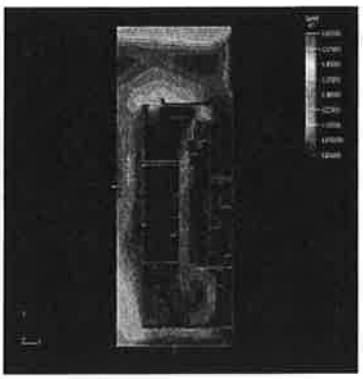
DESIGNING FOR NATURAL VENTILATION

Case 7

Case studies

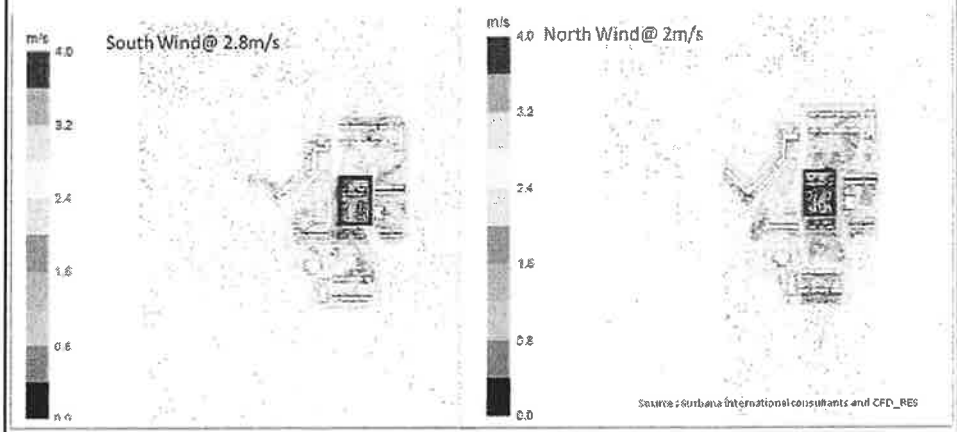


The project is a high density residential condo with full facilities targeted at mid market. The target was to achieve the highest Green Mark Platinum Award.



Source : Surbana International Consultants

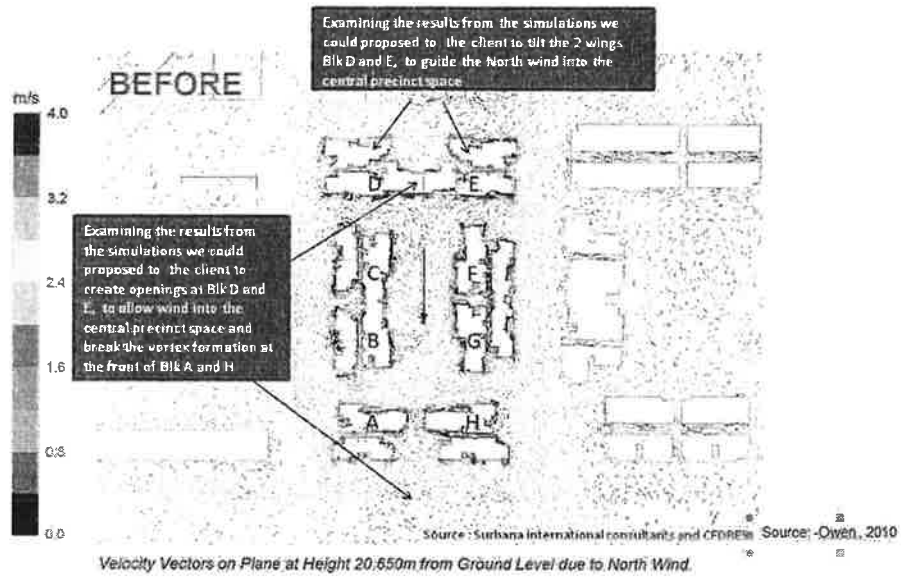
Case studies



Testing and design

We have to return to examine the wind flow at the Precinct level and seek to improve the wind speeds in our internal precinct space, as the internal precinct space has been enclosed leaving very little wind to get into the space due to the need to maximize efficiency and saleable area.

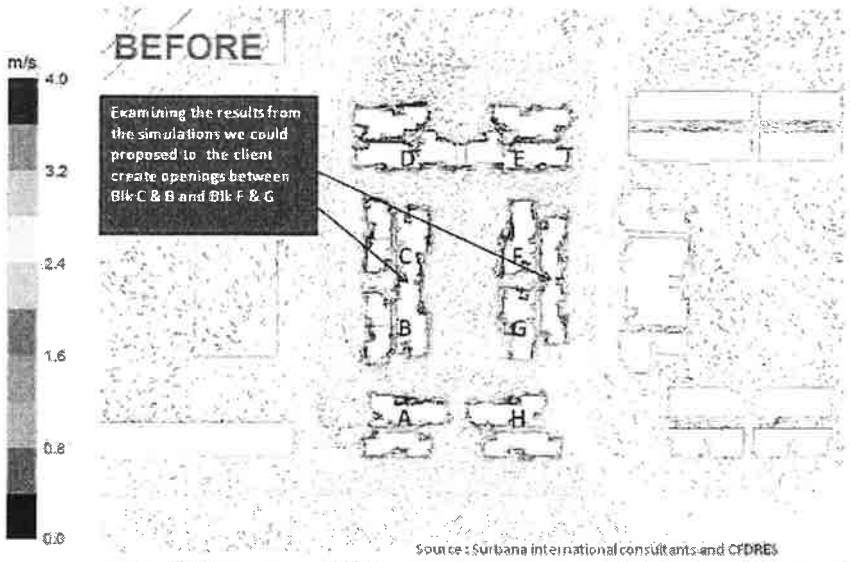
Case studies



Source: -Owen, 2010

Velocity Vectors on Plane at Height 20.650m from Ground Level due to North Wind.

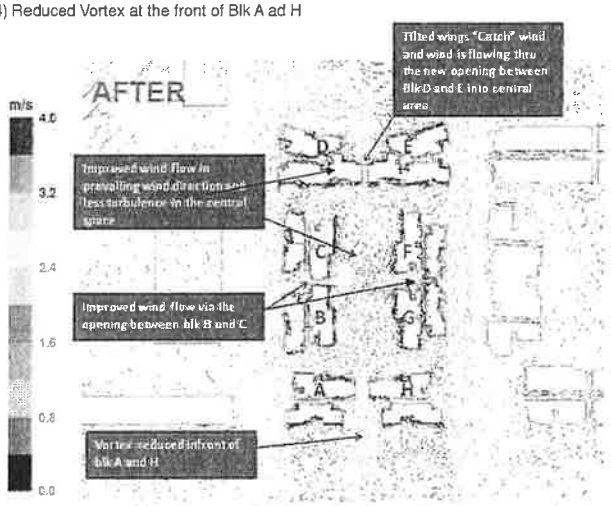
Case studies



Velocity Vectors on Plane at Height 20.650m from Ground Level due to South Wind. Source: -Owen, 2010

Case studies

4) Reduced Vortex at the front of Blk A ad H



Velocity Vectors on Plane at Height 20.650m from Ground Level due to North Wind.

Source: Surbana international consultants and CFDRE Source: -Owen., 2010

Improvement to the site plan

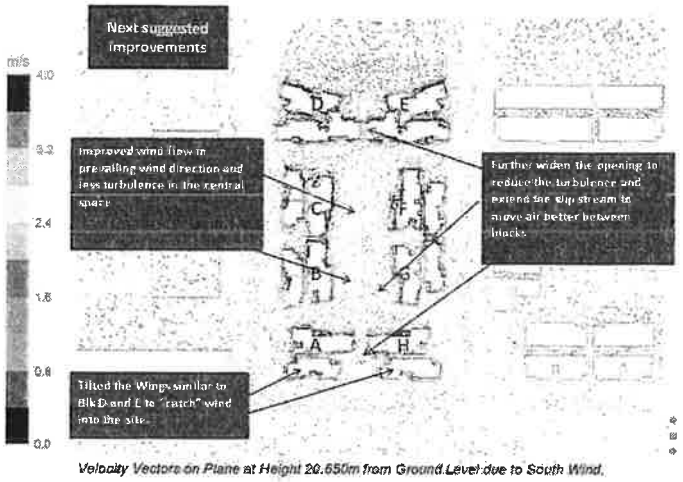
1) Tilted wings "Catch" wind and is flowing thru the new opening between Blk D and E into central area
2) Improved wind speed for entire development and less turbulence in the central space
Baseline Layout
3) Improved wind flows via the openings between Blk B and C as well as Blk F and G
4) Reduce Vortex in A & H

Wind speed Before and After

	Before	After	Improvement
North Wind	0.6984m/s	0.7525m/s	7%
South Wind	1.1085m/s	1.1595m/s	4%

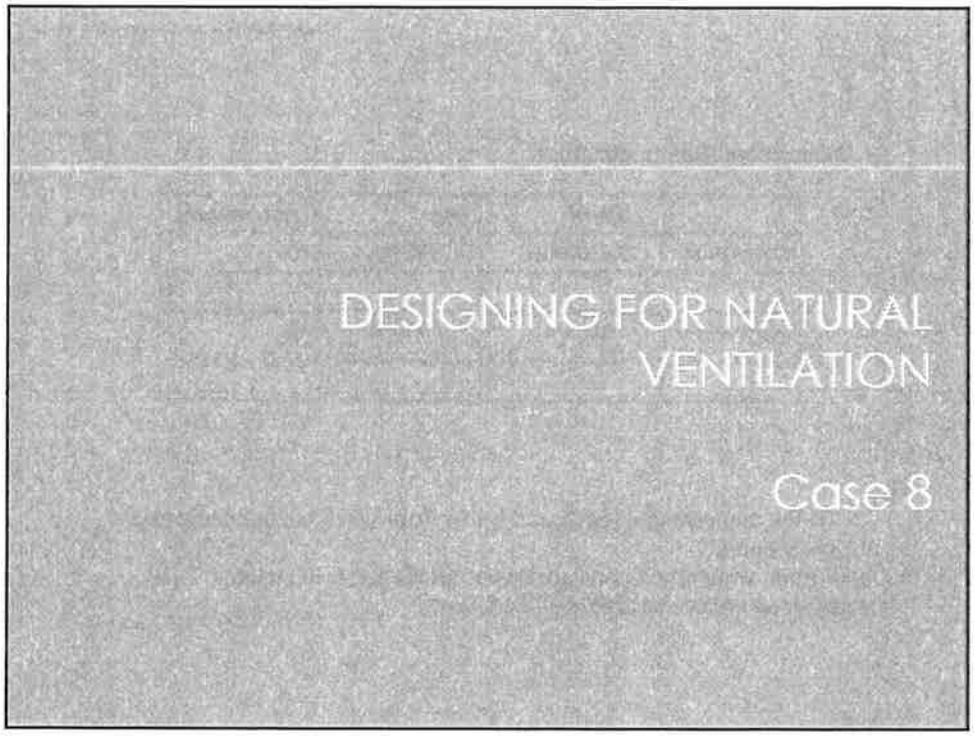
1) Tilt the south wing of Blk A and H to induce wind into the site similar to Blk D and E
2) Further widen the opening between Blk D and E to reduce turbulence on the north side of Blk D and E

Case studies



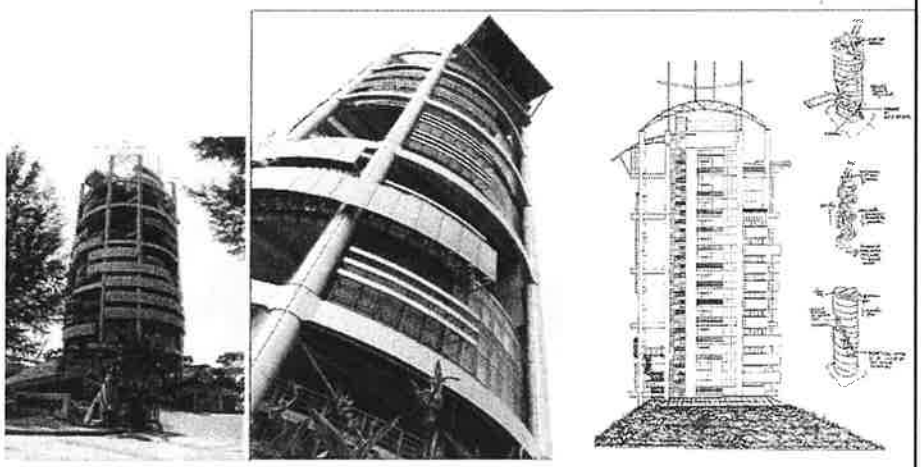
Velocity Vectors on Plane at Height 20.650m from Ground Level due to South Wind.

1) Tilt the south wing of Blk A and H to induce wind into the site similar to Blk D and E
2) Further widen the opening between Blk D and E to reduce turbulence on the north side of Blk D and E



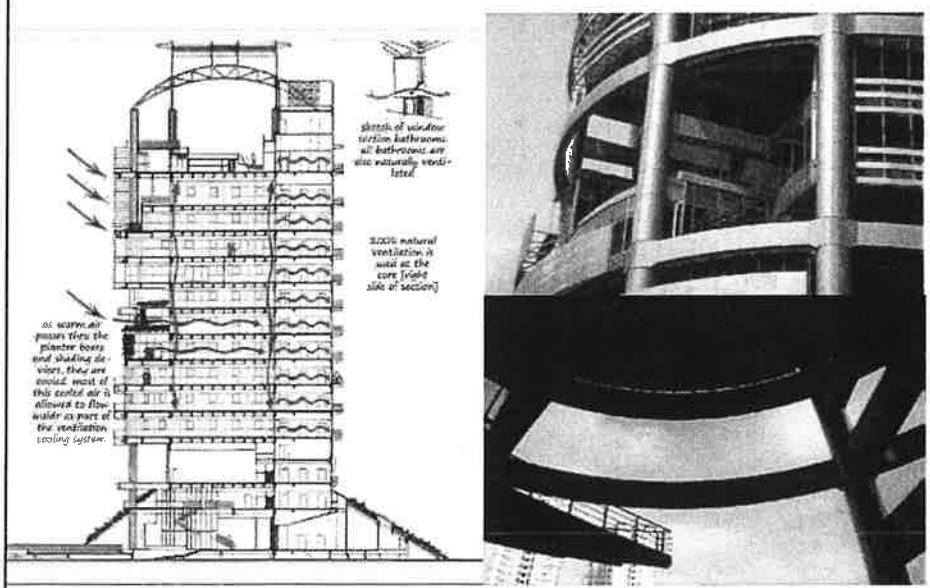
Case studies

Adapting Modern Architecture to the Tropics



Kenneth Yeang ,Menara Mesin, KL

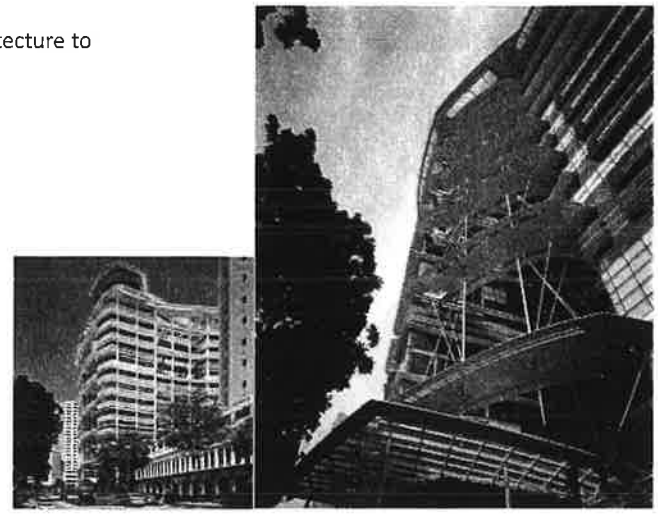
Case studies



Source http://www.slideshare.net/nuradilazaas/menara-mesiniaga-building-science-report

Case studies

Adapting Modern Architecture to the Tropics



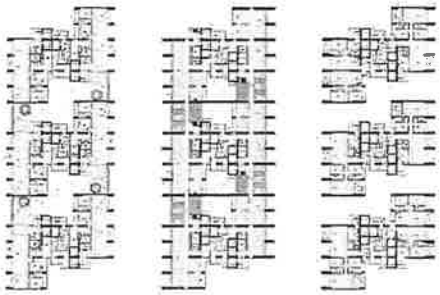
Kenneth Yeang , Singapore National Library

DESIGNING FOR NATURAL VENTILATION

Case 9

Case studies

adapts aspects of low-rise tropical housing to spaces in the sky.



THE MET (Bangkok)
WOHA



Source: www. Aga Khan Award for Architecture

BCA GREEN MARK

BCA Green Mark for New Buildings (Non-Residential)

2015

Effective 1 Dec 2016

Non – Residential Version 2015

Minimum Criteria Points Requirements

Indicator	Pre-Requisite Requirement	Minimum Points Requirement		
		Gold	Gold^{PLUS}	Platinum
1.01a	Climatic & Contextually Responsive Brief	-	1 pt	
1.01b	Integrative Design Process	-	2 pts	
1.01e	User Engagement	Green Fit-out Guideline - 1 pt		
1.02b	Integrated Landscape and Waterscape	-	• Commercial – 2 pts • Others – 1 pt	
1.03c	Ventilation Performance – Ventilation Simulation	-	For Gold^{PLUS} and Platinum projects with ≥ 2,000m^3 of naturally ventilated occupied spaces, the following wind speeds must be met via ventilation simulation: • Gold^{PLUS} – 0.4 m/s; • Platinum – 0.6 m/s *Complementary methods to compliance are available as described in 1.03c	
2.03a	Solar Energy Feasibility Study	For buildings with a footprint ≥ 1,000m^2 – 0.5 pt		
3.02a	Sustainable Construction	0.5 pt	2 pts	2.5 pts
3.02b	Embodied Energy	-	1 pt	
3.02c	Sustainable Products	2 pts	3 pts	4 pts
4.01a(i)	Indoor Air Quality Audit	0.5 pt		
4.01a(ii)	Post Occupancy Evaluation	-	0.5 pt	
4.01b(ii)	Enhanced Filtration Media	-	0.5 pt	
4.03d	System Handover and Documentation	-	1 pt	

Passive Design & Natural Ventilation, GMP 2017

Non – Residential Version 2015

P.02 Air Tightness and Leakage

Intent

Minimising air infiltration through the building envelope can reduce the energy required for air-conditioning and enhance occupant thermal comfort.

Scope

Applicable to all windows and curtain walls on the building envelope.

Assessment

For windows and curtain wall systems, air leakage rates shall not exceed the limit specified in *SS 212: 2007 – Specification for Aluminium Alloy Windows* and *SS 381: 1996 (2007) – Materials and Performance Tests for Aluminium Curtain Walls* respectively.

Source BCA, GreenMark Version 2015

Non – Residential Version 2015

1.03b Internal Spatial Organisation

Intent

Adopting passive design strategies in the internal spatial organisation of a building provides opportunities to enhance building performance

Scope

Applicable to all building developments.

Assessment

Points can be scored for the following:

- Locating non-thermally critical non-air-conditioned spaces, e.g. lift cores, staircases, toilets, electrical plantrooms etc on east and west facing walls to reduce thermal heat gain into occupied spaces – 1 point
- Prorating the number of transient common spaces, e.g. toilets, staircases, corridors, lift lobbies and atriums by the mode of ventilation against the total number of applicable spaces – Up to 2 points

[illegible]	Air-Conditioned	Mechanical Ventilation	Naturally Ventilated
Points	0	0.5	2

Source BCA, GreenMark Version 2015

Non – Residential Version 2015

1.03c Ventilation Performance

Intent

Naturally ventilated functional areas should be effectively designed to be thermally comfortable and healthy for the building occupants.

Scope

Applicable for naturally ventilated occupied spaces and gathering spaces such as building atria.

Assessment

A maximum 4 points can be scored for this sub-indicator based on the following options:

Ventilation Performance Checklist

Parameter	Description	Points
Openings towards prevailing wind directions	0.1 point for every 10% of units or rooms with openings facing towards the prevailing winds (North & South)	1
Depth of room vs opening *W: Limiting depth for effective ventilation* *H: Floor-to-ceiling height*	• **Single sided ventilation: W ≤ H** • **Cross Ventilation: W ≤ 5H** • **Combination with stack effect atria: W ≤ 15m from façade to atria.**	**≥50% of applicable spaces meet - 1 pt or ≥70% of applicable spaces meet - 2 pts**

Full Ventilation Simulation

Up to 4 points can be scored for ventilation simulations or wind tunnel testing conducted based on *Green Mark NRB: 2015 Technical Guide and Requirements Annex A: Ventilation Simulation Methodology and Requirements*. The simulation results and recommendations derived are to be implemented to ensure optimised natural ventilation. Where the wind speed result cannot be met, thermal comfort or air quality modeling should be performed and the relevant criteria stated in *Annex A* met. At least 70% of applicable naturally ventilated areas to meet:

Wind Driven Rain Simulation (Advanced Green Efforts)

1 point can be scored for wind driven rain simulation in compliance with *Annex A* to identify the most effective building design and layout that minimises the impact of wind-driven rain into naturally-ventilated occupied spaces.

Source BCA, GreenMark Version 2015

Non – Residential Version 2015

Full Ventilation Simulation

Up to 4 points can be scored for ventilation simulations or wind tunnel testing conducted based on *Green Mark NRB: 2015 Technical Guide and Requirements Annex A: Ventilation Simulation Methodology and Requirements*. The simulation results and recommendations derived are to be implemented to ensure optimised natural ventilation. Where the wind speed result cannot be met, thermal comfort or air quality modeling should be performed and the relevant criteria stated in *Annex A* met. At least 70% of applicable naturally ventilated areas to meet:

Points	Minimum weighted Average Wind Velocity	Thermal Comfort	Air Quality
3	Moderate (0.2m/s)	-	-
4	Good (0.4m/s)	-1.0 < PMV < +1.0	Air Change Rate ≥4 Air Exchange Efficiency ≥ 1
	Very Good (0.6m/s)	-0.8 < PMV < +0.8	Air Change Rate ≥10 Air Exchange Efficiency ≥ 1.2

Wind Driven Rain Simulation (Advanced Green Efforts)

1 point can be scored for wind driven rain simulation in compliance with *Annex A* to identify the most effective building design and layout that minimises the impact of wind-driven rain into naturally-ventilated occupied spaces.

Source BCA, GreenMark Version 2015

BCA GREEN MARK

BCA Green Mark for Residential Buildings Criteria

GM RB: 2016

For Pilot

Residential Version 2016 (Pilot)

3. To be eligible for Green Mark Platinum rating, it is a requirement to use ventilation simulation modeling and analysis to identify the most effective building design and layout. The simulation results and the recommendations derived are to be implemented to ensure good natural ventilation.

   - A minimum 70% of the selected typical dwelling units should have a weighted average wind velocity of 0.60 m/s;

   OR

   - A minimum 70% of selected typical dwelling units with "moderate" natural ventilation (with minimum weighted average wind velocity of 0.2m/s) and comply with the thermal comfort criteria;

   Other than dwelling units, common areas like staircases and lobbies (excluding those that are located in basement areas) should also be designed to be naturally ventilated (i.e. to provide open-able windows or other openings with aggregate area of not less than 5% of the space required to be ventilated).

Passive Design & Natural Ventilation, GMP 2017

Residential Version 2016 (Pilot)

1-03c Ventilation Performance

(i) Enhance dwelling unit indoor comfort through the provision of good natural ventilation design.

% of units with good natural ventilation	Points
≥70	3
≥50 and <70	2
≥30 and <50	1

Option 1 – Ventilation Simulation Modeling

Use of ventilation simulation modeling and analysis or wind tunnel testing to identify the most effective building design and layout to achieve good natural ventilation for all unit types.

Residential Version 2016 (Pilot)

Option 2 – Thermal Comfort Modelling (Fan+ Option)

Thermal comfort modelling shall be performed based on the following PMV equation and comply with the stated PMV range

PMV= -11.7853+0.4232T-0.57889V

PMV Range	PPD
-0.5<PMV<+0.5*	<10*

where PMV is Predicted Mean Vote
PPD is Predicted Percentage Dissatisfied
T is indoor air temperature (°C). Baseline of T is 29.5°C
V is indoor wind speed (m/s)

Prerequisite Requirement :
Green Mark Platinum –Minimum 70% of selected typical Dwelling units with good natural ventilation. Common areas are to be designed as naturally ventilated spaces.

Option 2a- Thermal Comfort Modelling (Fan+ Option)

Green Mark Platinum –Minimum 70% of selected typical Dwelling units with "moderate" natural ventilation (with minimum weighted average wind velocity of 0.2m/s) and comply with the thermal comfort criteria set out above. Common areas are to be designed as naturally ventilated spaces.

Residential Version 2016 (Pilot)

(ii) Effective building layout design and unit design reduce the needs of using air-conditioning.

Design for air flow within dwelling units

- Building layout design: Proper design of building layout that utilizes prevailing wind conditions to achieve adequate cross ventilation.
- Dwelling unit design: Good ventilation in indoor units through sufficient openings.

0.5 point for every 10% of units with window openings facing north and south directions

0.5 point for every 10% of living rooms and bedrooms designed with true cross ventilation

(up to 7 points)

Residential Version 2016 (Pilot)

The project team shall demonstrate through ventilation simulation modelling and analysis e.g computer fluid computation (CFD) to ensure that hot air can be effectively discharged and the declared efficiency of the air-conditioning system can achieved. Details for the housing of the condenser units such as clearance spaces and screens shall be considered.

Note:

For developments where air-conditioners are not provided, points will be scored and prorated under **1.03c (ii) Ventilation Performance**

Residential Version 2016 (Pilot)

4.01a Occupant Comfort

For design taking into account of non-prevailing wind and without the use of air-conditioner:

For living room only - 1 point
For all living room, bedrooms – 2 points

To encourage provision of assisted mechanism to achieve thermal comfort for occupant residential spaces

(Up to 2 points)

Residential Version 2016 (Pilot)
Advance green efforts

5.01a Passive Design Strategies

To encourage design that optimized prevailing wind conditions and facilities air flow such as

- For development with multiple blocks, stagger blocks such that blocks behind are able to receive wind penetrating through the gaps between the blocks in the front row or arrange building according to ascending height with lower height in front and towards the direction of prevailing wind
- Provision of either void decks at the ground floor or void spaces in between buildings to encourage air flow through and around buildings
- Carry out macro ventilation simulation to check block layout to ensure passive design been considered from the early design stage

Extent of Coverage : 80% of applicable areas

1 point for each strategy
(Up to 3 points)

Passive Design & Natural Ventilation, GMP 2017

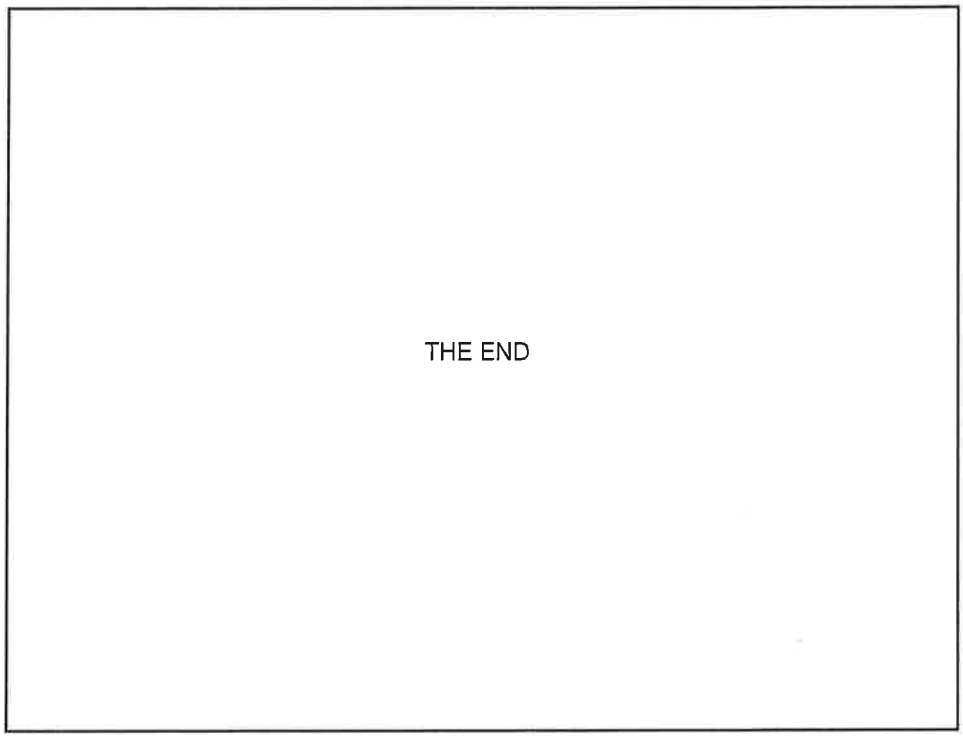
Residential Version 2016 (Pilot)
Advance green efforts

5.01b Wind Driven Rain Simulation

To encourage design that use wind driven rain simulation modelling to identify the most effective building design and layout that minimize the impact of wind-driven rain into naturally ventilated common areas such as lift lobbies and corridors, drop-off area and communal space such as sky garden.

For naturally-ventilated common areas such as lift lobbies and corridors
2 points - No noticeable penetration of WDR
1 point – some but acceptable degree of penetration of WDR

For drop-off area and communal space such as sky garden
1 point - No noticeable penetration of WDR
0.5 point – some but acceptable degree of penetration of WDR


THE END

BCA GMP Core Module Design and Simulation for Natural Ventilation

Dr. Poh Hee Joo (pohhj@ihpc.a-star.edu.sg)
Dr. Xu Xiangguo, George (xu_xiangguo@ihpc.a-star.edu.sg)

Institute of High Performance Computing, A*STAR

April 2017

Course Objectives

- To equip GMP students with the basic skill set of applying CFD tool to building ventilation design
  - Creating 3D geometry model
  - Meshing the computational model
  - Applying appropriate boundary condition and numerical settings
  - Analyzing CFD result for design and improvement
  - Establishing result verification and validation

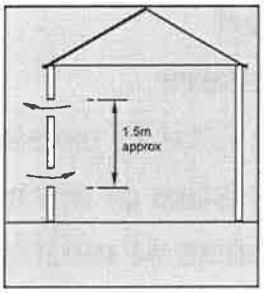
Day 2 (9:00am – 10:30am)

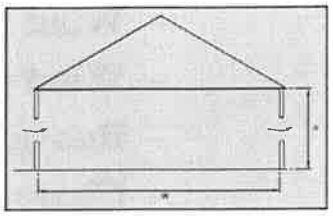
- Introduction to CFD in Green Building Application
  - Review about natural ventilation design
  - CFD-aided natural ventilation design of green buildings, and case studies

Review about Natural Ventilation Design

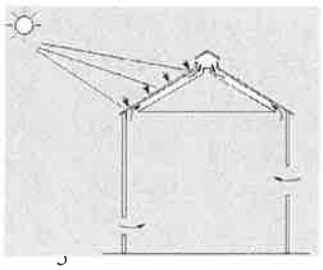
Natural Ventilation Methods

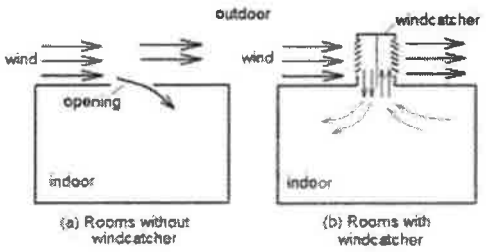
- In *Natural Ventilation* the airflow is due to wind and buoyancy through cracks in the building envelope or purposely installed openings.

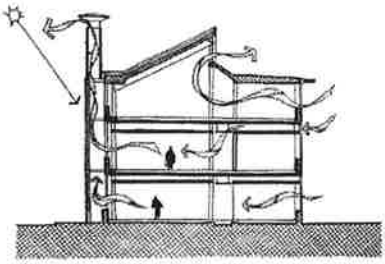




✓ *Single-Sided Ventilation:*
✓ *Cross-Ventilation:*
✓ *Stack Ventilation:*
✓ *Wind Catchers:*
✓ *Solar-Induced Ventilation:*







Natural Ventilation Methods/…

Hybrid Ventilation

In *Hybrid* (*Mixed Mode*) *Ventilation* the airflow is due to wind and buoyancy through purposely installed openings in the building envelope supplemented, when necessary, by mechanical systems.



The mechanical component of the hybrid system can be a fan for increasing the ventilation rate, and/or a heat exchanger for heating or cooling the outdoor supply air.



Natural Ventilation Parameters

Factors Influencing the airflow through openings

- **Wind speed**
- **Wind pressure**
- **Buoyancy (stack) pressure**
- **Characteristics of openings (C_d)**
- **Effective area of multiple openings**

Indoor Environmental considerations

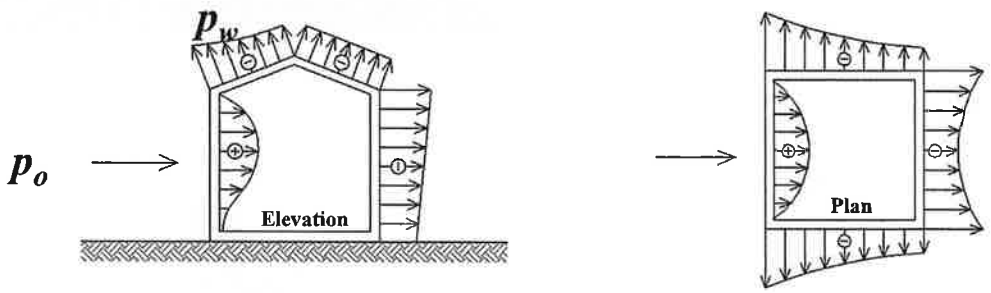
- **Thermal comfort**
- **Indoor air quality**

Natural Ventilation - Wind Pressure Coefficient

The static pressure coefficient (C_p) is defined as:

$$C_p = (p_w - p_o) / (\tfrac{1}{2} \rho V^2) \qquad (3)$$

where p_w **= static pressure at some point on the building (Pa)**
p_o **= static pressure of the free stream (Pa)**
ρ **= density of free stream (kg/m^3)**
V **= free stream velocity normally calculated at building height or other reference height (m/s)**



Wind pressure coefficients on a pitched roof building

8

Flow through an Opening

The flow through an opening can be calculated using the formula:

$$Q = A_{eff} [2 \Delta p / \rho]^{1/2} \qquad (6)$$

where Δp **is the pressure difference across the opening**

ρ **is the fluid density.**

The effective area (A_{eff}) is given by: $A_{eff} = C_d A$ **(7)**

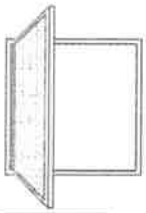
For natural ventilation openings the discharge coefficient (C_d) is not only dependent on type of opening and wind pressure but also on wind direction.

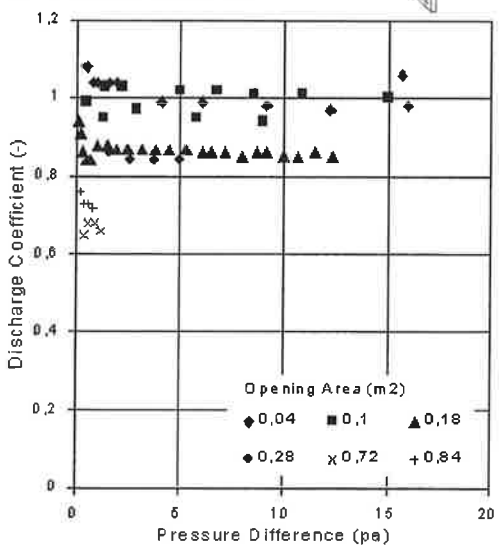
Data on C_d for natural ventilation openings is limited particularly its variation with wind direction. C_d is one of the main difficulties in manual calculation procedures and in network flow models for natural ventilation.

9

Flow through an Opening / …

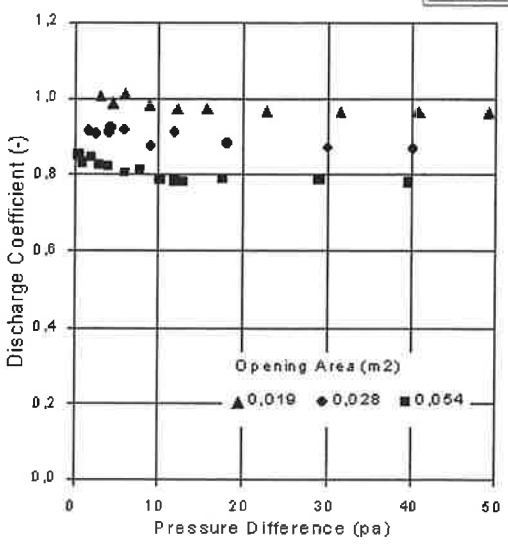
Discharge coefficient for side hung window





Discharge coefficient For bottom hung window

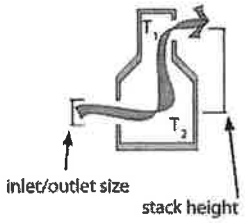


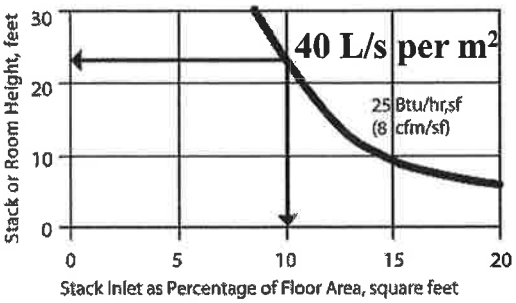


(Heiselberg *et al*, 2001)

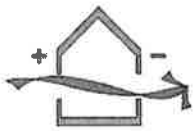
Simplified Flow Calculation for NV

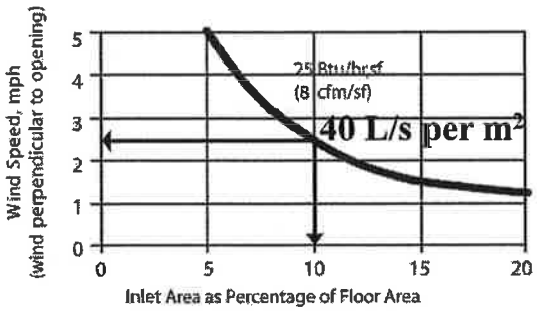
Stack





Cross-Flow



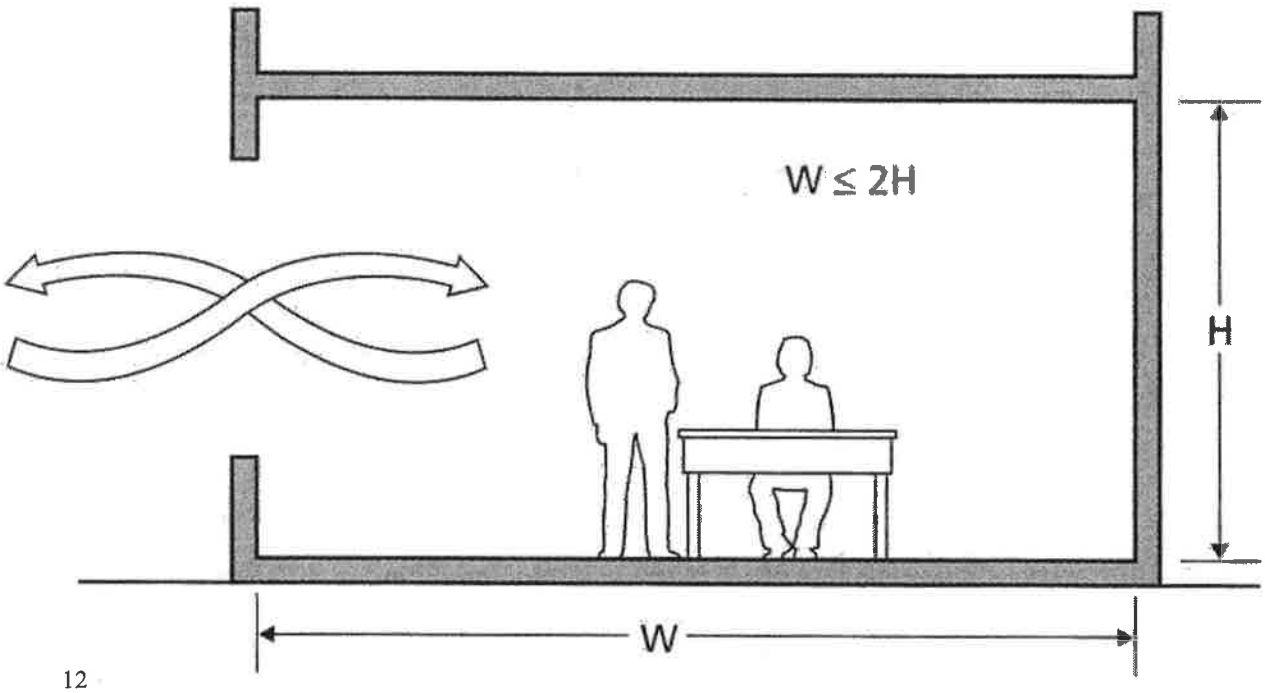


1 cfm/ft^2 = 5 L/s per m^2 1 mph = 0.45 m/s

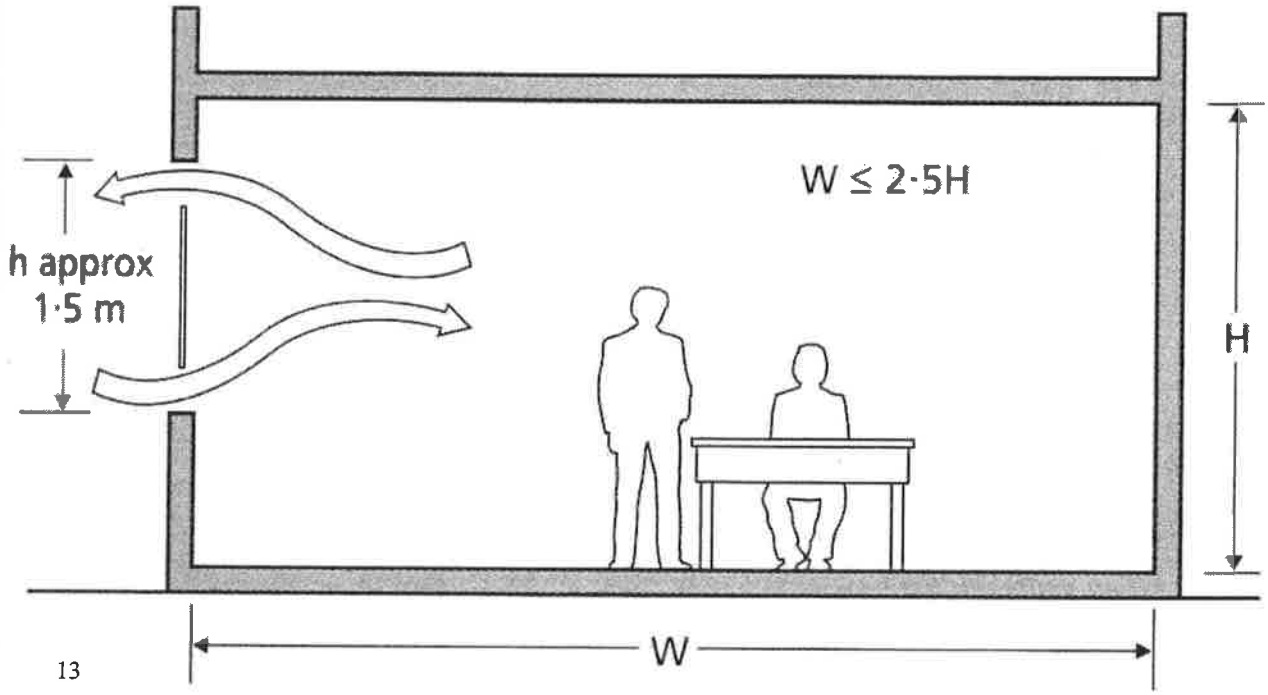
(Brown & Huang, 2006)

11

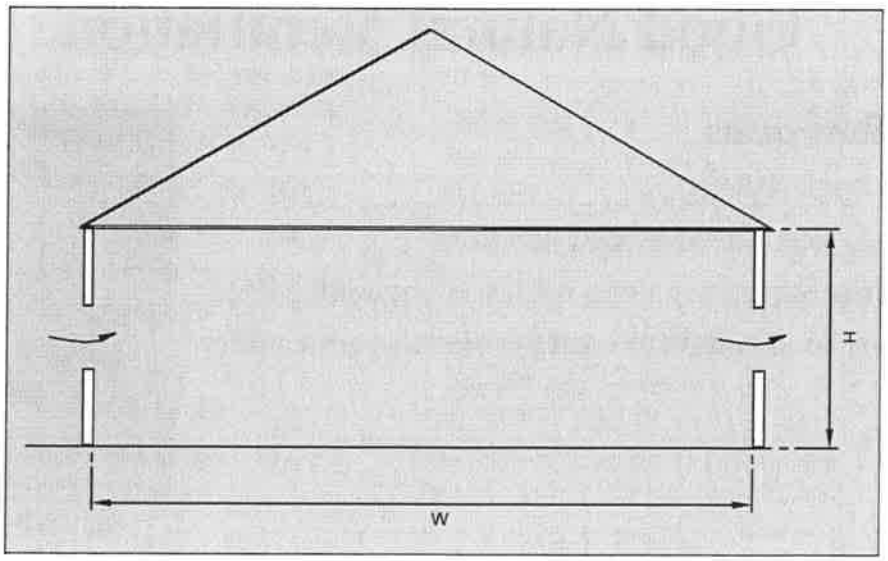
Single-sided Ventilation: Single opening



Single-sided Ventilation:
Double openings



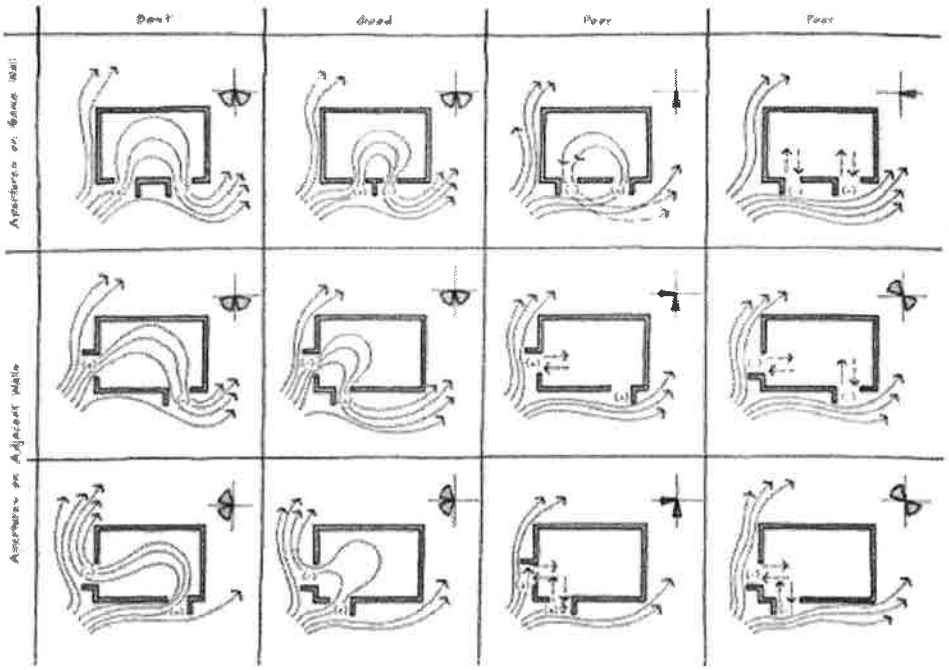
Cross-Ventilation:



The depth should be < 5 H

Hazim Awbi, Ventilation of Building, 2003

Wind Catchers



Different wing walls of better and worse effectiveness, on same wall and adjacent walls.

(Image from Sun, Wind, and Light, p. 184 by G.Z. Brown and Mark DeKay, published by Wiley)

Hazim Awbi, Ventilation of Building, 2003

15

Strategies At Estate Level To Achieve Good Natural Ventilation

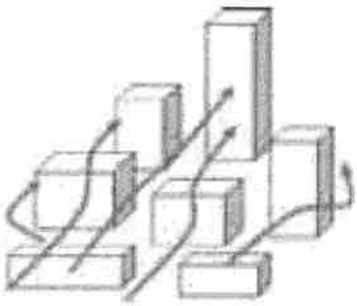
1. Create flow paths.
   - Link open spaces
   - Create open plazas at road junctions
   - Low-rise structures along routes of prevailing wind
   - Greater road widths to increase overall permeability



Reference : Building and Construction Authority, 2010, Building Planning and Massing

Strategies At Estate Level To Achieve Good Natural Ventilation

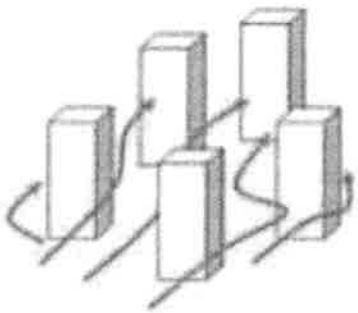
2. Arrange buildings according to ascending heights.
   - Lower rise buildings in front and towards direction of prevailing wind.



Reference : Building and Construction Authority, 2010, Building Planning and Massing

Strategies At Estate Level To Achieve Good Natural Ventilation

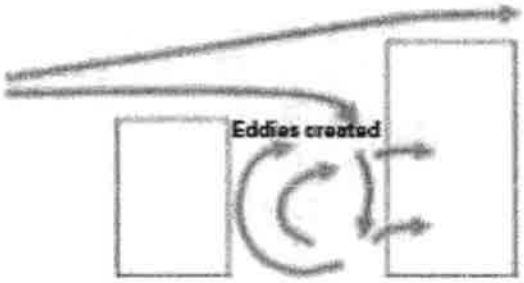
3. Stagger buildings.
   - Blocks behind should be able to receive the wind penetrating through the gaps between the blocks in the front row.



Reference : Building and Construction Authority, 2010, Building Planning and Massing

Strategies At Estate Level To Achieve Good Natural Ventilation

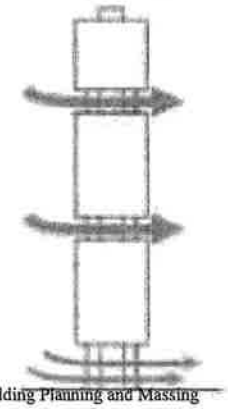
4. Create downwash wind.
   - Building design should consider capturing the downwash wind to reach the street level. Eddies may form in the canyon and this may assist in providing air movement within the canyon as well as into the buildings.



Reference : Building and Construction Authority, 2010, Building Planning and Massing

Strategies At Estate Level To Achieve Good Natural Ventilation

5. Improve building permeability.
   - Buildings should be as permeable as possible to channel airflow to the blocks in the back row. Sky gardens and double volume void decks can increase the permeability of blocks.



Reference : Building and Construction Authority, 2010, Building Planning and Massing

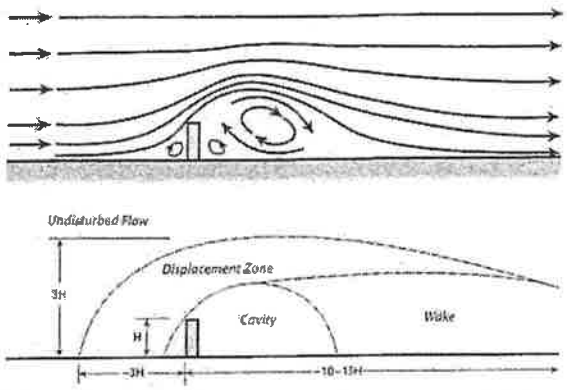
Urban Canopy Flow

- Urban Airflow – underlying principle
  - Wind near the ground: airflow and disturbance
  - Wind in the urban canopy
  - Wind modification in Urban Boundary layer
  - Effect of atmospheric stability on urban airflow
  - Urban scale wind

Reference : Urban microclimate : designing the spaces between buildings

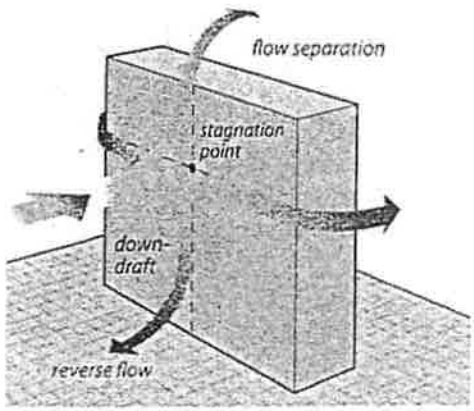
Urban Canopy Flow

- Wind near the ground – obstruction by isolated building



Schematic section showing typical pattern of wind flow over an isolated building

1. Wind approach the building (upwind)
2. Wind over the roof
3. Wind some distance behind (downwind/wake)

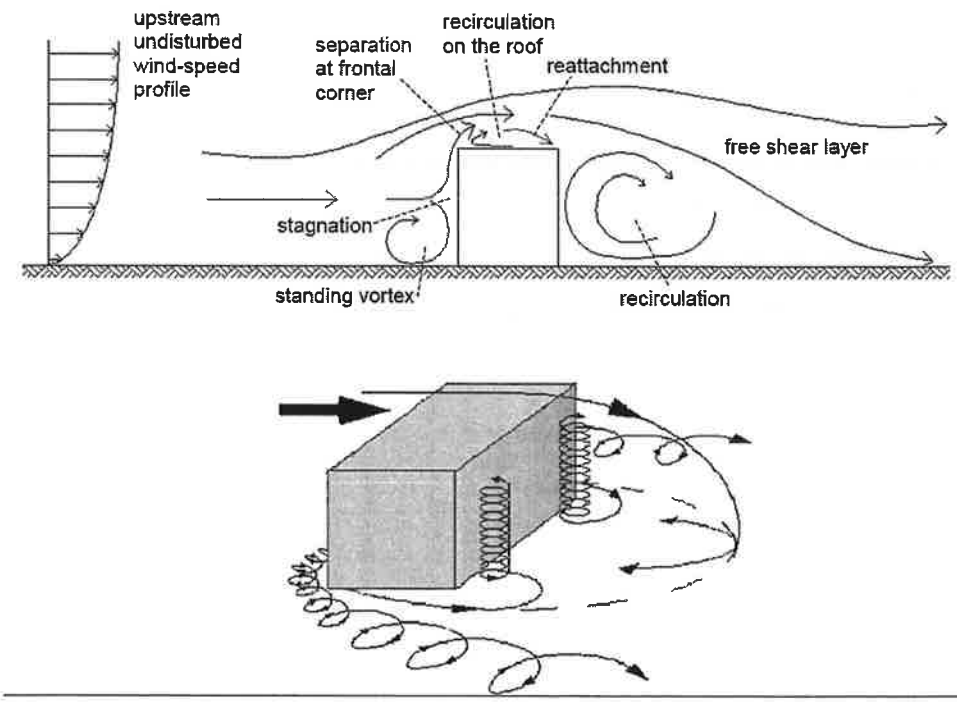


Wind flow pattern around tall slab building

Reference : Urban microclimate : designing the spaces between buildings

Urban Canopy Flow

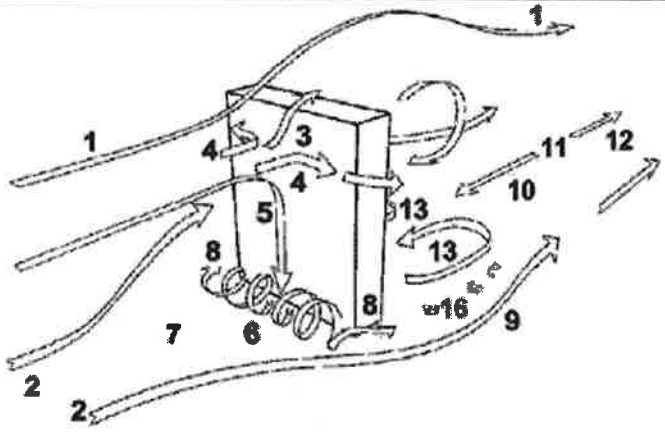
- Wind flow around a single building



23 Courtesy by Prof. Blocken, Eindhoven University of Technology

Urban Canopy Flow

- Wind flow around a single building

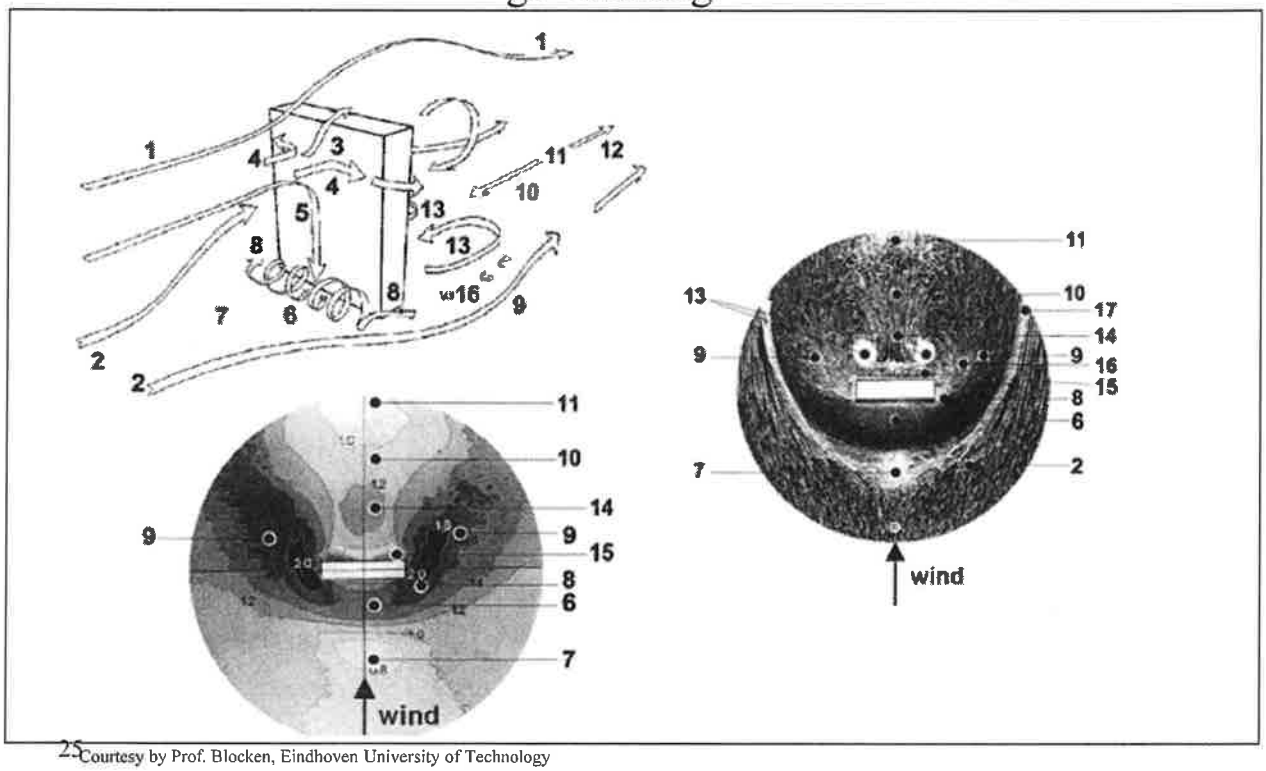


1. Flow over building
2. Oncoming flow
3. Flow from stagnation point over building
4. Flow from stagnation point around vertical building edges
5. Downflow from stagnation point
6. Standing vortex, base vortex or horseshoe vortex
7. Stagnation flow in front of building near ground level
8. Corner streams (vortex wrapping around corners)
9. Flow around building sides at ground level (adding to corner streams)
10. Recirculation flow
11. Stagnation region behind building at ground level.
12. Restored flow direction
13. Large vortices behind building
16. Small vortices in shear layer

24 Courtesy by Prof. Blocken, Eindhoven University of Technology

Urban Canopy Flow

- Wind flow around a single building – wind tunnel validation

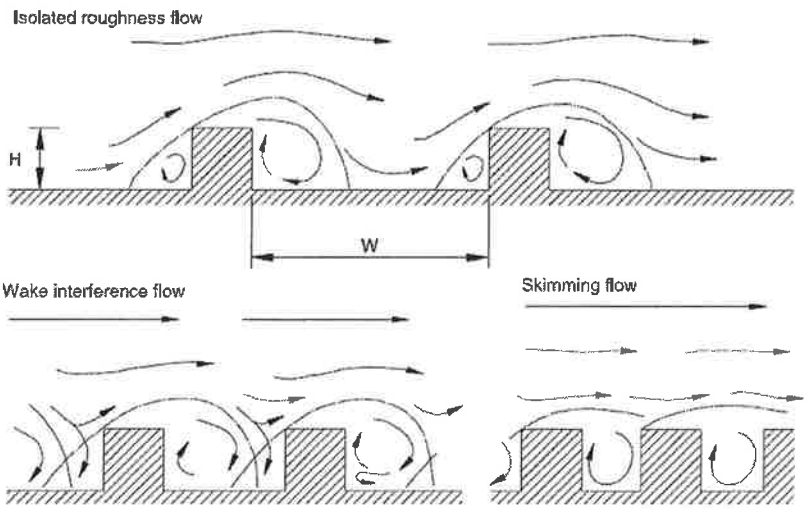


25 Courtesy by Prof. Blocken, Eindhoven University of Technology

Velocity Ratio

Urban Canopy Flow

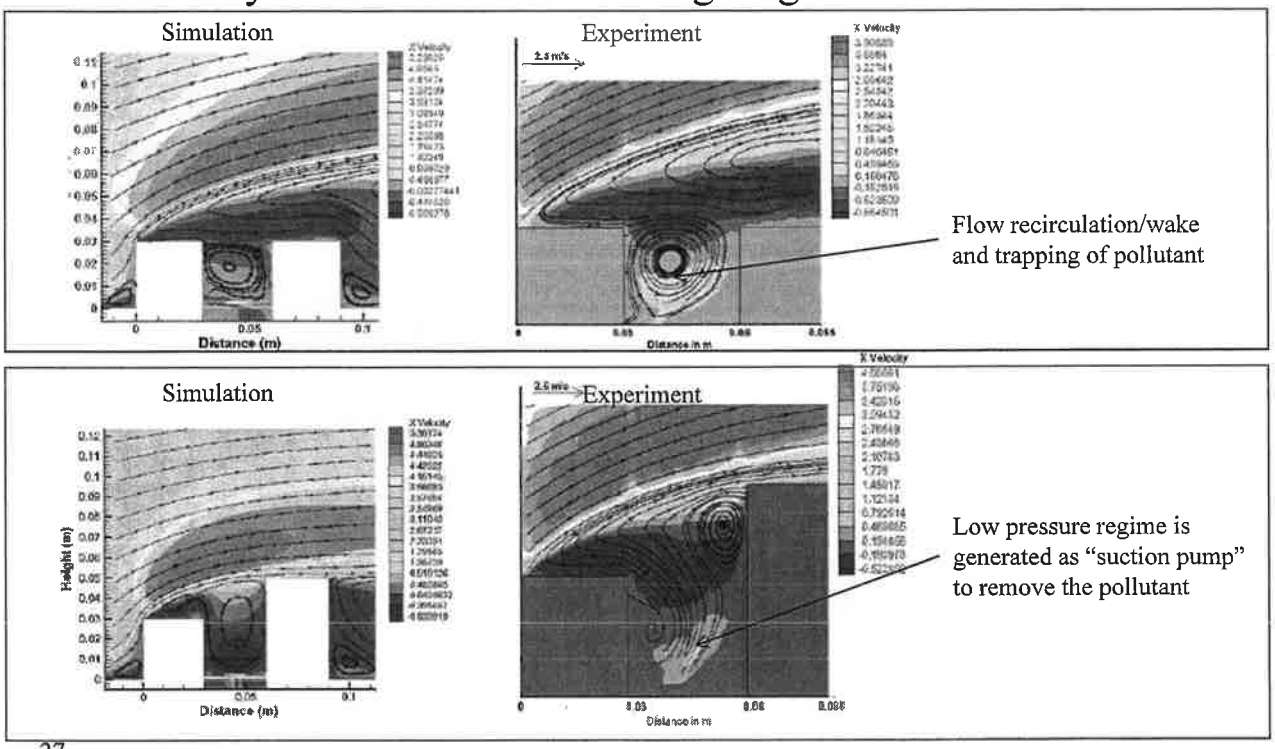
- Wind in the urban canopy – wind flow regimes
  1. Isolated Roughness Flow (IRF) → H/W<0.3
  2. Wake Interference Flow (IRF) → 0.3<H/W<0.7
  3. Skimming Flow (SF) → H/W>0.7



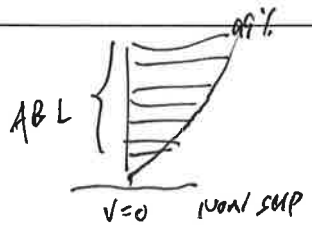
26 Reference : Boundary Layer Climates, T.R. Oke (1987)

Urban Canopy Flow

- Street canyon with different building height

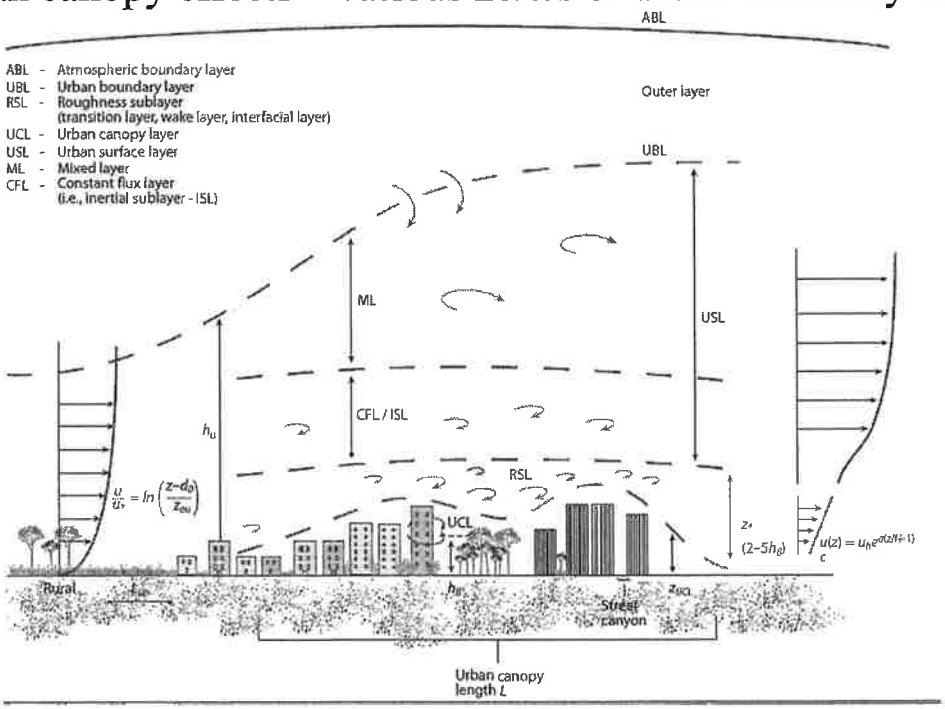


27 **Reference : Jeroen van Beeck (2002)**



Urban Canopy Flow

- Urban canopy effects – various zones of urban boundary layer



28 Reference : H.J.S. Fernando, Fluid Dynamics of Urban Atmospheres in Complex Terrain, Annu. Rev. Fluid Mech. 2010. 42:365–89

Urban Canopy Flow

- Layered structure of ABL over urban areas for moderately strong wind speeds (>3m/s)

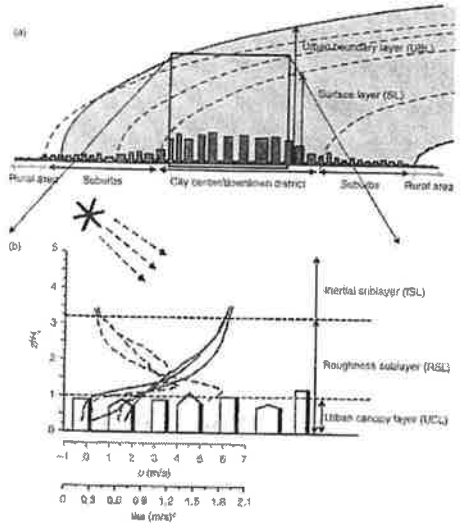


Fig 9.1 – Metropolitan sustainability

Reference : Mestayer, P.G., et al, The urban Boundary Layer field campaign in Marseille, Boundary Layer Meteorology. 2005. 114:315–365

Urban Canopy Flow

- Development of thermal driven urban circulation for weak wind (<3m/s)

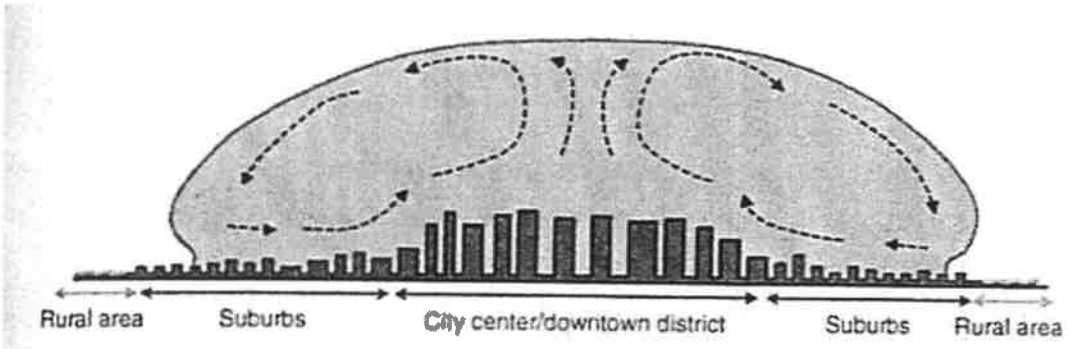
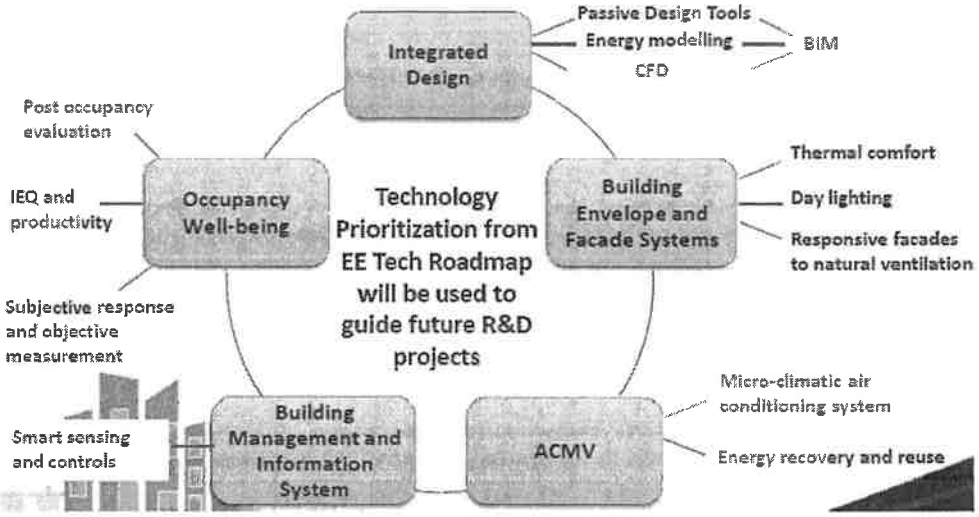


Fig 9.2 – Metropolitan sustainability

Reference : Mestayer, P.G., et al, The urban Boundary Layer field campaign in Marseille, Boundary Layer Meteorology. 2005. 114:315–365

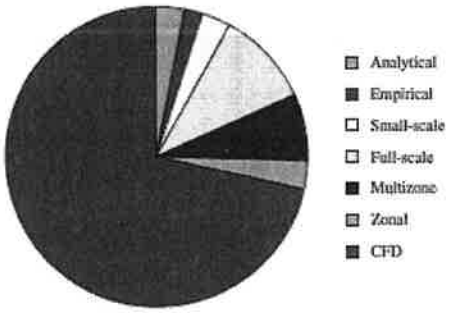
Building Energy Efficiency Technology Road Mapping

1. Air Conditioning & Mechanical Ventilation (ACMV)
2. Building Envelope & Façade System (BEFS)
3. Building Management & Information System (BMIS)
4. Integrated Design Approach and Tools (ID)
5. Occupancy Well-being (OW)



Ventilation Performance Assessment Method:

- Review & Recent Application



1. Analytical models – minimal contribution
2. Empirical models – minimal contribution
3. Reduced-scale experiments - generate data to validate numerical models
4. Full-scale experiments - generate data to validate numerical models
5. Multizone models - main tool for predicting ventilation performance in an entire building
6. Zonal models - limited applications and could be replaced by the coarse-grid fluid dynamics models
7. Computational Fluid Dynamics (CFD) models - most popular (70%), trend to couple with other building simulation model, application is mainly for indoor air quality, natural ventilation, and stratified ventilation

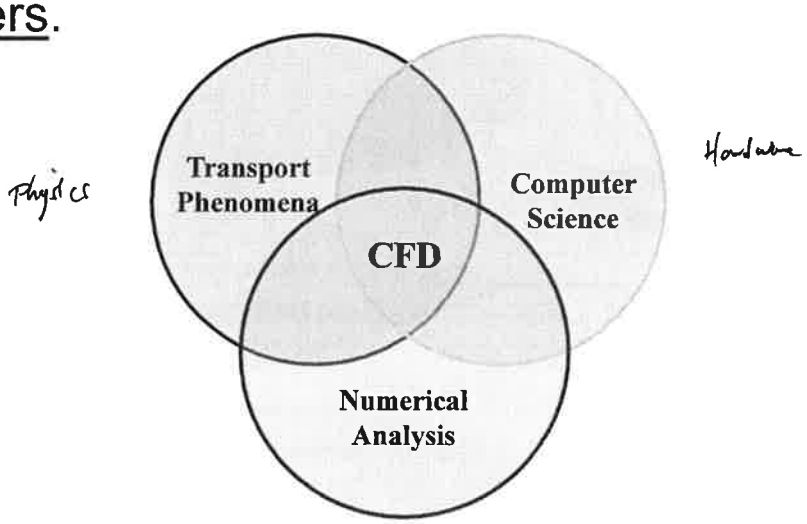
Qingyan Chen, Building and Environment, 44 (2009) 848–858

CFD-aided Natural Ventilation Design of Green Buildings

Introduction

What is CFD

- **C**omputational **F**luid **D**ynamics (**CFD**) is the scientific technology which is based on numerical methods to simulate the transport phenomena using computers.



Introduction

Mathematical Modeling

Mathematical Model – A mathematical representation to describe the behavior of a system

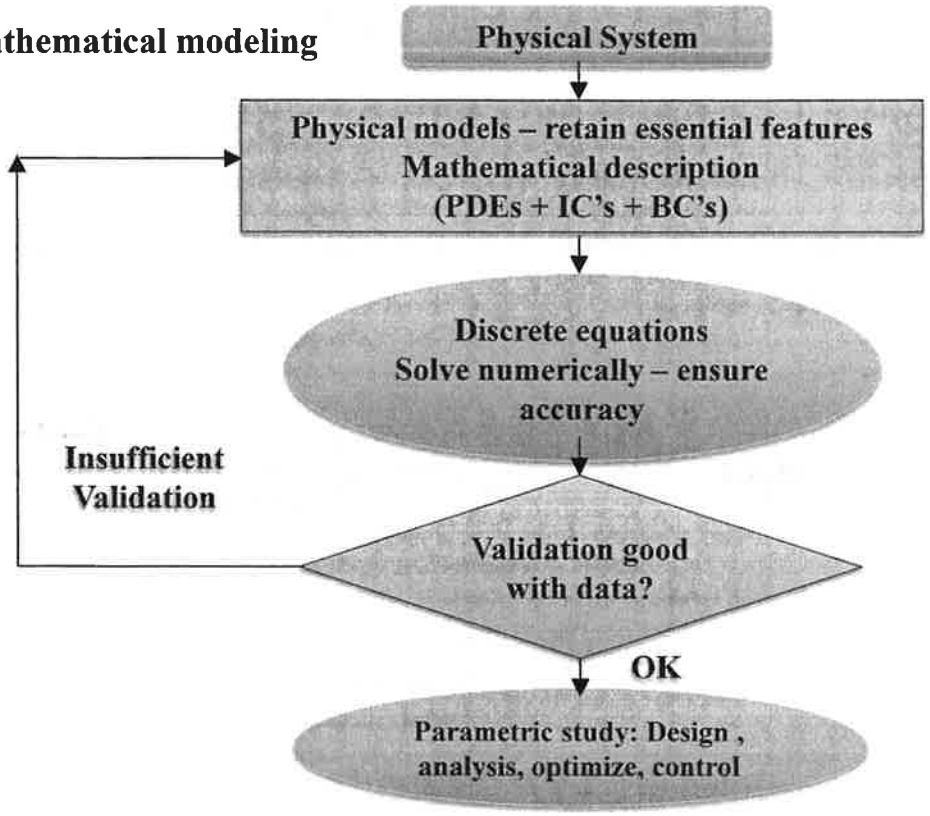
A representation of the essential aspects of a real system (or a system to be constructed) which presents knowledge of that system in a usable form. Should lead to innovative engineering design concepts

By definition, a "model" is approximate – different levels of models can be constructed

Models may be physics-based or observation based (empirical)

Introduction

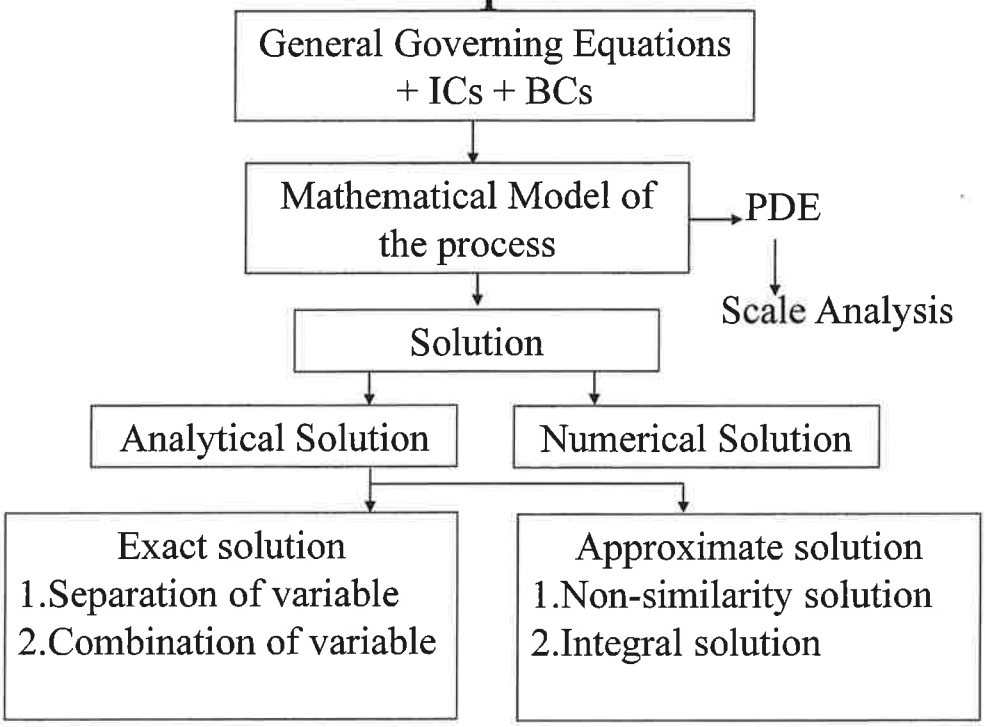
Outline of mathematical modeling



Modeling Momentum, Heat and Mass Transport Problems

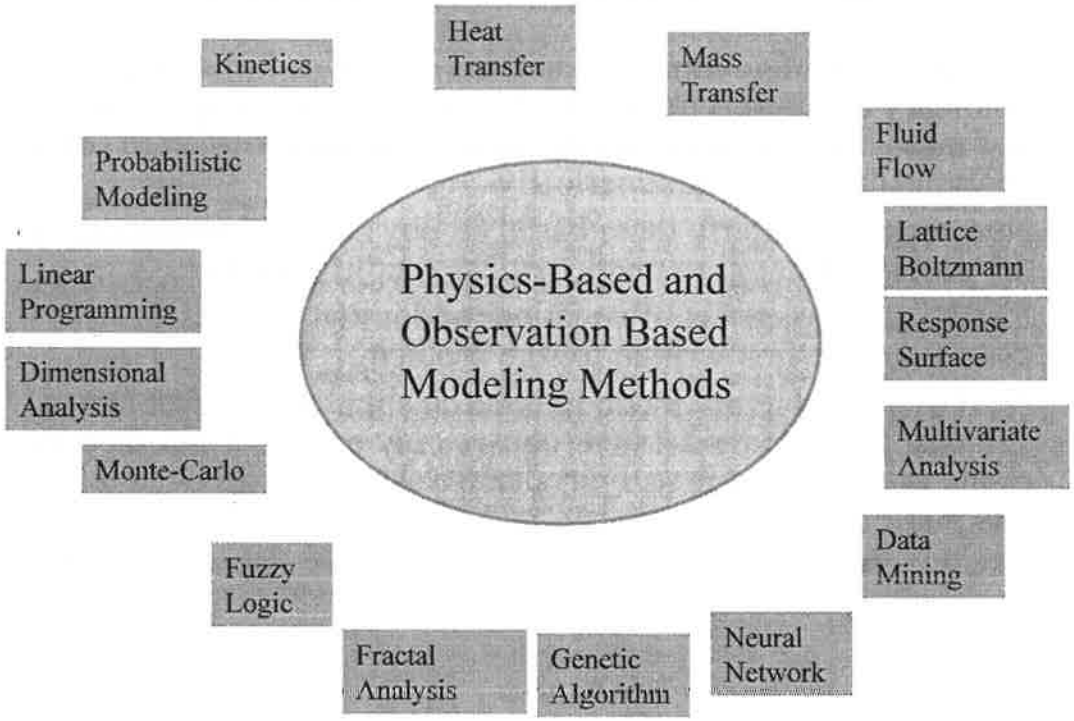
- Complete set of governing equations as well as initial and boundary conditions needed in the modeling in momentum, heat and mass transfer processes are in general too complex to solve analytically and even numerically
- Necessary to simplify them based on physical understanding of the problem leading to mathematical simplifications
- Firstly, a complete set of equations and initial/boundary conditions are first written for the problem
- Next assumptions are made in order to simplify the problem; a simplest possible model that focuses only on important aspects of the problem (but is still physically real) is almost desirable at this point
- Finally, either analytical or numerical solution is sought for the problem

Modeling Momentum, Heat and Mass Transport Problems



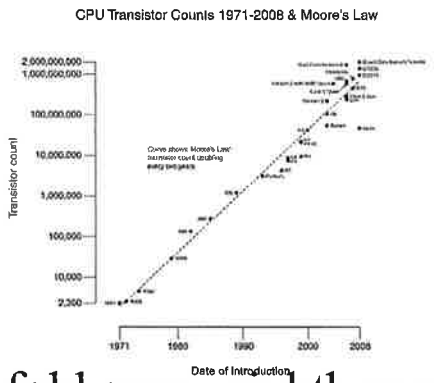
Mathematical Modeling Techniques

- Levels of mathematical modeling depends on accuracy and the resolution required for problem at hand – *Make everything simple but not simpler (Albert Einstein)*



Introduction

- Recent increased in compute power has propelled CFD models development and also fluid dynamics research in general



http://en.wikipedia.org/wiki/File:Transistor_Count_and_Moore%27s_Law_-_2008.svg, accessed on 18 June 2010

- Computer cost reduce ten fold every eighth year
- IHPC Fujitsu supercomputer systems Jan 2010 has 35 teraflops

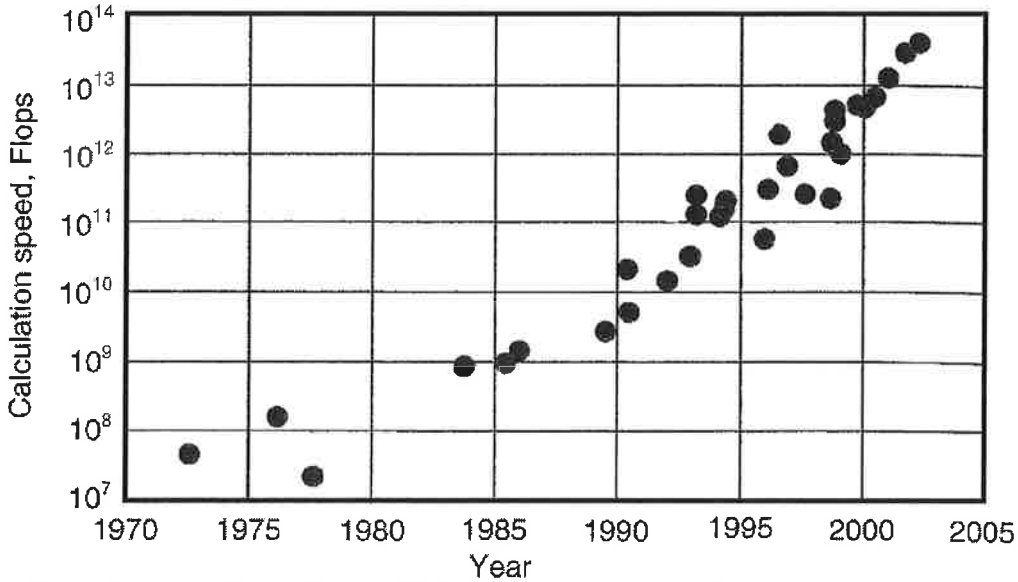
*Teraflops: One trillion floating point operations per second

Introduction

- CFD calculation is getting cheaper as time progress
  - Computer speed is increasing more rapidly than the cost
  - Continuous development of new, flexible and application-oriented CFD software
  - Improved algorithm in grid generation (preprocessor), numerical method (solver) and result viewing (postprocessor) software

Introduction

Development in Computer Cost



Petascale computing, using sophisticated supercomputers able to undertake a quadrillion (10^15) calculations per second, will provide unprecedented computing capability to solve highly complex and challenging scientific and engineering problems

Introduction

Why to simulate industrial processes

- Expensive to experiment; often impossible
- Simulation can provide trends design/operation guidelines- must have validation to enhance confidence
- Useful for process control, optimization of processes
- Can help improve energy efficiency
- Can assist in reducing environmental impact
- Helps innovation via numerical evaluation of novel concepts

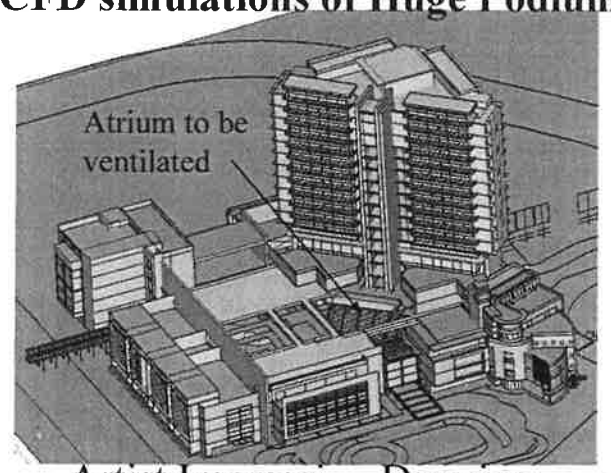
Introduction

Application of CFD Codes in Building Design

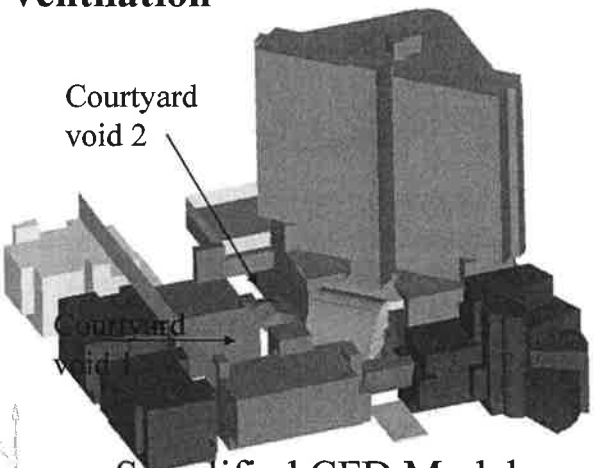
•Wind flow around buildings
•Analysis of Room air movement
•Prediction of air jet diffusion
•Prediction of contaminant dispersal
•Modeling emission from materials and equipment in buildings
•Indoor air quality prediction
•Thermal comfort assessment
•Mean age of air and ventilation effectiveness predictions
•Prediction of fire and smoke spread

CFD model demonstration

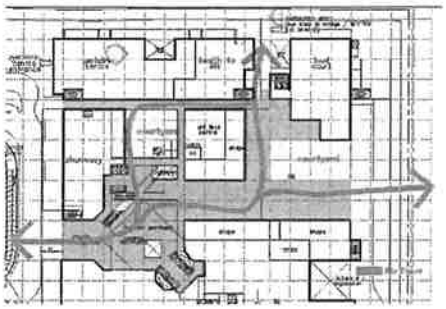
CFD simulations of Huge Podium Ventilation



Artist Impression Drawing



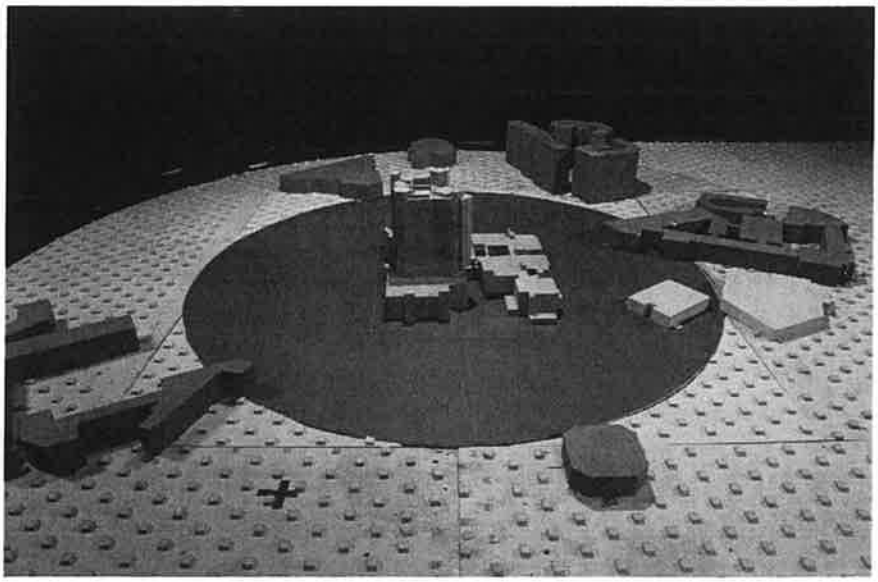
Simplified CFD Model



Objective:
Using CFD to strategize the creation of wind path through deep podium space

CFD model demonstration

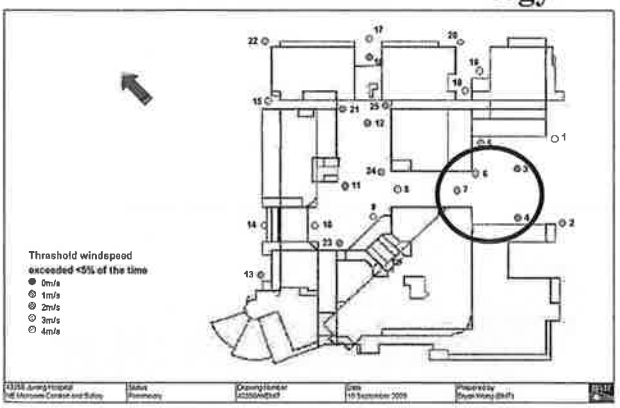
Wind Tunnel tests to validate CFD Simulations

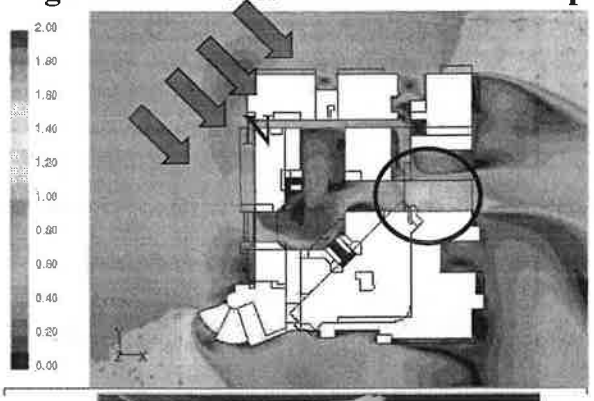


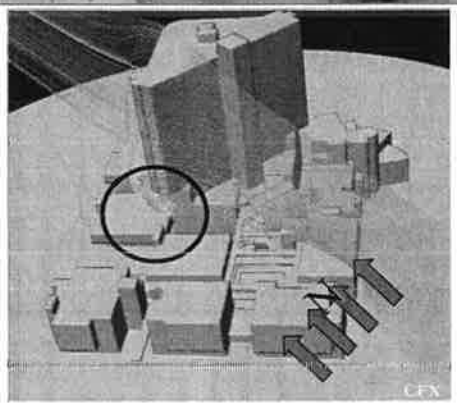
Wind tunnel model (Courtesy of BMT Fluid Mechanics Limited)

CFD model demonstration

CFD Assist Architectural Strategy for Huge Podium Ventilation in local hospital





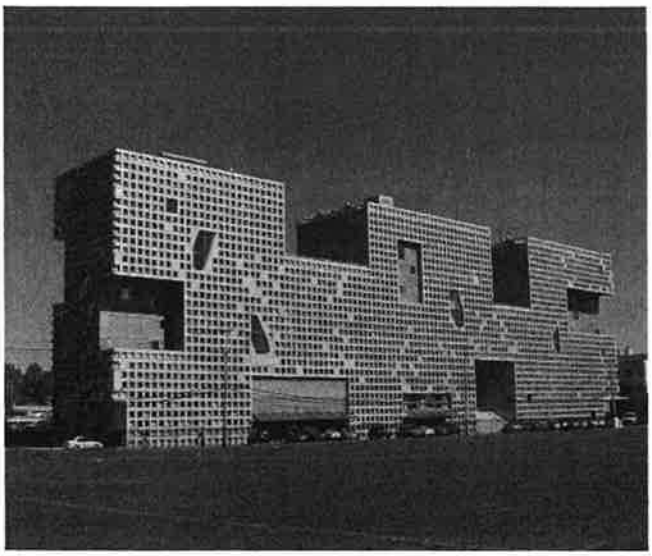


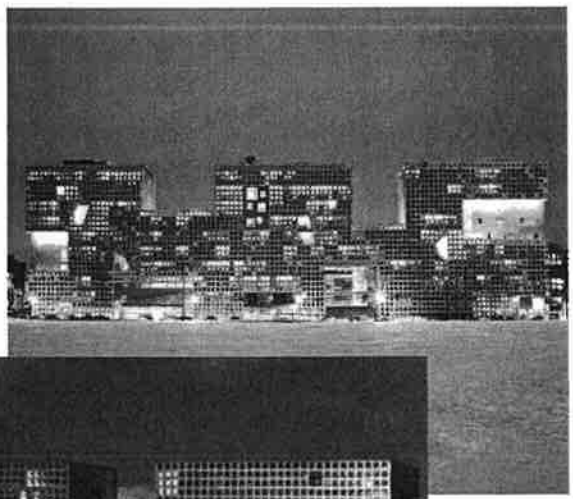
Comparison between Wind Tunnel & CFD simulation results

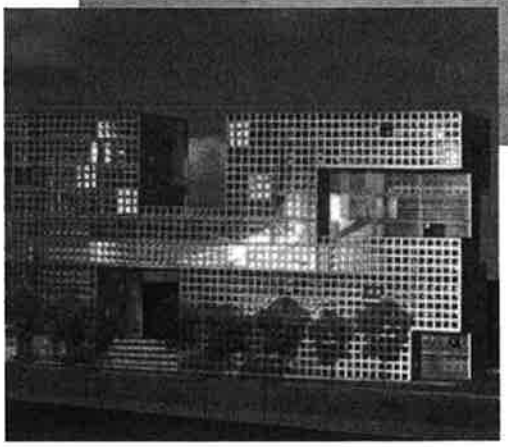
The downwash benefits spaces after void 2 at the eastern portions of buildings

47

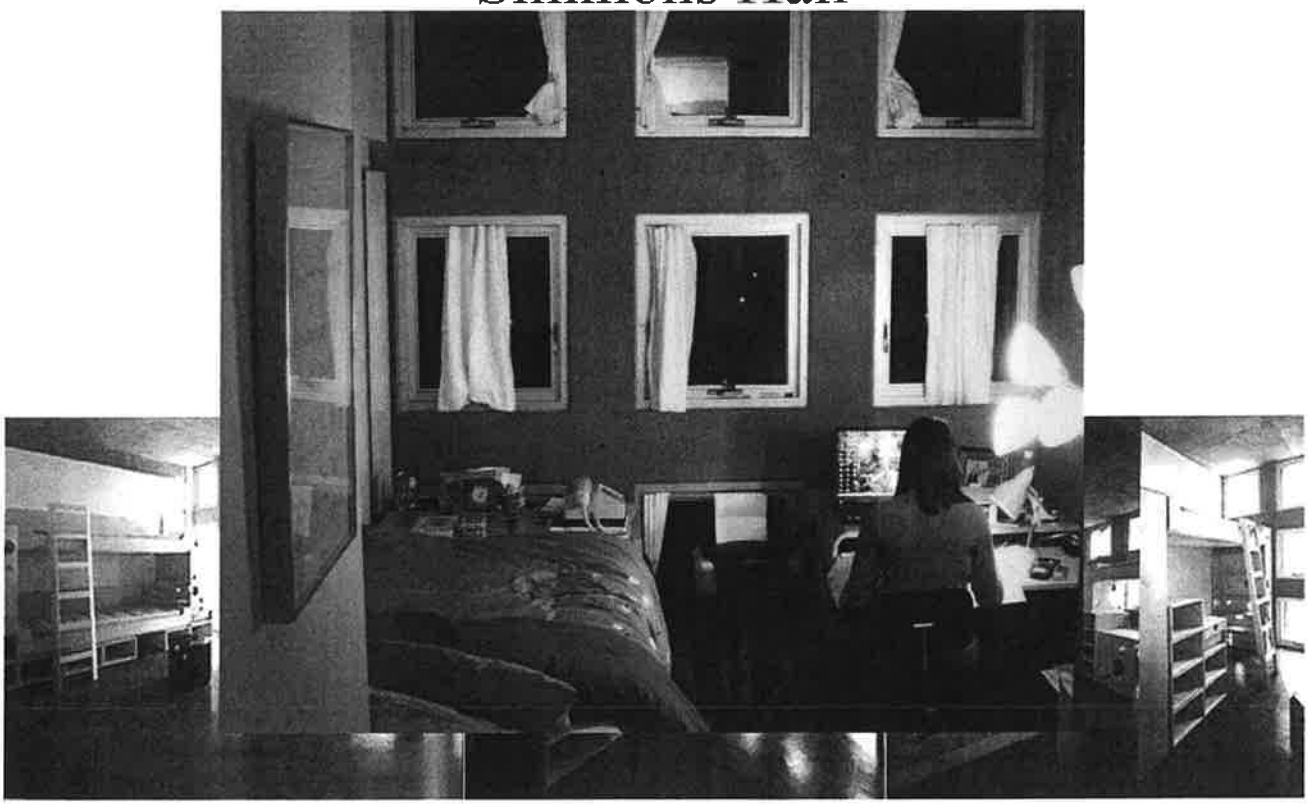
Single-sided Ventilation: Simmons Hall



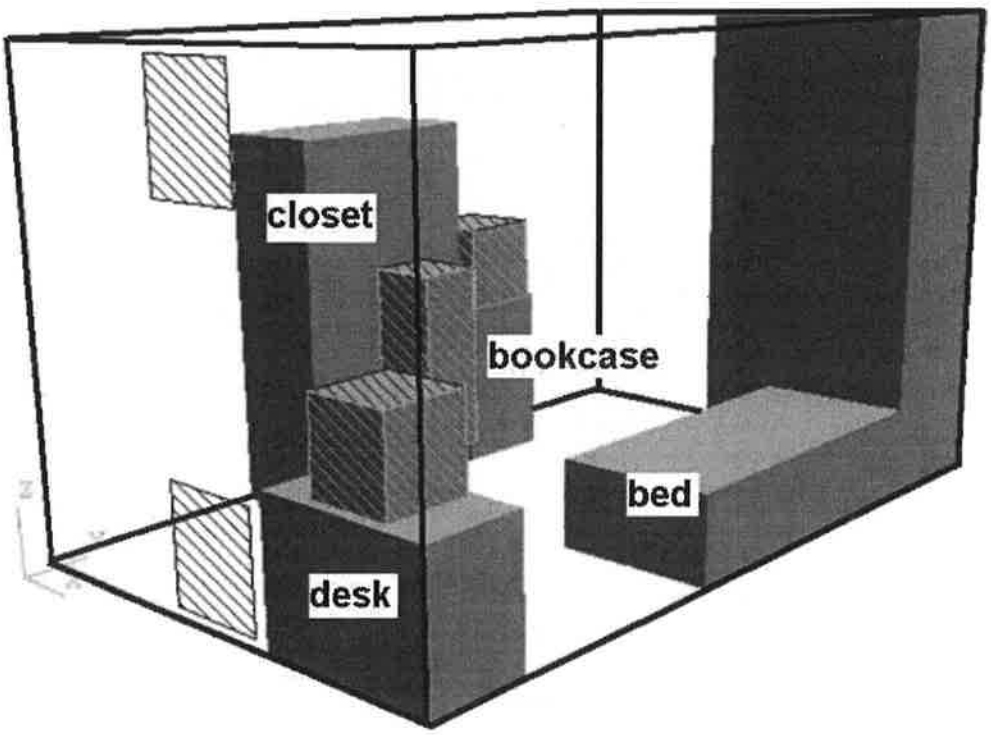




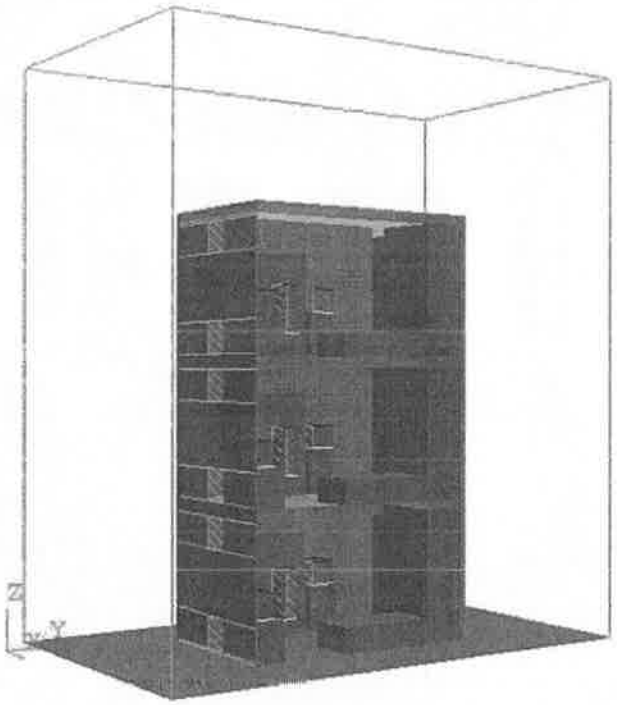
Single-sided Ventilation:
Simmons Hall



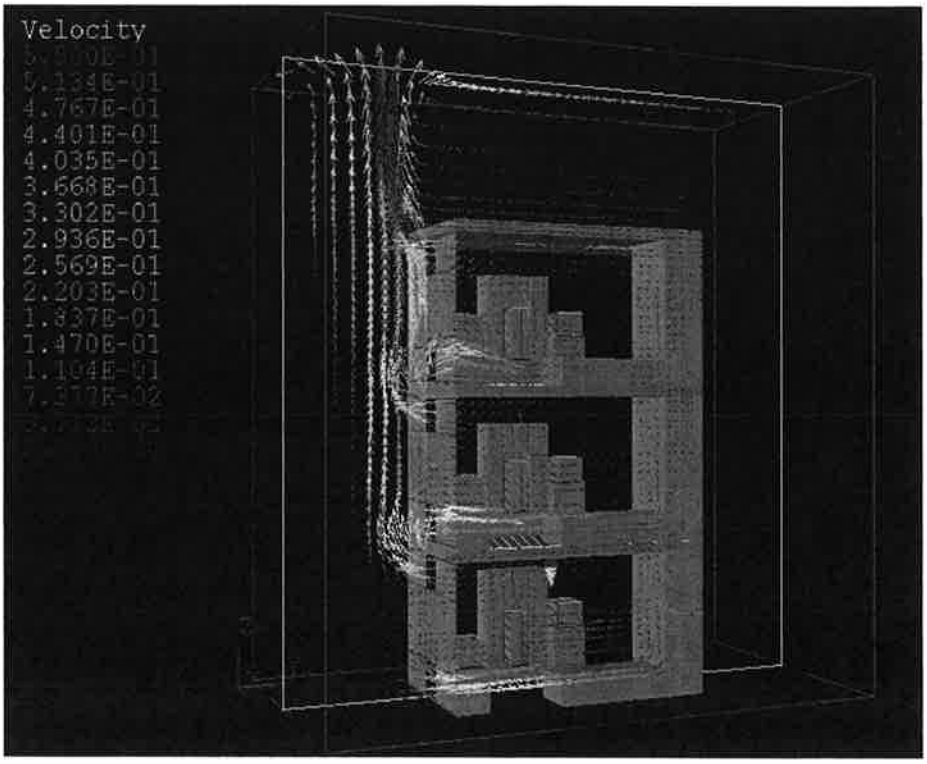
Single-sided Ventilation:
Room model



Single-sided Ventilation:
Room model

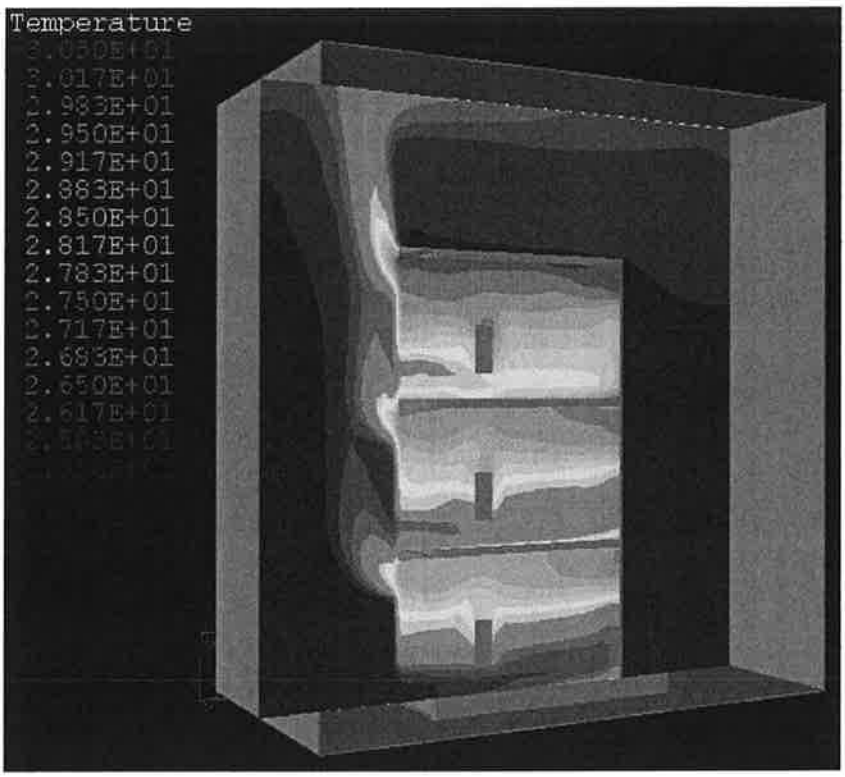


Single-sided Ventilation:
Room air velocity distribution (stack only)



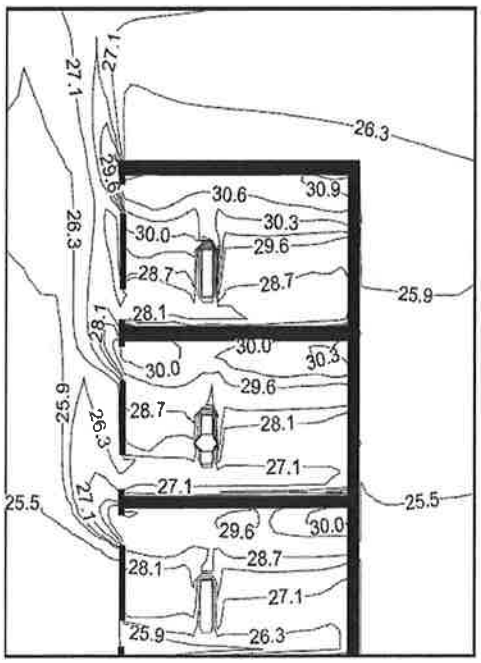
Single-sided Ventilation:

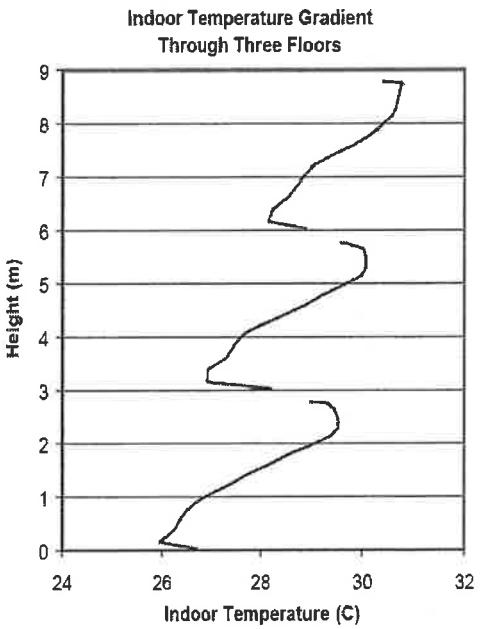
Room air temperature distribution (stack only)



Single-sided Ventilation:

Room air temperature distribution (stack only)

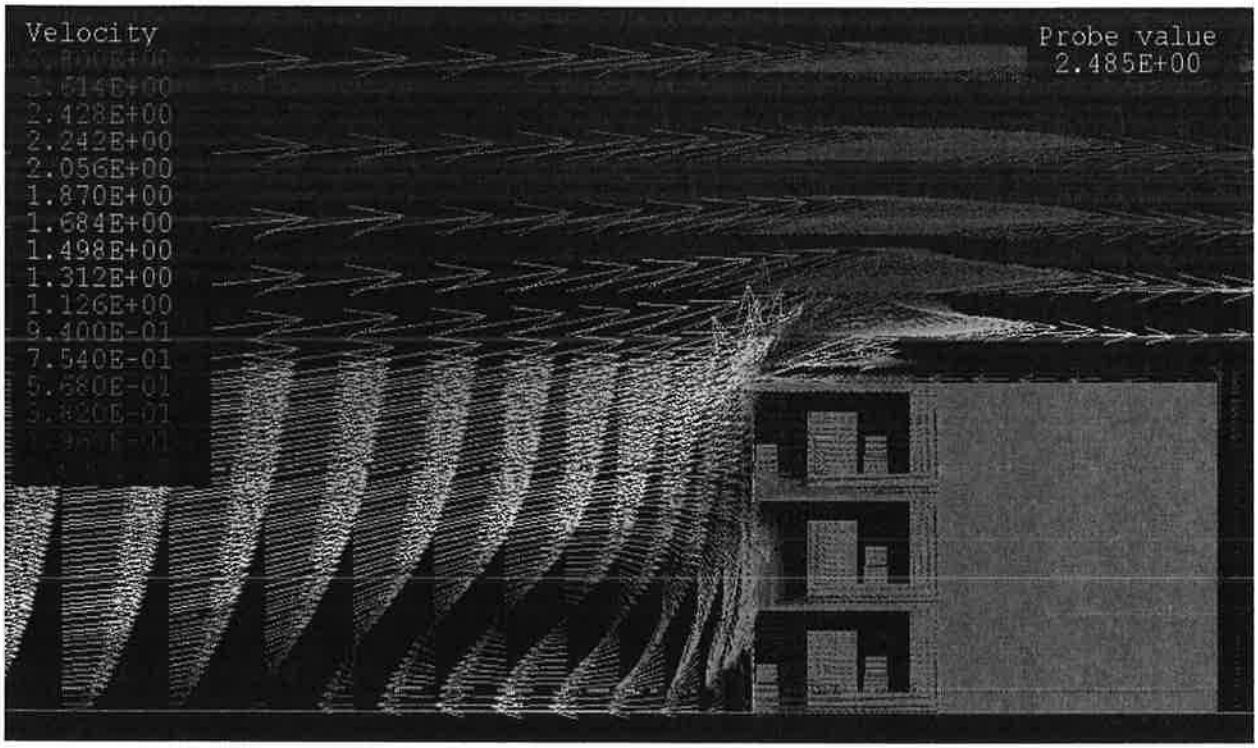




Single-sided Ventilation:

Room air velocity distribution

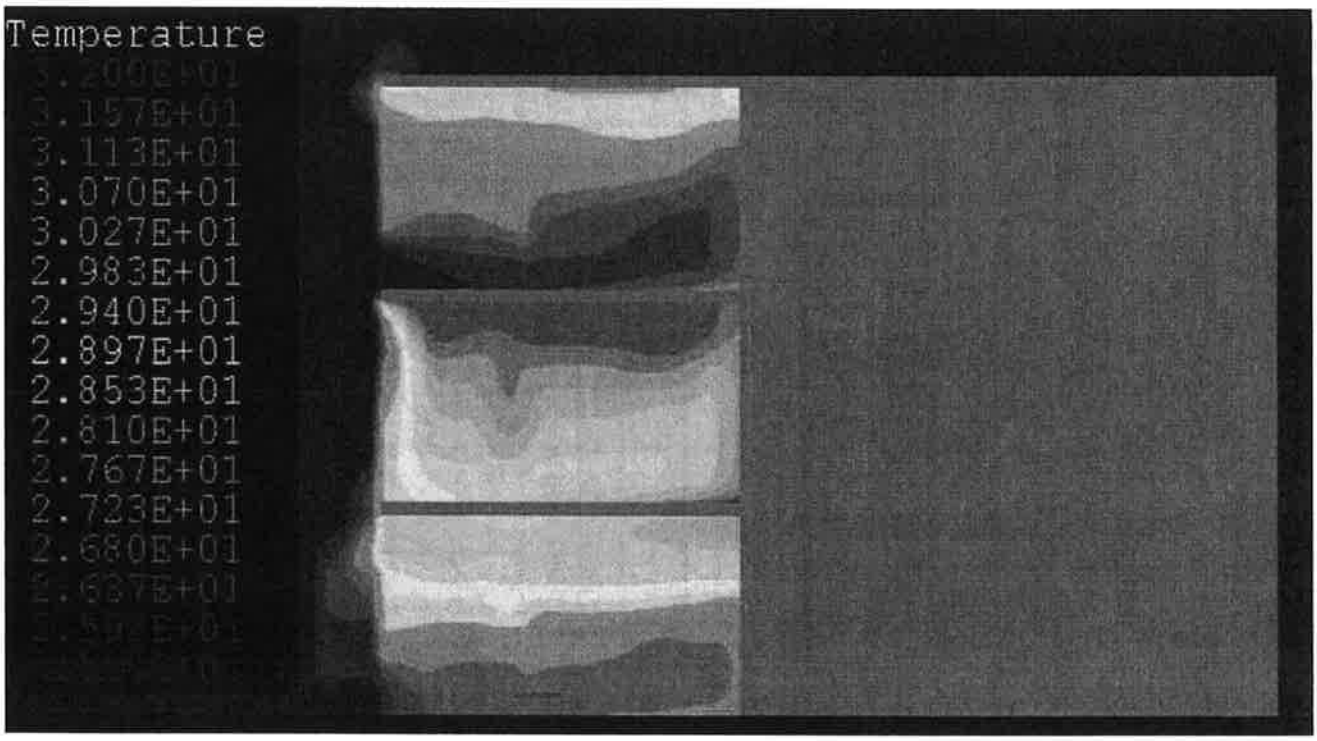
(combined wind and stack effect)



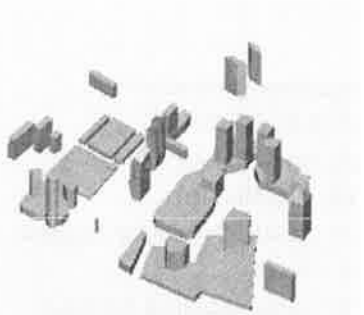
Single-sided Ventilation:

Room air temperature distribution

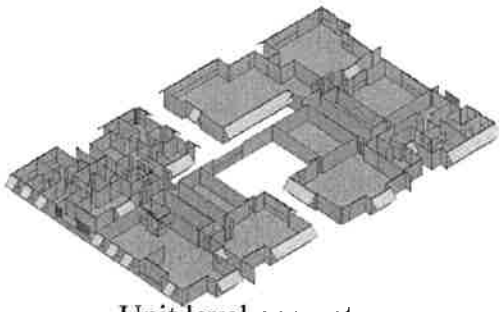
(combined wind and stack effect)



BCA Green Mark Projects



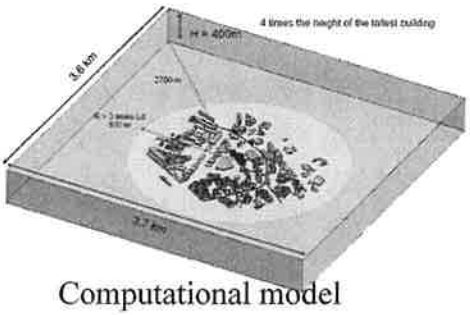
Computational model



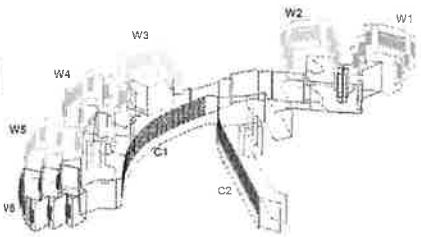
Unit level geometry



CFD result



Computational model

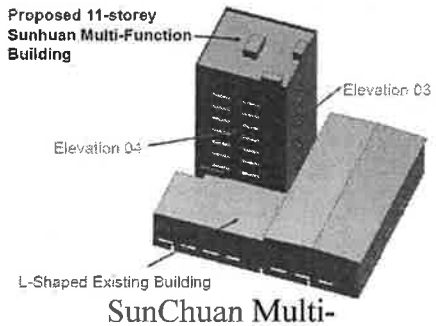


Unit level geometry

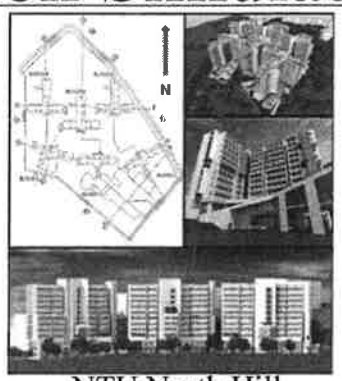


CFD result

BCA Green Mark Assessment for Natural Ventilation Simulation



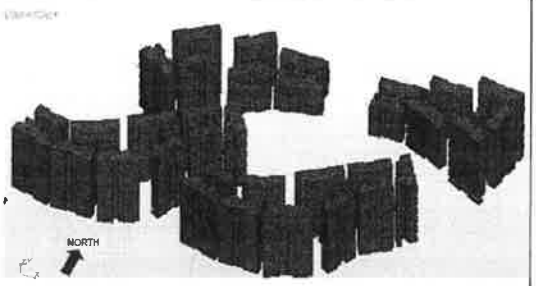
SunChuan Multi-Function Buildings



NTU North Hill Residential hall



Singapore Polytechnics Sport Arena



HDB Water Cascadia

BCA Green Mark Assessment for Natural Ventilation Simulation

- 2011
  - South Beach Mixed development
  - Simon Lane residential housing
- 2012
  - Marina South Mixed Development – Res. Component
  - Ng Teng Fong General Hospital
  - Chesnut Ave Eco-Sanctuary condominium
  - Buit Panjang Ring Road School
  - Prince Charles Primary School
  - Woodlands Specialized Secondary School, SSNT
- 2013
  - BCA Academic Workshop
  - NTU Graduate Hall
  - Tuas Shipyard development
  - Sengkang EC
  - SunChuan Multi-Function Buildings
  - Singapore Polytechnics Sport Arena
  - HDB Water Cascadia
- 2014
  - NTU Undergraduate Residential Hall
  - NTU North Hill Residential hall
  - Bukit Panjang hawker center
  - Yishun Community Hospital
  - Sengkang General and Community Hospital
  - NTU Faculty Housing
  - HDB integrated development in Admiralty
  - Ci Yuan Community Club/ Hawker Centre
  - JTC Tuas Epicenter
  - NUS Sport Center
- 2015
  - Qutram Community Hospital
  - Changi Depot

Day 2 (11:00am – 12:30pm)

- Introduction to CFD in Green Building Application
  - CFD methodologies recommended by BCA
    - Residential building (RB)
    - Non-Residential Building (NRB)
  - Available CFD tools and capabilities – a consultant's perspective
  - Case studies and examples of CFD simulation results used in optimizing natural ventilation in the built environment

CFD Methodologies for Natural Ventilation Simulation in Green Building Design

- Residential Buildings (RB)
- Non-Residential Buildings (NRB)

CFD Methodology for RB

- GM 4.1; Code for Environmental Sustainability of Buildings, 3rd Edition

Item	Category	Details
C2	**Simulation Software**	3D, Steady-state, body comforming computational grid, at least k-ε turbulence modeling with standard wall function
C3	**Ventilation Simulation Methodology**	
C3.1	***Temperature***	33°C
C3.2	***Computational Domain***	Surrounding buildings within minimum 3 times of the longest distance for the development, topo for hilly terrains, alignment with wind flow direction, domain height is at least 3 times of the tallest building in the vicinity
C3.3	***Computational Grid***	0.1-0.2m for unit, 0.5-1.0 for all building and ground level, 10m at far field with min 50m away from the ground
C3.4	***Prevailing winds***	Log-law profile with reference height of 15m, prevailing winds from the north (2.0m/s), the northeast (2.9), the southeast (3.2) and the south (2.8m/s)
C3.5	***Two best prevailing winds***	*max*(Vn, Vne), *max*(Vse, Vs) Vn, Vne, Vse and Vs are the area-weighted avg velocities of concern (1.2m above the floor)

https://www.bca.gov.sg/Envsuslegislation/others/Env_Sus_Code2013.pdf, Appendix C

CFD Methodology for RB

- GM 4.1; Code for Environmental Sustainability of Buildings, 3rd Edition

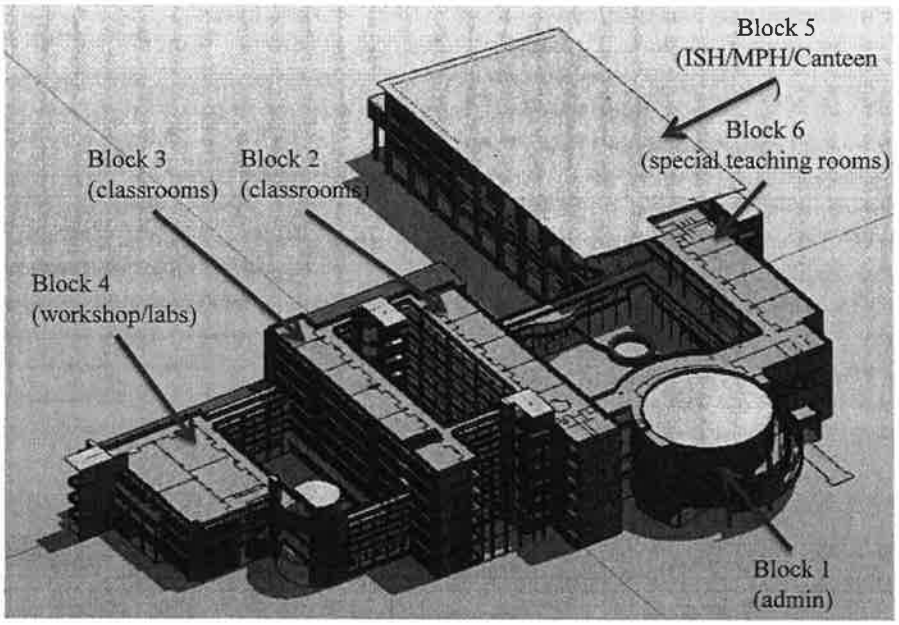
Item	Category	Details
Stage 1	**Development Model**	
(i)	Typical unit	Up to 5 typical units according to their number (descending order)
(ii)	Massing model	Inclusion of NV corridors
(iii)	Evaluation level	*min* (mid-height level, 20)
(iv)	Pressure difference	0.5m away from the openings for typical units
(v)	Representative units	±10% pressure difference range
Stage 2	**Unit Model**	
(vi)	Unit layout	All doors and windows are open except for main doors, for the selected units
(vii)	Area-weighted average velocity	Evaluated at the cutting plane at 1.2m above the floor of respective selected unit
C3.6	*Overall percentage*	$\% = \sum_{k}^{n} \frac{\Sigma(\text{No. of Selected Units for Each Layout} \times \text{Area-Weighted Average Wind Velocity})}{\text{Total Number of Selected Units} \times 0.60\ \text{m/s} \times n} \times 100\%$

n – the number of selected prevailing wind conditions

https://www.bca.gov.sg/Envsuslegislation/others/Env_Sus_Code2013.pdf, Appendix C

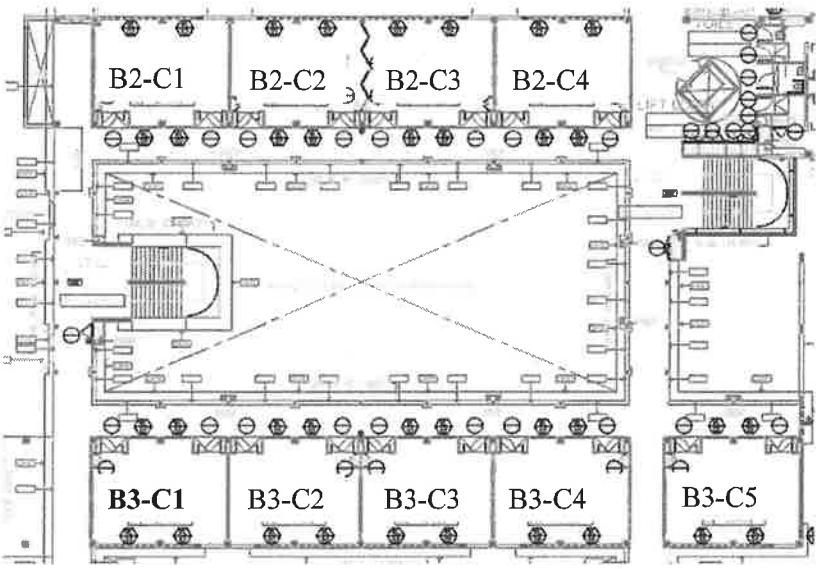
CFD Methodology for RB - Case study

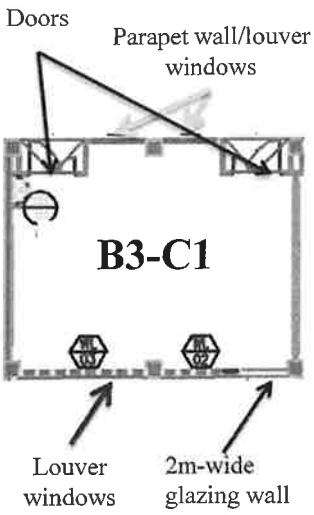
- Design – 3D View of the development



CFD Methodology for RB - Case study

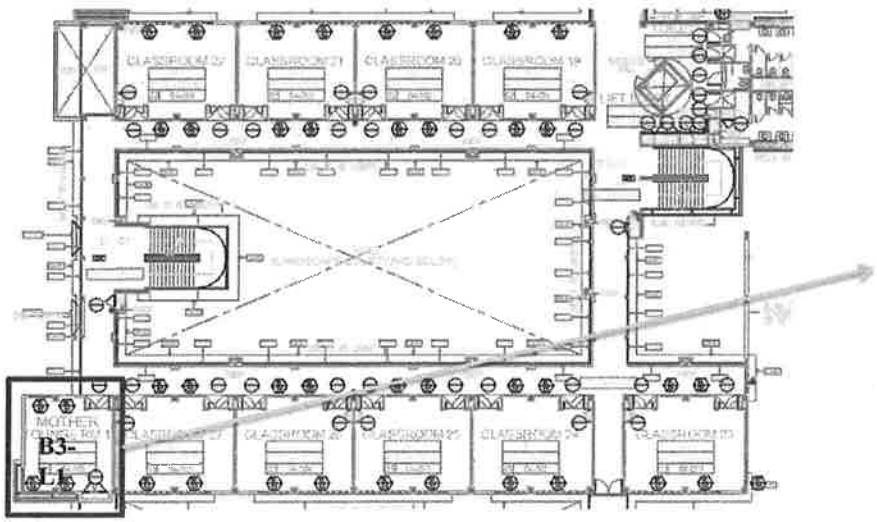
- Design – classrooms on 3rd level (plan view)

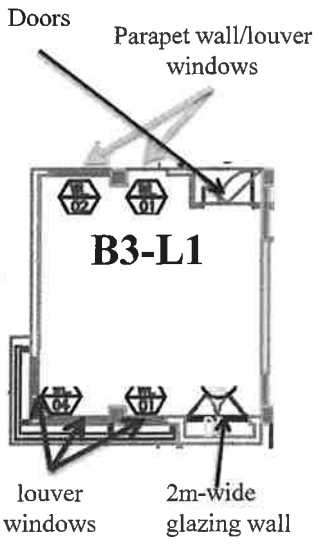




CFD Methodology for RB - Case study

- Design – mother tongue language room on 4th level (plan view)





CFD Methodology for RB - Case study

- Design – classrooms with high usage (based on NV design)

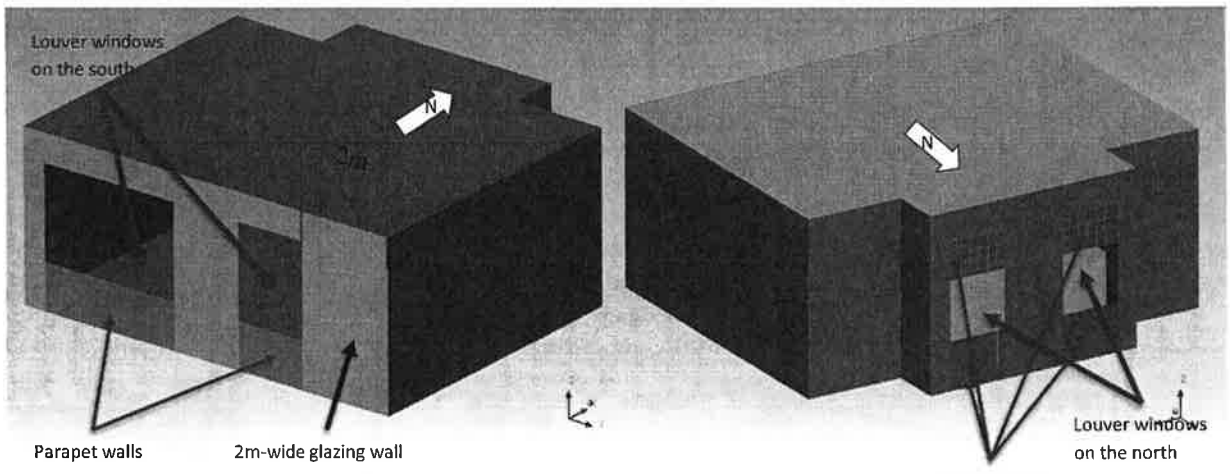
Number of classrooms in different types on different storeys of the teaching blocks

Storey	Typical classrooms	
	Typical classrooms	Mother tongue rooms
2	9	0
3	9	0
4	9	1
5	9	1
6	9	1
sum	45	3

* All typical classrooms are $60m^2$ in area and take the same design layout.

CFD Methodology for RB - Case study

- Design – Layout of the typical classroom in **Baseline** design

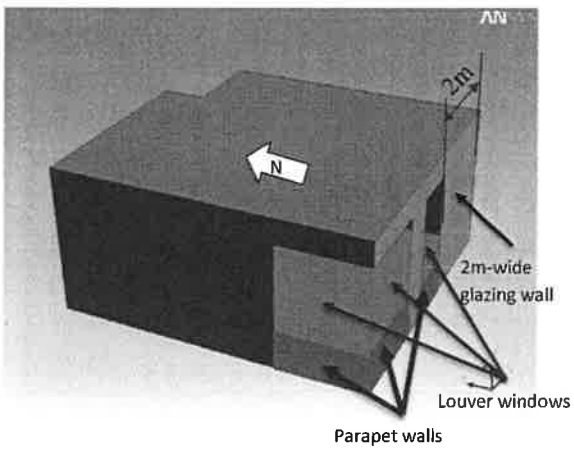


(a) South side

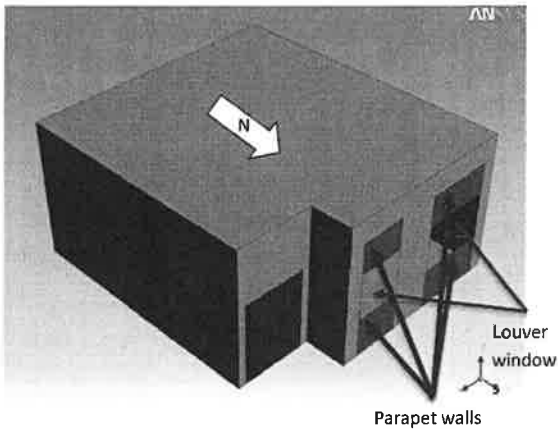
(b) North side

CFD Methodology for RB - Case study

- Design – Mother tongue language rooms in **Baseline** design



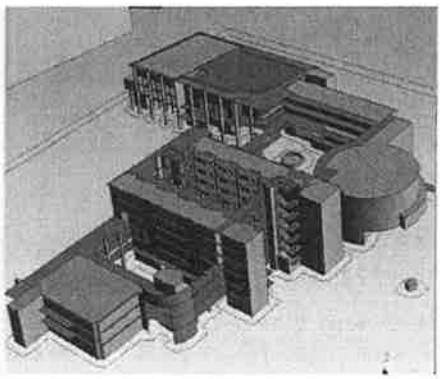
(a) South side



(b) North side

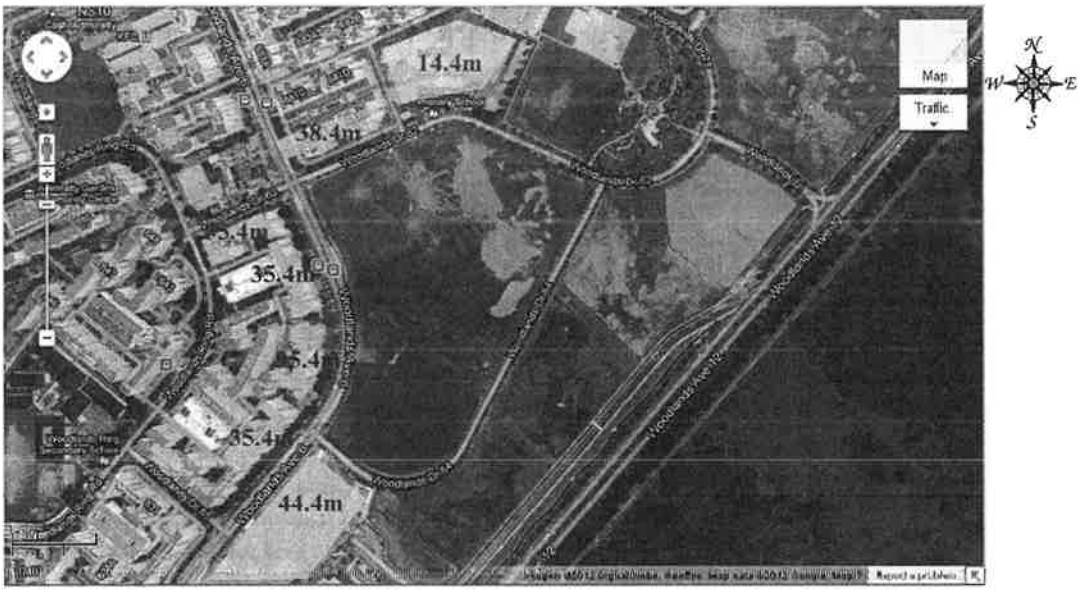
CFD Methodology for RB - Case study

- Objective
  - To verify whether at least 70% of natural-ventilated classrooms can receive good ventilation as required for BCA GM Platinum rating
  - To improve the design, if necessary
- Natural ventilation simulations
  - Two large-scale models
    - **Development models**
      - **To determine the representative units in subsequent simulations**
    - **Units models**
      - **To determine area-weighted average wind velocity**



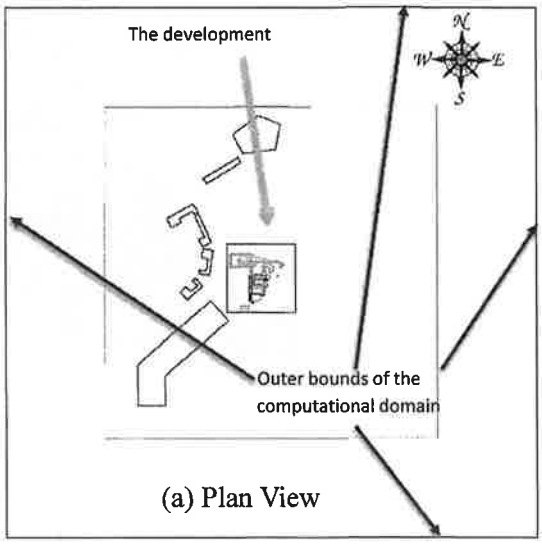
CFD Methodology for RB
- Case study

•Computational domains – immediate surrounding buildings

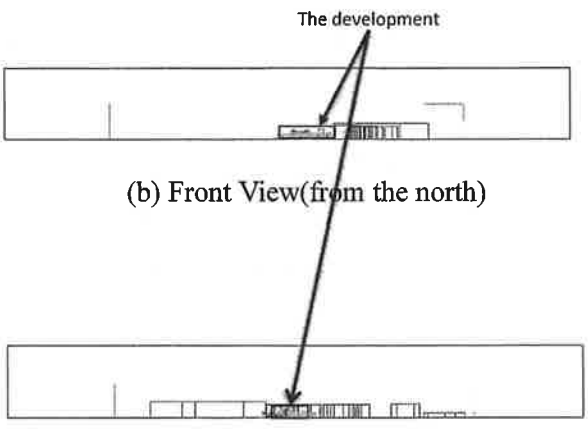


CFD Methodology for RB
- Case study

•Computational domains – for North and South winds



(a) Plan View

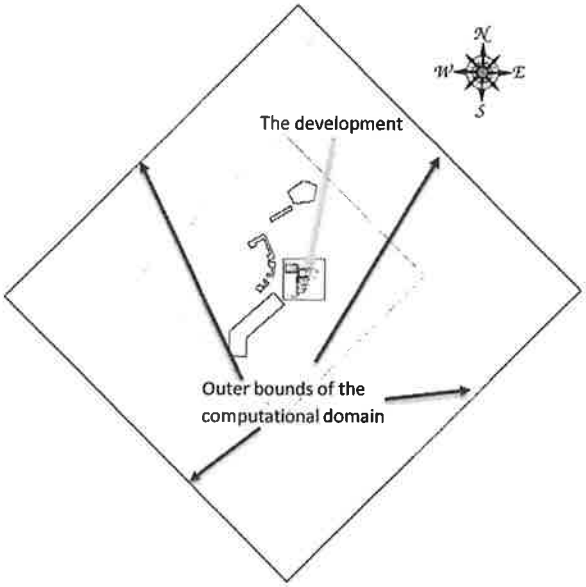


(b) Front View(from the north)

(c) Side View (from the east)

CFD Methodology for RB
- Case study

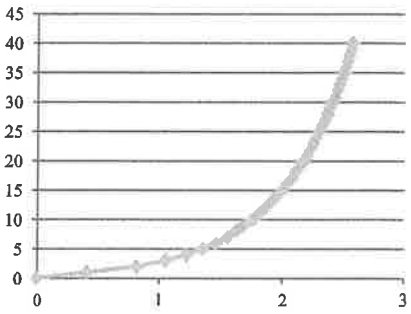
•Computational domains – for Northeast and Southeast winds



CFD Methodology for RB
- Case study

• Prevailing wind conditions

Wind Direction	Wind Speed (m/s)
North	2.0
Northeast	2.9
South	2.8
Southeast	3.2



•Inbound log-law wind profile

✓ $U = U_{ref} \ln(Z/Z_0) / \ln(Z_{ref}/Z_0)$

Where Z_{ref} = *15m*, Z_0 = *0.5m*

CFD Methodology for RB
- Case study

Computational Mesh

Mesh size functions selected in the study

Locations of source	Mesh size at source	Expansion ratio
Classrooms	≤0.2	1.2
Surface of buildings	≤0.5	1.2
Ground	≤1.0	1.2
Surrounding buildings	≤2.0	1.2
Far field surfaces	≤10.0	1.2

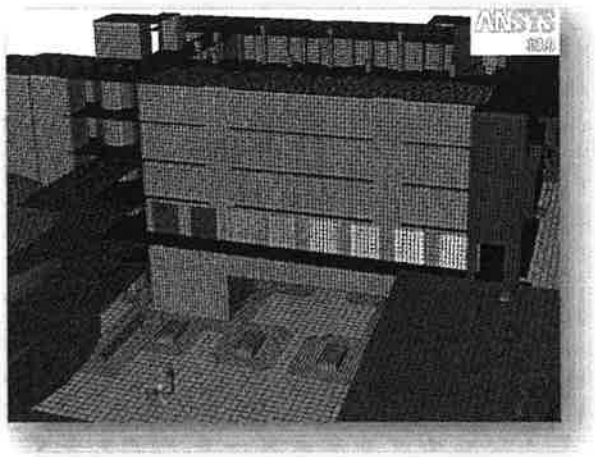
*Global mesh size setting: $0.05 \leq \Delta x \leq 10.0$

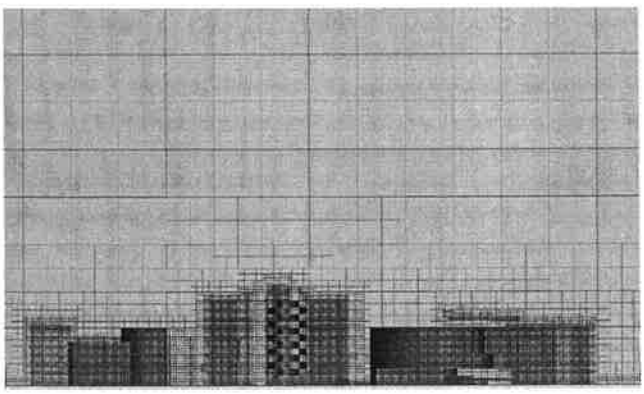
⟹ >10 million hex-dominated elements

To include if column is more than 10% of clear height

CFD Methodology for RB
- Case study

- Hexahedra-dominated cut-cell mesh



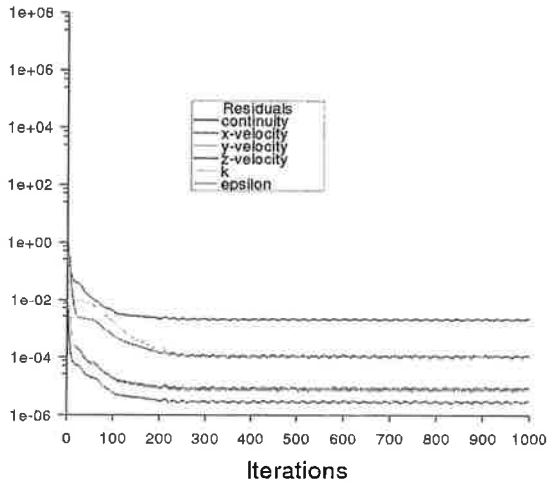


CFD Methodology for RB - Case study

- Steady-state simulations
  - ✓ Isothermal condition – T = 33°C
  - ✓ Realizable k-ε to simulate turbulence effects
  - ✓ First-order discretization schemes for approximations of partial derivatives
  - ✓ SIMPLE solution procedure
- Boundary conditions
  - ✓ Velocity Inlet conditions subjected to in-bound log-law wind profile are imposed at inflow surface.
  - ✓ Pressure Outlet conditions are set at the opposite.
  - ✓ Symmetry conditions are used on the two sides and top bounding surfaces for computational domains.
  - ✓ Interior conditions are imposed at respective louver windows of selected units (for Unit models only).
  - ✓ Non-slip Wall conditions are used by default for all other building surfaces and the ground.

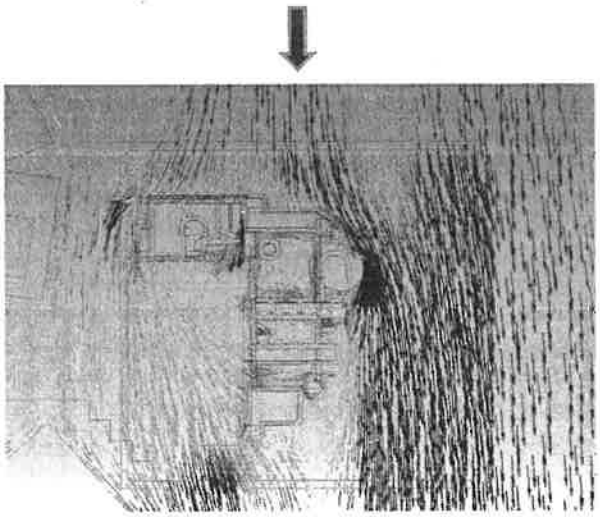
CFD Methodology for RB - Case study

- Convergence history
  - ✓ Targeted residuals for all variables: 1.0×10^{-4}
  - ✓ Computations are terminated when residuals curves become flatten.

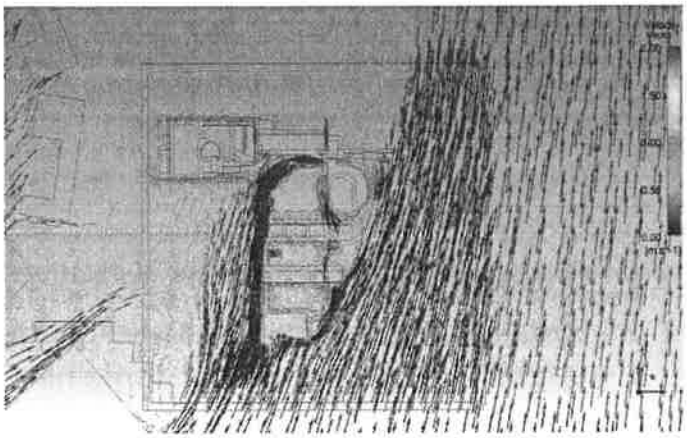


CFD Methodology for RB
- Case study

- Development Model
  - ✓ Airflow behaviors – @1.2m above the floor of 3rd-storey



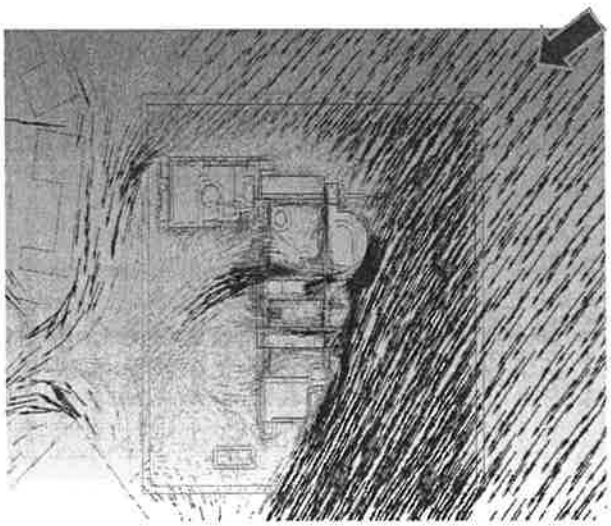
North Wind



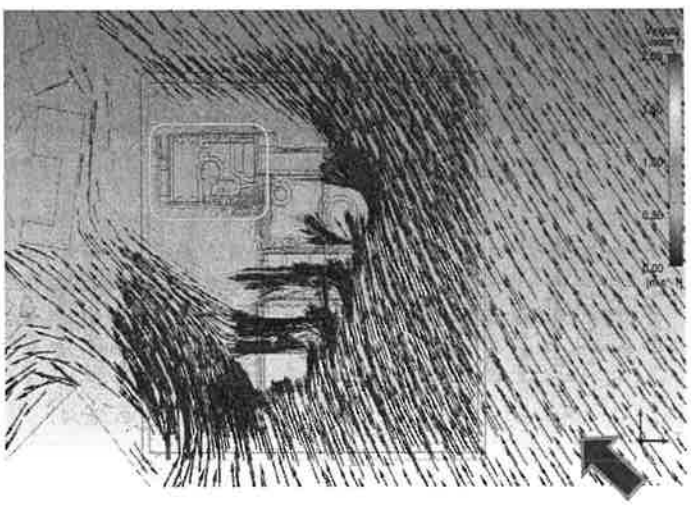
South Wind

CFD Methodology for RB
- Case study

- Development Model
  - ✓ Airflow behaviors – @1.2m above the floor of 3rd-storey



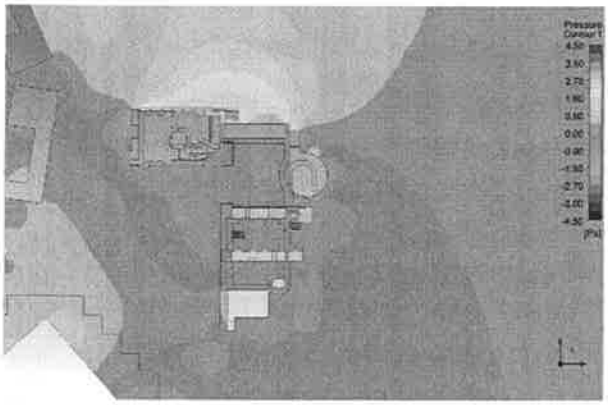
Northeast Wind



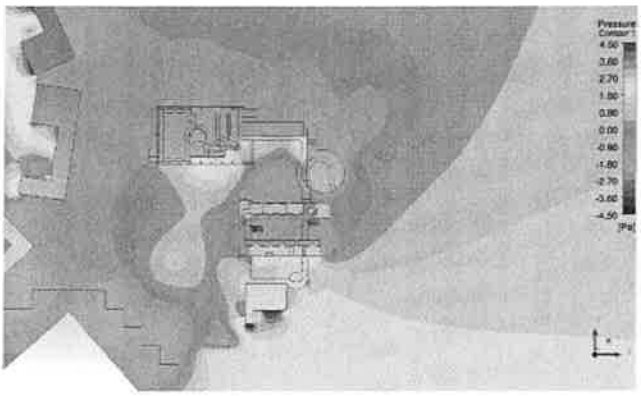
Southeast Wind

CFD Methodology for RB - Case study

- Development Model
  - ✓ Pressure contours – @1.2m above the floor of 3rd-storey



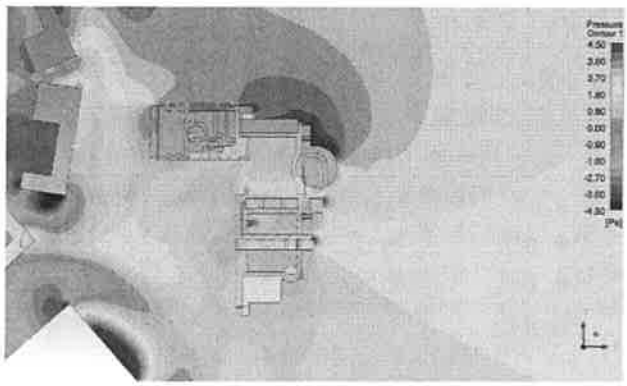
North Wind



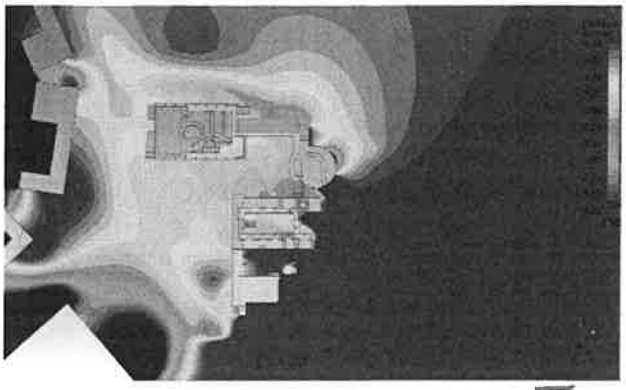
South Wind

CFD Methodology for RB - Case study

- Development Model
  - ✓ Pressure contours – @1.2m above the floor of 3rd-storey



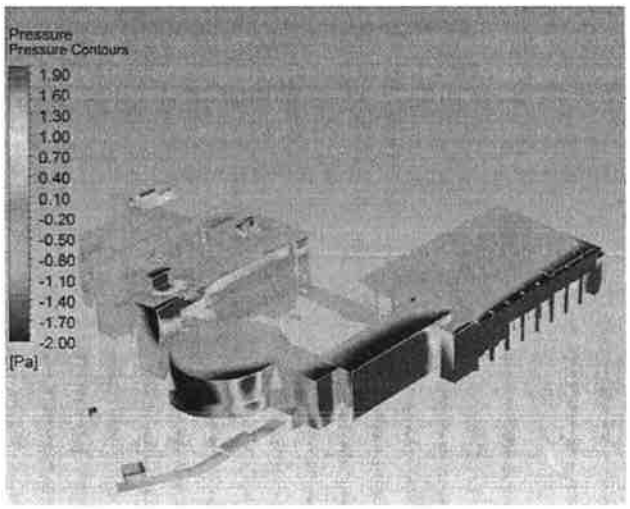
Northeast Wind



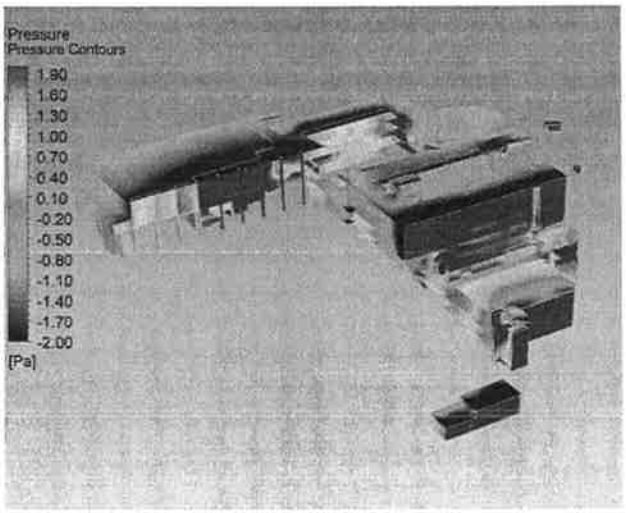
Southeast Wind

CFD Methodology for RB
- Case study

- Development Model
  - ✓ Pressure contours – on building surfaces



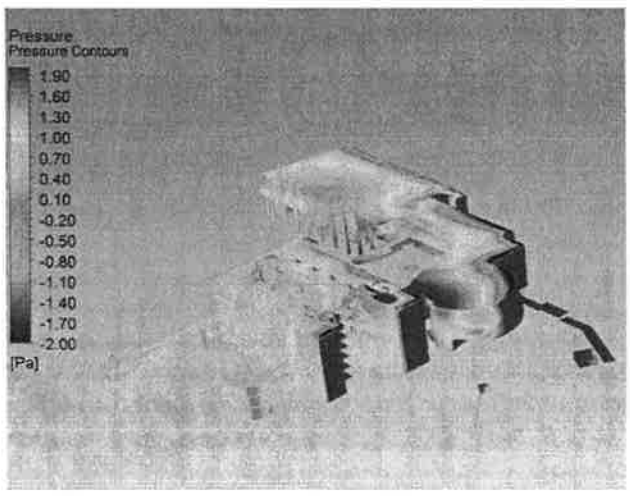
North Wind



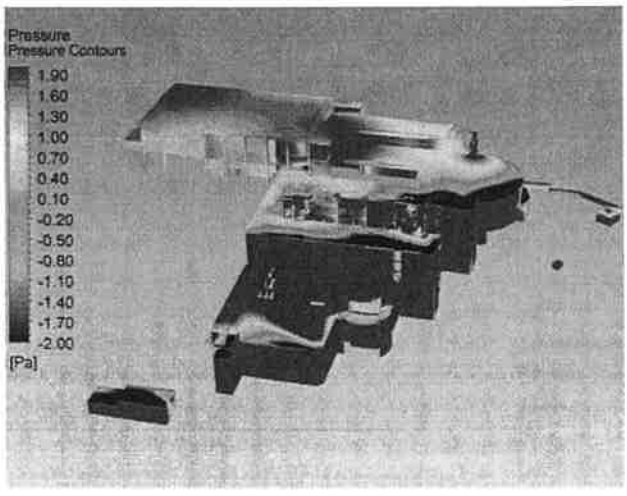
South Wind

CFD Methodology for RB
- Case study

- Development Model
  - ✓ Pressure contours – on building surfaces



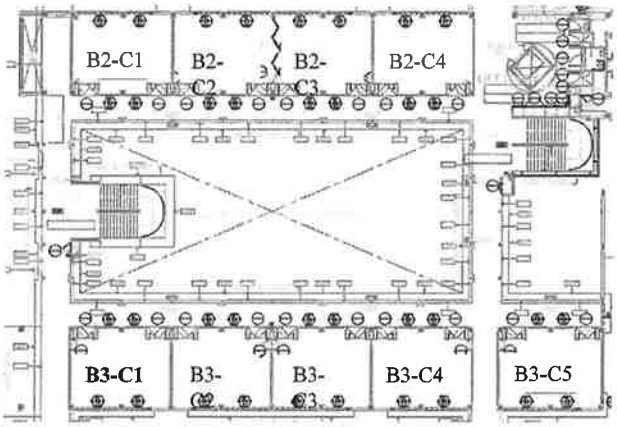
Northeast Wind



Southeast Wind

CFD Methodology for RB - Case study

- Development Model
  - ✓ Monitored pressure locations
    - Surfaces @0.5m away from louver windows for **B2-C1**, **B2-C2**, **B2-C3**, **B2-C4**, **B3-C1**, **B3-C2**, **B3-C3**, **B3-C4**, **B3-C5** at 3rd-storey
    - Area averaged pressures instead of point-based pressures



CFD Methodology for RB - Case study

- Development Model
  - ✓ Monitored pressure differences

	NORTH		SOUTH		NORTHEAST		SOUTHEAST		AVERAGE INDIVIDUAL UNIT
Location	Pressure	Abs Pressure Diff	Pressure	Abs Pressure Diff	Pressure	Abs Pressure Diff	Pressure	Abs Pressure Diff	Abs Pressure Dif
win-b2-c1-n	-0.393	0.250	-1.174	0.430	2.623	1.657	0.680	0.887	0.806
win-b2-c1-s	-0.643		-0.744		0.966		1.567		
win-b2-c2-n	-0.290	0.276	-1.130	0.229	2.815	1.632	0.453	1.212	0.837
win-b2-c2-s	-0.565		-0.901		1.183		1.665		
win-b2-c3-n	-0.245	0.308	-1.011	0.276	2.535	1.703	0.390	1.652	0.985
win-b2-c3-s	-0.554		-0.734		0.832		2.042		
win-b2-c4-n	-0.246	0.319	-0.707	0.324	2.366	1.701	0.032	3.309	1.413
win-b2-c4-s	-0.566		-0.383		0.665		3.341		
win-b3-c1-n	-0.617	0.423	-0.916	2.730	1.153	0.192	1.214	3.393	1.684
win-b3-c1-s	-1.040		1.815		0.961		4.606		
win-b3-c2-n	-0.589	0.239	-0.902	3.014	1.192	0.381	0.773	4.707	2.085
win-b3-c2-s	-0.828		2.113		0.811		5.480		
win-b3-c3-n	-0.612	0.175	-0.827	2.641	1.041	0.230	0.689	5.031	2.019
win-b3-c3-s	-0.787		1.813		0.812		5.719		
win-b3-c4-n	-0.643	0.007	-0.802	2.654	1.062	0.170	-0.125	7.502	2.583
win-b3-c4-s	-0.637		1.852		0.893		7.377		
win-b3-c5-n	-0.697	0.185	-1.131	4.279	1.648	0.778	1.672	9.029	3.568
win-b3-c5-s	-0.512		3.149		0.870		10.701		
Average	-	0.242	-	1.842	-	0.938	-	4.080	1.776

CFD Methodology for RB - Case study

- Development Model
  - ✓ Monitored pressure differences

Calculated average pressure drops for all monitored classrooms

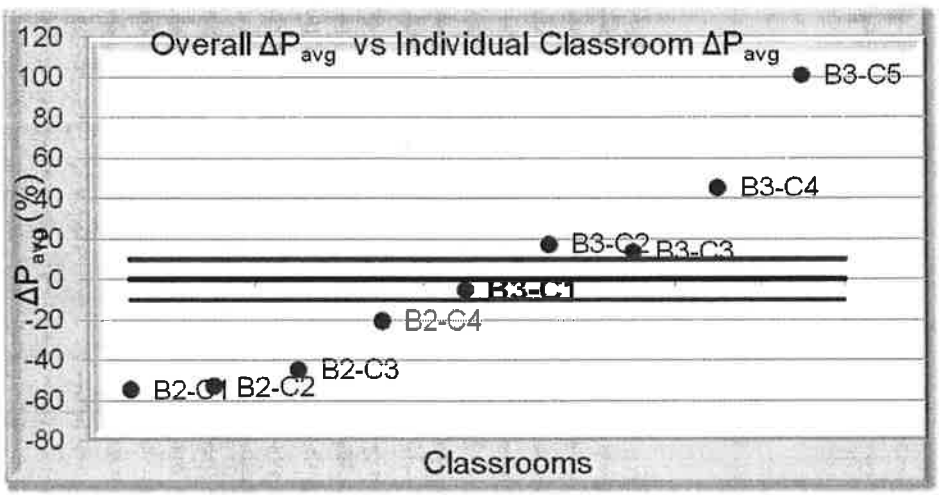
	Average pressure drop (Pa)	Total average pressure drop (Pa)	Lower bound of total average pressure drop (-10% less) (Pa)	Upper bound of total average pressure drop (+10% less) (Pa)	Percentage* (%)
B2-C1	0.806	1.776	1.598	1.953	-54.61
B2-C2	0.837				-52.85
B2-C3	0.985				-44.54
B2-C4	1.413				-20.40
B3-C1	1.684				-5.14
B3-C2	2.085				17.43
B3-C3	2.019				13.71
B3-C4	2.583				45.48
B3-C5	3.568				100.92

* Percentage =(average pressure drop – total average pressure drop)/ total average pressure drop x100%

CFD Methodology for RB - Case study

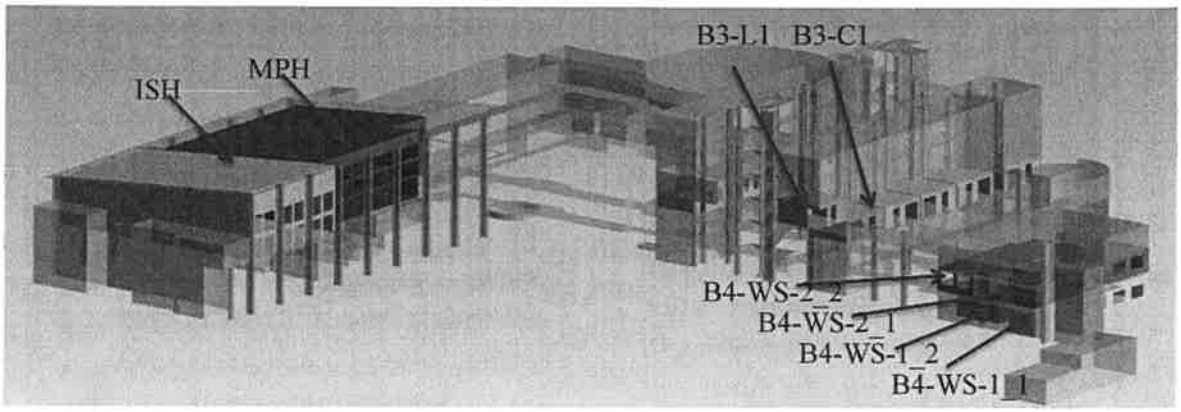
- Development Model
  - ✓ Determination of representative units

$$\Delta p_{avg} = \frac{\sum_{j=1}^{4}\sum_{i=1}^{9}\Delta p_{i,j}}{4\times 9}$$



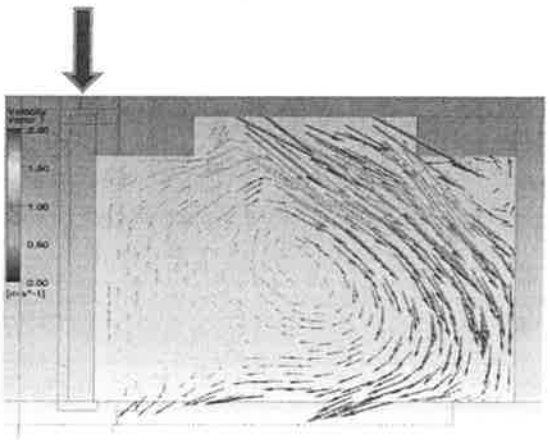
CFD Methodology for RB
- Case study

- Selected Units for Unit Models

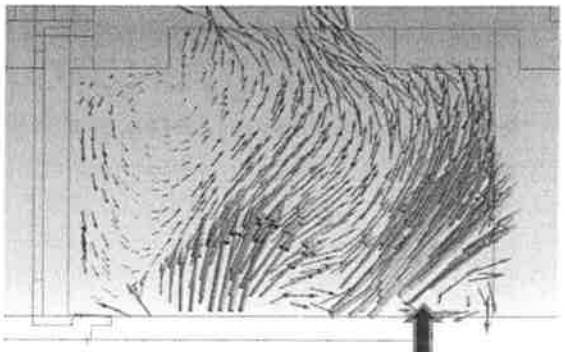


CFD Methodology for RB
- Case study

- Unit Model
  - ✓ Vector plots through Classroom-B3-C1@1.2m above the floor



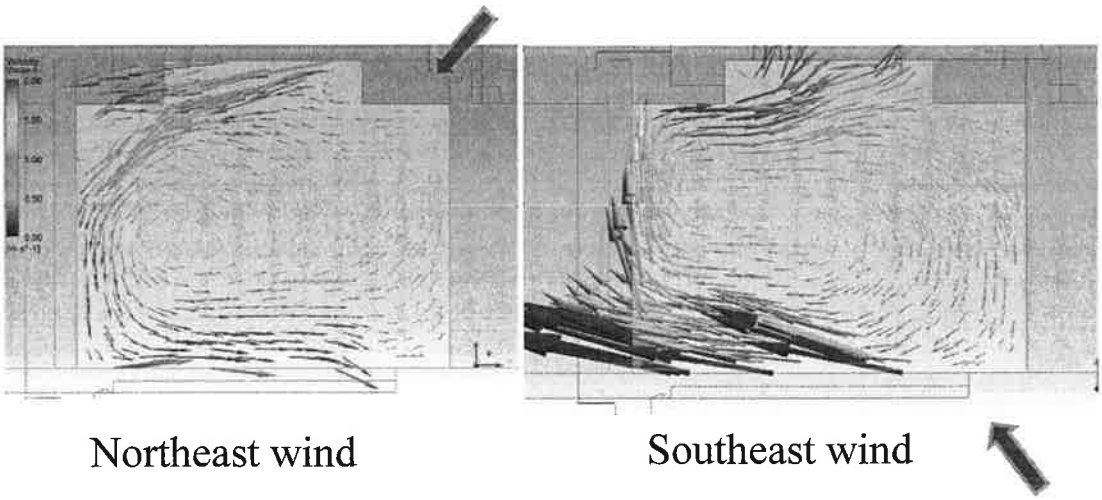
North wind



South wind

CFD Methodology for RB
- Case study

- Unit Model
  - ✓ Vector plots through Classroom-B3-C1@1.2m above the floor

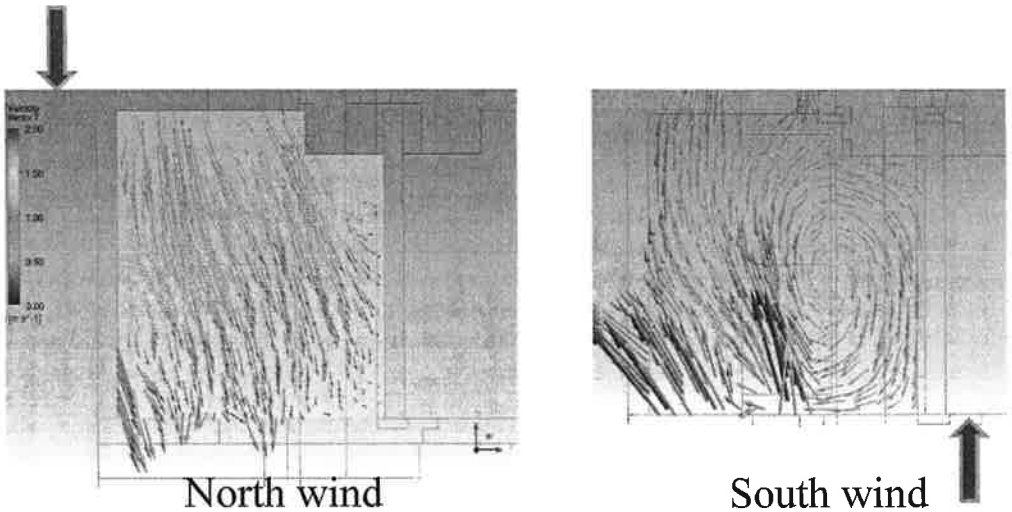


Northeast wind

Southeast wind

CFD Methodology for RB
- Case study

- Unit Model
  - ✓ Vector plots through Classroom-B3-L1@1.2m above the floor

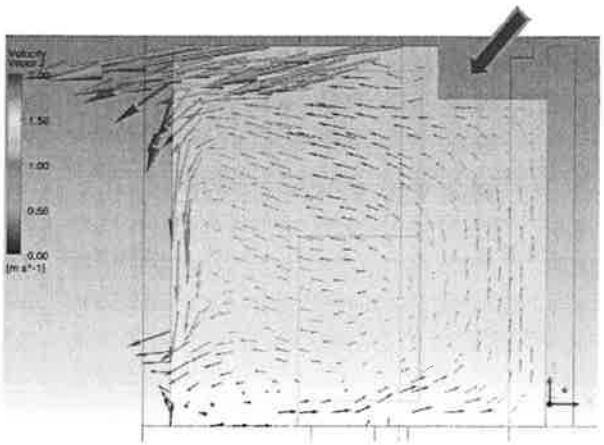


North wind

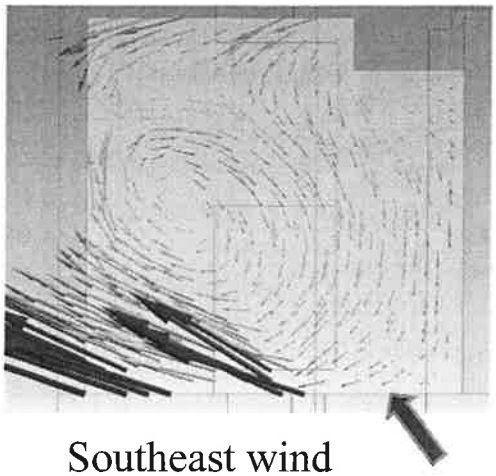
South wind

CFD Methodology for RB
- Case study

- Unit Model
  - ✓ Vector plots through Classroom-B3-L1@1.2m above the floor



Northeast wind



Southeast wind

CFD Methodology for RB
- Case study

- Unit Model
  - ✓ Area-weighted average wind speeds – **Baseline** model
    - – With 2-m wide wall for glazing prevention
    - – 1.1m-height parapet walls at the bottom

Type of Rooms	No of rooms	Speed of Airflow (m/s)					% vs 0.6m/s
		north	northeast	south	southeast	AVG	
Classroom	45	0.17	0.13	0.42	0.49	0.33	57.0%
Language room	3	0.35	0.14	0.69	0.51	0.52	

CFD Methodology for RB - Case study

- Massive efforts to improve ventilation conditions

Model No.	Typical Classrooms				
	Side facing the exterior			Side next to the corridor	
	Parapet wall at the bottom	Louvers sizing	wall for glazing prevention	Parapet wall at the bottom	Louver sizing
Baseline*	1.1m	4.5m w x 1.9m h	2m wide	1.1m	2.6m w x 1.1m h
Improved 1 (IMP-1)*	0m	4.5m w x 3.0m h#	2m wide	1.1m	2.6m w x 1.1m h
Improved 2 (IMP-2)*	0m	4.5m w x 3.0m h#	2m wide	1.1m	2.6m w x 1.9m h
Improved 3 (Final)**	0m	6.2m w x 3.0m h#	2m wide	1.1m	3.6m w x 1.9m h

Fixed louvers are installed at the bottom on the corridor sides, followed with adjustable louvers at the top.

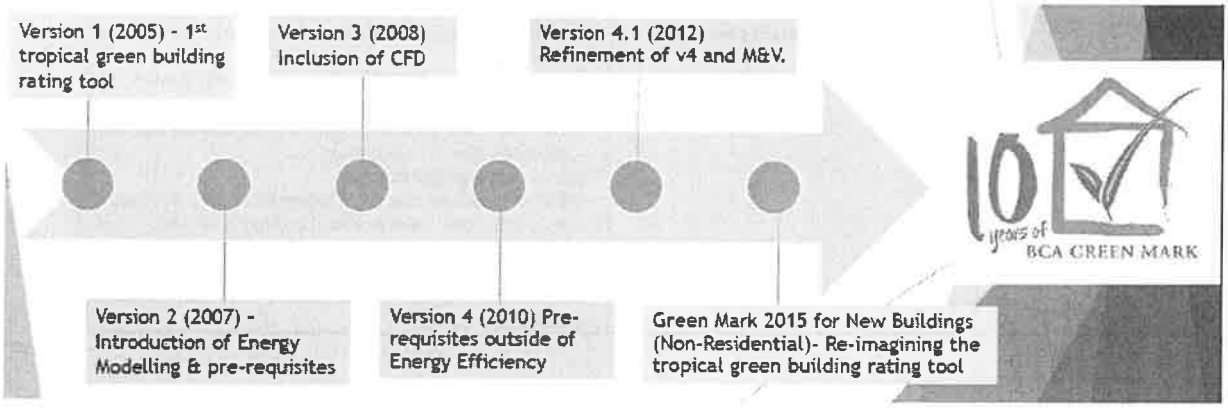
CFD Methodology for RB - Case study

- Area-weighted average wind speeds – Final model
  - With 2-m wide wall for glazing prevention on the south
  - Full-height louver windows on the south (adjustable at the top + fixed louvers at the bottom)
  - Adjustable louver windows plus parapet walls on the north

Type of Rooms	no of rooms	Speed of Airflow [m/s]					Overall percentage (/0.6m/s)
		north	North-east	south	South-east	average	
B3-C1	45	0.26	0.18	0.46	0.86	0.56	89.7%
B3-L1	3	0.42	0.18	0.81	0.51	0.61	
B4-WS-1_1	1	0.05	0.07	0.51	0.30	0.29	
B4-WS-1_2	1	0.05	0.02	0.05	0.08	0.06	
B4-WS-2_1	1	0.07	0.08	0.37	0.26	0.23	
B4-WS-2_2	1	0.19	0.28	0.06	0.43	0.36	

BCA Green Mark Certification Standard

- The BCA Green Mark scheme is a benchmark scheme for evaluating environmental sustainability in buildings starting from 2005



Sharing with Permission from BCA

Green Mark 2015 for NRB

CFD Simulation Methodology & Requirements	Green Mark 4.1 (released 2012)	Green Mark 2015 (released 2016)
General	Ventilation Simulation	Ventilation Simulation, Thermal Comfort, Indoor Air Quality & Wind Driven Rain
CFD Simulation software	Minimum Requirement of using Standard k-epsilon turbulence model, coupled with standard wall function	Recommended to use enhanced RANS eddy viscosity model or Reynolds Stress Model
CFD Computational Domain	Minimum 3 times or more the length of longest distance measured across the boundary of development. The domain height shall be extended, approximately 3 times the height of the tallest building within the defined vicinity	As a general guideline, the direction blockage ratio (BR_L & BR_H) along lateral and vertical direction should be less than 17%. It is also important to ensure that the blockage ratio (BR) arising from the projection of building frontal to the domain enclosure should not be larger than 3%. The surrounding buildings reside within 500 m distance stream wise and span wise from the edge of development of interest should be modelled explicitly. The ground surface beyond surrounding buildings site can be modelled implicitly with Davenport-Wieringa roughness classification

Green Mark 2015 for NRB

CFD Simulation Methodology & Requirements	Green Mark 4.1 (released 2012)	Green Mark 2015 (released 2016)
CFD Grid Size	As a guide, the dimension of the computational element should be set at 0.1 to 0.2 m in the apartment unit, 0.5 to 1.0 m at all buildings and ground level and 10 m at the far field boundary with a minimum of 50 m away from the ground	As a guide, the dimension of the computational elements is advised to follow the following principles (i) To carry out proper domain decomposition to ensure good quality mesh can be obtained. (ii) Hexahedra or prism body-fitted grid are preferred. (iii) Grid independent test shall be performed at the functional space through grid refinements in areas with sharp gradients. (iv) In terms of the computational cell quality, the skewness of the cell is advised no greater than 0.9. (v) The maximum stretching ratio for near building cell size should be kept to be less than 1.4. The recommended grid size are: Within the functional spaces of interest [0.1-0.5m]; Building of interest [0.5-1.0m]; Surrounding building [1.0-5.0m]; From ground surface to 10m height in vertical direction [0.5 – 1.0m]; Ten to H_{max} height in vertical direction, (H_{max} is the height of the tallest building among the group of buildings modeled explicitly) [1.0 – 5.0m]

Green Mark 2015 for NRB

CFD Simulation Methodology & Requirements	Green Mark 4.1 (released 2012)	Green Mark 2015 (released 2016)
CFD Boundary Conditions and Turbulence Modelling	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a
	<table><tr><th>Wind Direction</th><th>Mean Speed (U_{ref}) (m/s)</th></tr><tr><td>North</td><td>2.0</td></tr><tr><td>North-East</td><td>2.9</td></tr><tr><td>South</td><td>2.8</td></tr><tr><td>South-East</td><td>3.2</td></tr></table> The inbound vertical wind profile shall assume to be given by the Logarithmic Law with reference height at 15.0m.	<table><tr><th>Wind Direction</th><th>Mean Speed (U_{ref}) (m/s)</th></tr><tr><td>North</td><td>2.0</td></tr><tr><td>North-East</td><td>2.9</td></tr><tr><td>South</td><td>2.8</td></tr><tr><td>South-East</td><td>3.2</td></tr></table> The inbound vertical wind profile shall assume to be given by the Logarithmic Law with reference height at 15.0m. The wind profile shall be determined by using following equations $U(z) = \frac{u^*_{ABL}}{\kappa} \ln\left(\frac{z+z_0}{z_0}\right)$ (Eq.1) $k(z) = \frac{u^{*2}_{ABL}}{\sqrt{C_\mu}}$ (Eq.2) $\varepsilon(z) = \frac{u^{*3}_{ABL}}{\kappa(z+z_0)}$ (Eq.3) $u^*_{ABL} = \frac{U_{ref}\,\kappa}{\ln\left(\frac{h+z_0}{z_0}\right)}$ (Eq.4) where u^*_{ABL} is the atmospheric boundary layer (ABL) friction velocity, κ is the von Karman constant (0.42) and C_μ is a

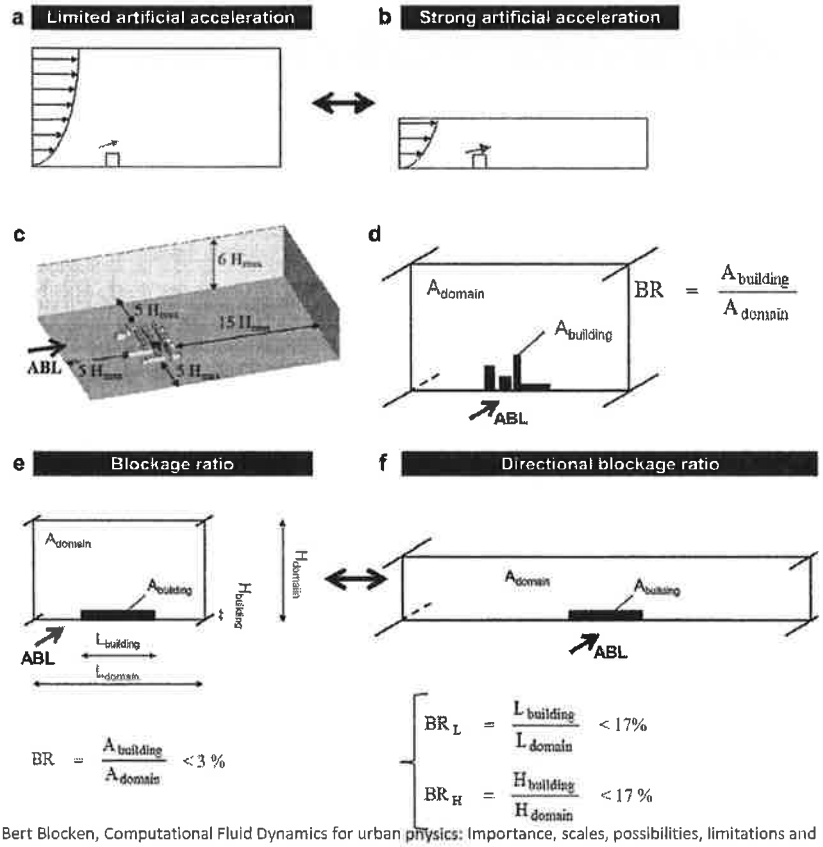
Green Mark 2015 for NRB

CFD Simulation Methodology & Requirements	Green Mark 4.1 (released 2012)	Green Mark 2015 (released 2016)
CFD Boundary Conditions and Turbulence Modelling	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a
		constant, generally taken equal to 0.09, z_0 is aerodynamic roughness length, U_{ref} is the specified velocity at reference height h. The aerodynamic roughness length z_0 for wind profile should be selected from Table C3.3b [Updated Davenport-Wieringa Roughness Classification] to match the terrain category of the development site of interest, including the tree/greenery effect (b) *Ground Surface* Using appropriate roughness parameters is an essential component of accurate simulation of Atmospheric Boundary Layer (ABL) flow. There are two types of roughness parameter that shall be applied on difference surface areas as listed in Table C3.3c [Roughness Parameter Setting]. (i) Aerodynamic roughness length z_0. (ii) Equivalent sand-grain roughness height k_S The region of inlet, approach and incident flow at the upstream of computational domain should be modelled with appropriate aerodynamics roughness length z_0 as well as the relationship between equivalent sand-grain roughness height ks with the corresponding aerodynamics roughness length z_0. (c) *Top and Lateral Surface of Domain* Use zero velocity gradients and zero normal gradients, i.e.

Green Mark 2015 for NRB

CFD Simulation Methodology & Requirements	Green Mark 4.1 (released 2012)	Green Mark 2015 (released 2016)
CFD Boundary Conditions and Turbulence Modelling	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a	(a) *Inlet Atmospheric Boundary Condition* The prevailing wind condition such as the mean speed and direction for Singapore shall be taken from meteorological data, Table C3.3a
		(c) *Top and Lateral Surface of Domain* Use zero velocity gradients and zero normal gradients, i.e. "symmetry" condition, for all variables at the top and lateral surface when the top and lateral boundaries of the domain are far away enough from the buildings (refer to the requirements on the domain size). (d) Outlet Surface of Domain Use zero static pressure as the boundary condition at the outlet surface of computational domain
CFD Discretization Schemes	N/A	In all circumstances, the users should attempt to apply 2nd order discretization schemes, which are preferred over 1st order discretization schemes to avoid numerical diffusion
CFD Convergence Criteria	N/A	To ensure the changes in solution variables from one iteration to the next are negligible through residuals monitoring with at least 4 orders of magnitudes is achieved. In addition, monitoring points should be defined in the region of interest and the velocities in those points should be recorded to ensure that the flow has reached steady values when simulation is converged properly

CFD Computational Domain for NV

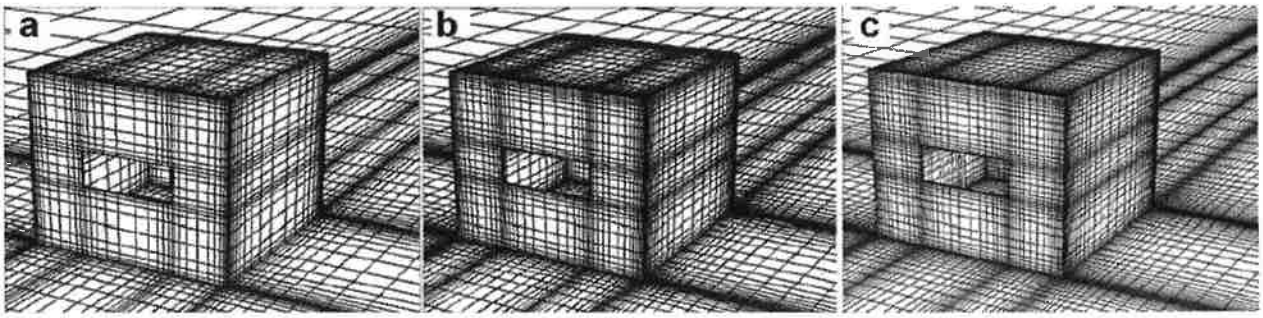


1. Minimum distance between urban/building models and boundary of domain
2. Maximum allowable blockage ratio
3. Combination of 1 & 2

- At this juncture, it is proposed to have the inclusion of surrounding buildings to be the same as GM4 (3 times of development length)

Bert Blocken, Computational Fluid Dynamics for urban physics: Importance, scales, possibilities, limitations and ten tips and tricks towards accurate and reliable simulations, Building & Environment, 2015, 1 – 27

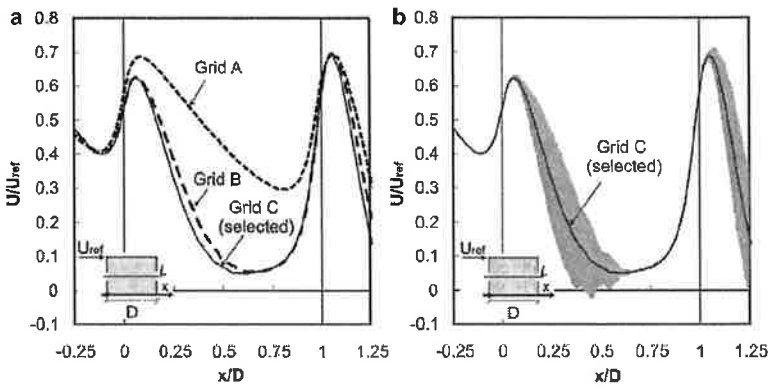
CFD Mesh Independence Study for NV



Course Grid A – 0.2 million cells

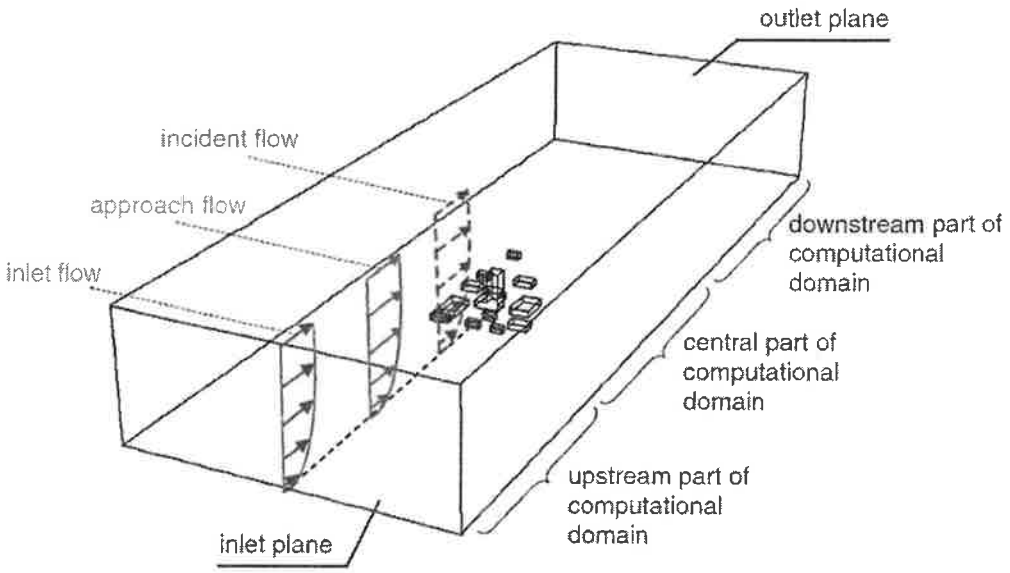
Medium Grid B – 0.3 million cells

Fine Grid C – 0.5 million cells



R. Ramponi, B. Blocken, CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters, Building and Environment 53 (2012) 34-48

ABL setting for NV

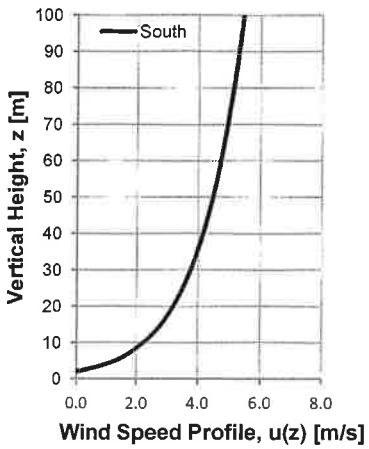


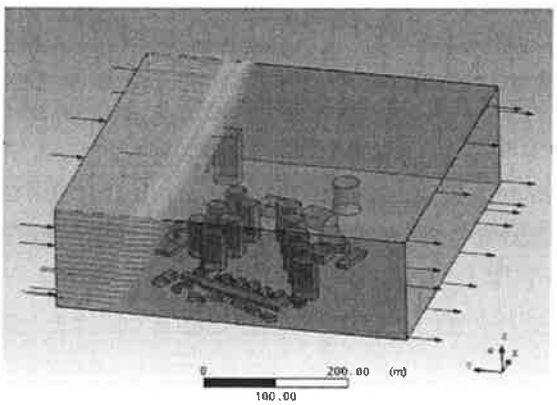
- ✓ Ensure horizontal homogeneity in the upstream and downstream domains.
- ✓ The vertical mean wind speed and turbulence profiles are in *"equilibrium"* with the roughness characteristics of the ground surface.

Natural Ventilation - Wind Speed

$$u(z) = u_{ref} \frac{\ln\left(\frac{z}{z_0}\right)}{\ln\left(\frac{z_{ref}}{z_0}\right)}$$

Wind Direction	Mean Velocity (U_{ref}) (m/s)
North	2.0
North-East	2.9
South	2.8
South-East	3.2

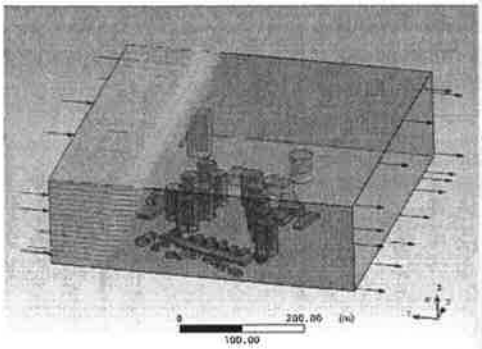




115

Natural Ventilation - Wind Speed

$$u(z) = u_{ref} \frac{\ln\left(\frac{z}{z_0}\right)}{\ln\left(\frac{z_{ref}}{z_0}\right)}$$



Aerodynamic roughness, Z_0

z_0 (m)	Landscape Description
0.0002 Sea	Open sea or lake (irrespective of the wave size), tidal flat, snow-covered flat plain, featureless desert, tarmac, concrete, with a free fetch of several kilometres
0.005 Smooth	Featureless land surface without any noticeable obstacles and with negligible vegetation; e.g. beaches, pack ice without large ridges, moraas, and snow-covered or fallow open country
0.03 Open	Level country with low vegetation (e.g. grass) and isolated obstacles with separations of at least 50 obstacle heights; e.g. grazing land without windbreaks, heather, moor and tundra, runway area of airports.
0.10 Roughly open	Cultivated area with regular cover of low crops, or moderately open country with occasional obstacles (e.g. low hedges, single rows of trees, isolated farms) at relative horizontal distances of at least 20 obstacle heights.
0.25 Rough	Recently-developed "young" landscape with high crops or crops of varying height, and scattered obstacles (e.g. dense shelterbelts, vineyards) at relative distances of about 15 obstacle heights.
0.50 Very rough	"Old" cultivated landscape with many rather large obstacle groups (large farms, clumps of forest) separated by open spaces of about 10 obstacle heights. Also low large vegetation with small interspaces such as bush land, orchards, young densely-planted forest.
1.0 Closed	Landscape totally and quite regularly covered with similar-size large obstacles, with open spaces comparable to the obstacle heights; e.g. mature regular forests, homogeneous cities or villages.
≥ 2.0 Chaotic	Centres of large towns with mixture of low-rise and high-rise buildings. Also irregular large forests with many clearings.

Singapore

116

Recommended Meshing Methodology

- Recommended mesh sizing distribution

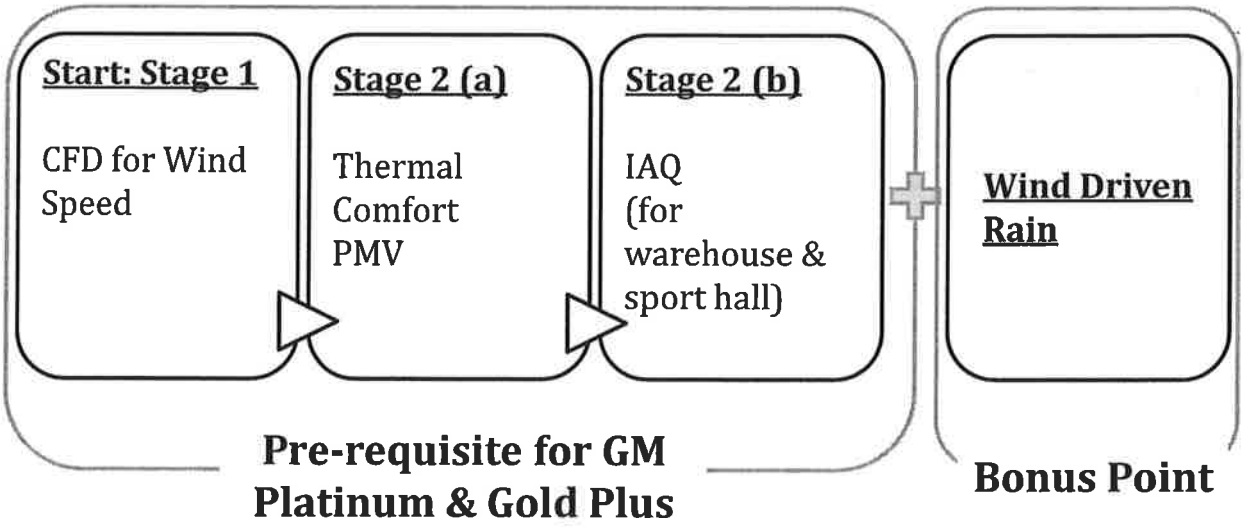
Location	Grid Size (m)
Within the functional spaces of interest	0.1-0.5
Building of interest	0.5-1.0
Surrounding building	1.0-5.0
From ground surface to 10m height in vertical direction	0.5 – 1.0
From 10m to H_{max} height along vertical direction	1.0 – 5.0

*H_{max} is the height of the tallest building among the group of building modelled explicitly)

- Recommended meshing strategy
  - ✓ To use hexahedral cells in the rectangular domain of NV space.
  - ✓ Tetrahedral cells can be used to model the surrounding site features of the NV space.
  - ✓ Hybrid pyramid or cut-cell mesh can be adopted at the interface.

Assessment Criteria

GM 2015 - 1.03c: Ventilation Simulation



Assessment Criteria

- GM 2015 for NRB

Points	Wind Speed Result	Thermal Comfort	Air Quality
3	Moderate (0.2m/s)*	-	-
4	Good (0.4m/s)	-1.0 <PMV <+1.0	Air Change Rate ≥4 Air Exchange Effectiveness of ≥ 1.0
(Platinum)	Very Good (0.6m/s)	-0.8 <PMV <+0.8	Air Change Rate ≥10 Air Exchange Effectiveness of ≥ 1.2

* Area average speed at 1.2m above the ground for at least 70% of NV occupied spaces; Needed for both Gold^{PLUS} and Platinum rating design

- Neutral comfort zone: 80% of acceptable thermal comfort condition
- ASHRAE (2004) and Singapore weather condition

Neutral comfort condition can be provided for

- ≈ 80% of the year, > 0.6 m/s
- ≈ 75% of the year, > 0.4 m/s
- ≈ 70% of the year, > 0.2 m/s

(Ref: Wang & Wong (2007))

Assessment Criteria

- GM 2015 for NRB
  - Thermal Comfort (PMV)
    - Applicable for mechanical-assisted natural ventilation

Building type	PMV equation
Industrial buildings	PMV = -4.974 + 0.202 DBT - 0.181 WIND
Hospitals	PMV = -8.405 + 0.322 DBT - 0.686 WIND

  - Indoor Air Quality (ACH, AEE)

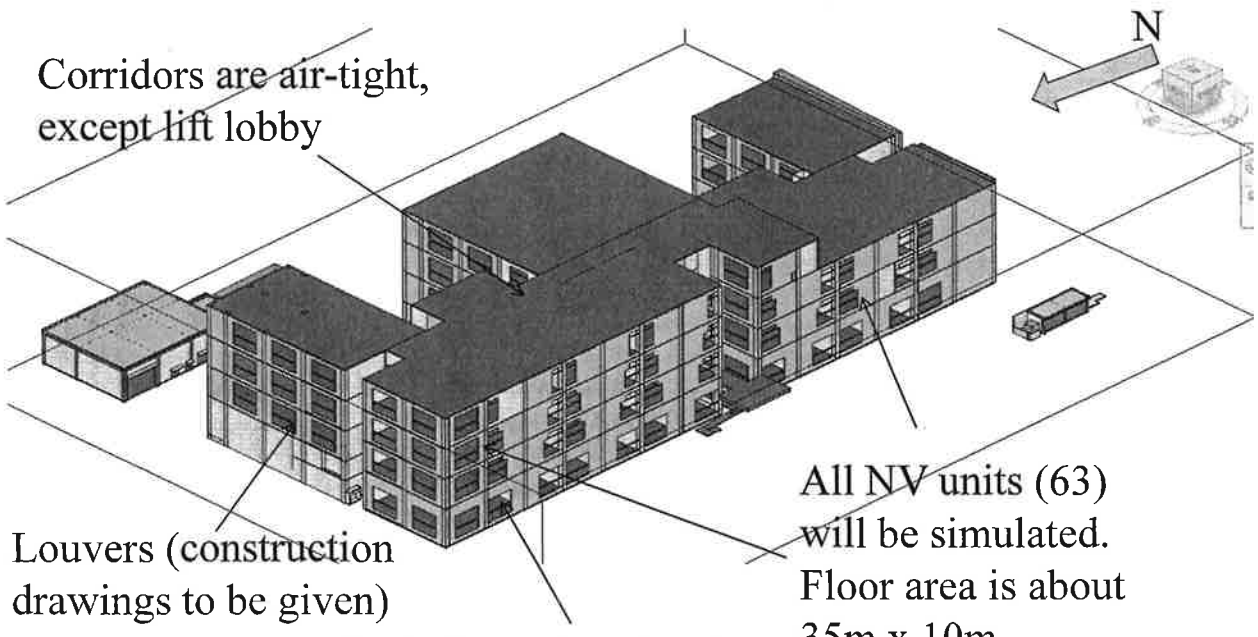
$$ACH = \frac{3600\,Q}{Vol},$$

$$\varepsilon_a = \frac{\frac{1}{\sum A_i}\sum C_{i,out}A_i}{C_{eq}}$$

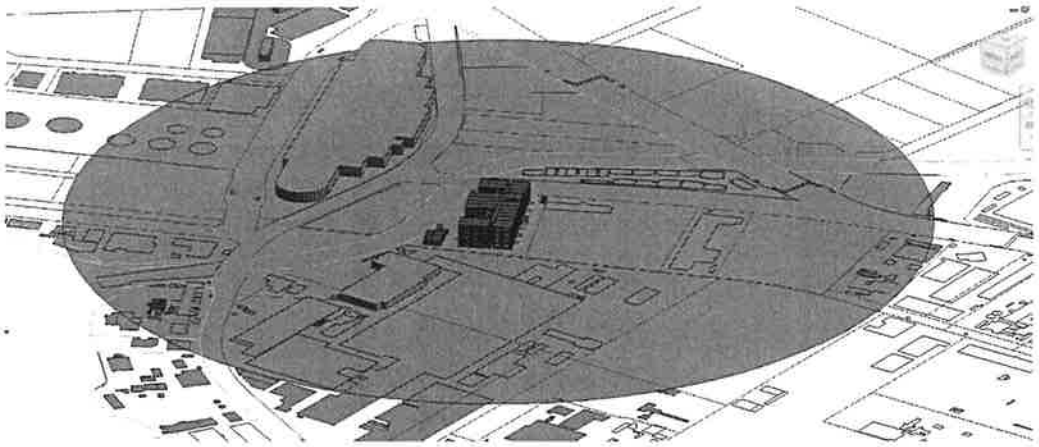
where

- Q is the air flow through the functional units (m^3/s)
- Vol is the volume of the room (m^3)
- ε_a is air exchange effectiveness
- A_i is face element area at the out-flow openings
- $C_{i,out}$ is the concentration of tracer gas in the outflow air
- C_{eq} is average tracer gas concentration in the room when steady state reached, which represent the equilibrium conditions

Test Case Study for NRB



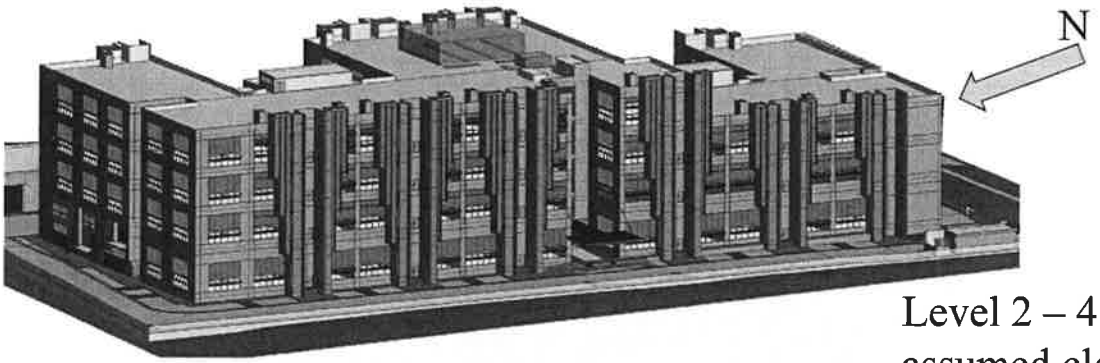
Surrounding Buildings & ABL Flow



Atmospheric Boundary Layer simulation:

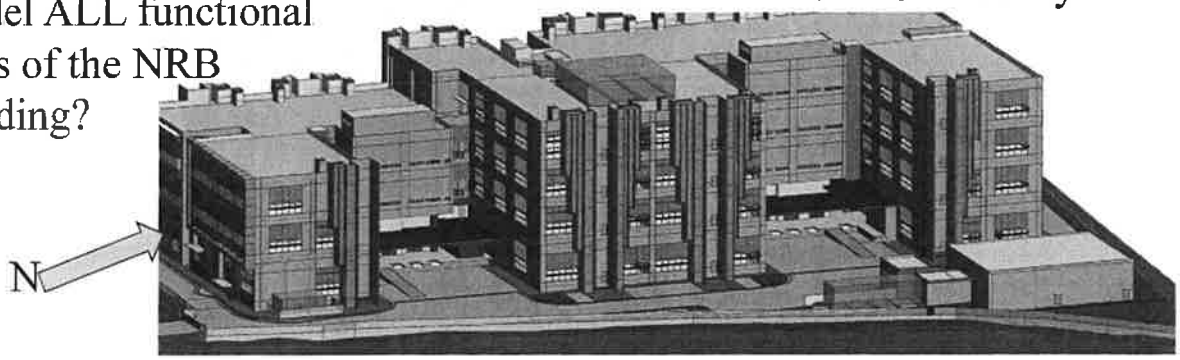
1. Horizontal ABL homogeneity ABL in upstream and downstream
2. Sufficiently high mesh resolution in vertical direction near ground
3. First cell center point (y_p) to be larger than physical roughness height (k_s)
4. Relationship between equivalent sand-grain roughness height (k_s) and corresponding aerodynamic roughness height, y_o

Geometrical Description – Whole Building



Level 2 – 4 is assumed closed for this CFD study

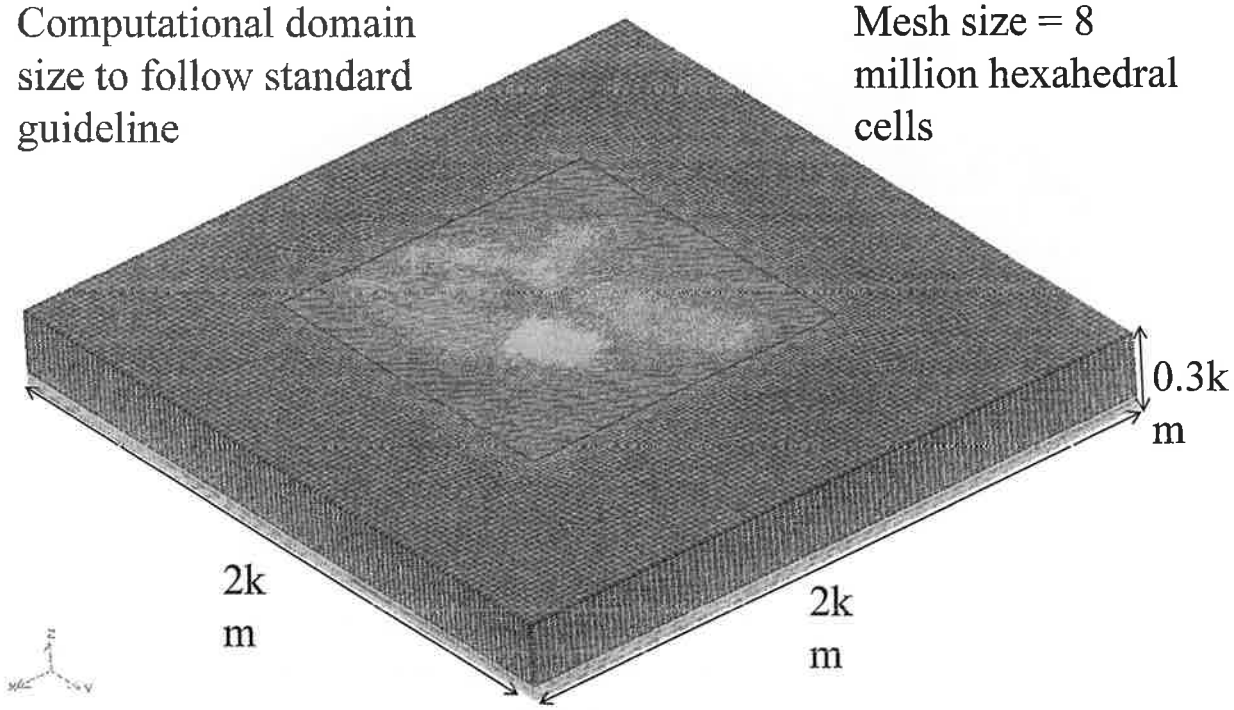
Questions: Should we model ALL functional units of the NRB building?



Computational Domain Decomposition & Meshing

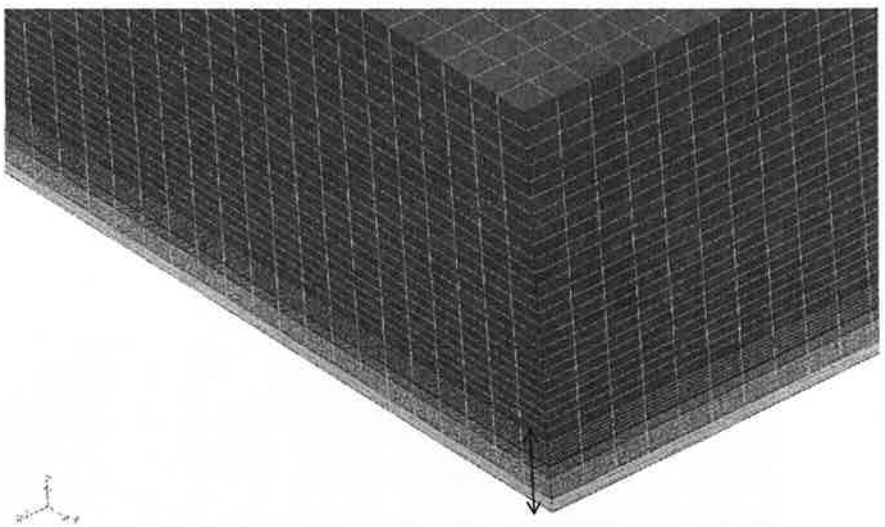
Computational domain size to follow standard guideline

Mesh size = 8 million hexahedral cells



Computational Domain Decomposition & Meshing

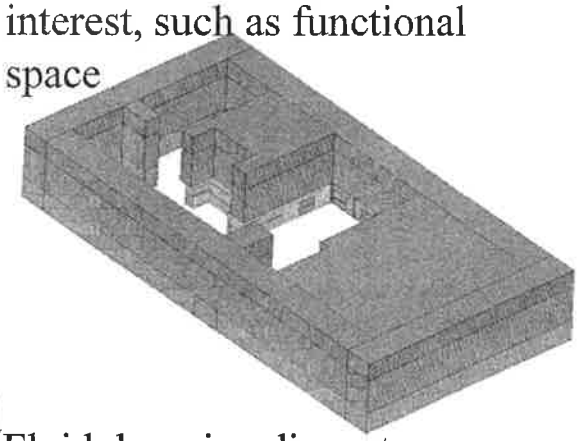
Sufficiently high mesh resolution in vertical direction near ground is necessary in order to capture the ABL flow



0.5m cell size for 10m height; and max 2m cell size up to 50m height

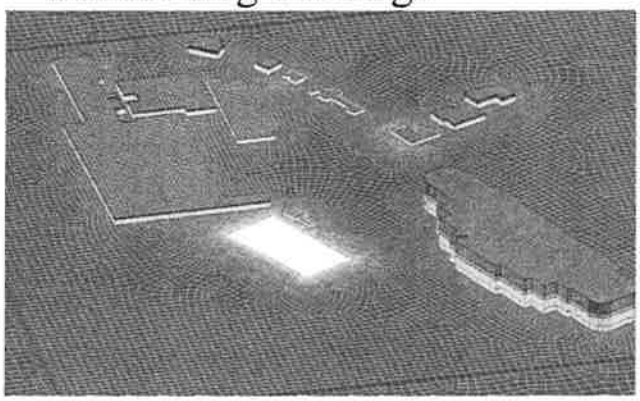
Computational Domain Decomposition & Meshing

To ensure proper domain decomposition and high quality cells (e.g. hexahedral) being applied at the domain of interest, such as functional space



Fluid domain adjacent to functional space

Proper domain decomposition for the fluid domain adjacent to functional unit & windows; and mesh size distribution for surrounding buildings



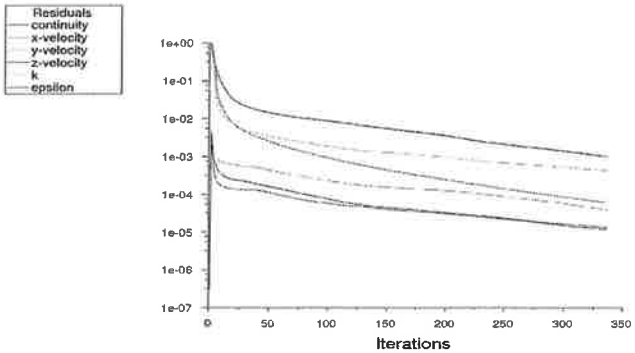
5m mesh size for lateral side of surroundings buildings

Preliminary Simulation Results – Convergence Criteria

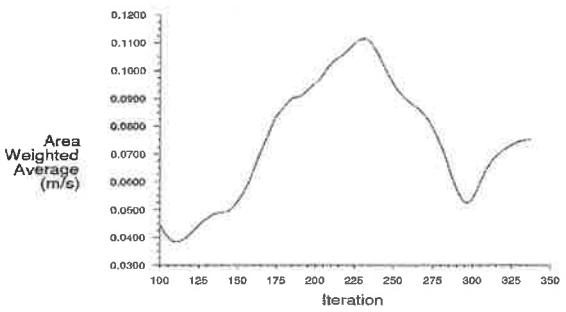
To ensure "real" convergence is achieved; and in all circumstance try to use second order discretization scheme

Convergence is reached when:

- Changes in solution variables from one iteration to the next are negligible.
  - Residuals provide a mechanism to help monitor this trend.
- Overall property conservation is achieved
  - Imbalances measure global conservation
- Quantities of interest (e.g. drag, pressure drop) have reach steady values.
  - Monitor points track quantities of interest



Residual, converged after 340 iterations

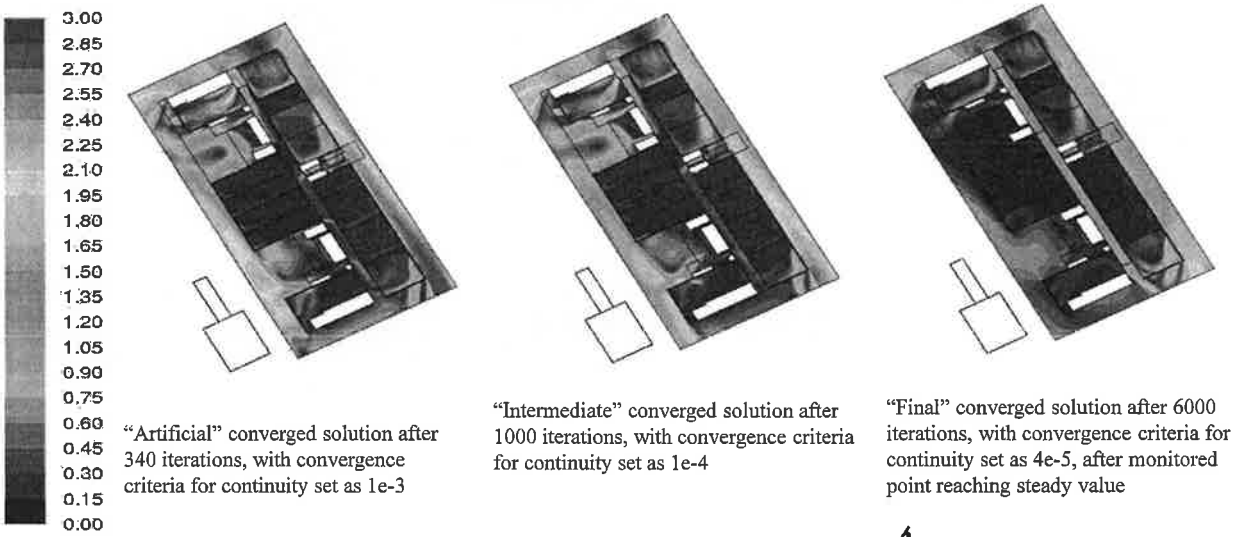


However, monitor point value is still fluctuating

Preliminary Simulation Results – Velocity @ 1.5m height

Good CFD setting with the proper convergence criteria; especially with 2nd order discretization scheme

- Proper convergence criteria (e.g. residual setting & monitor quantity of interest) is crucial to ensure good accuracy results
- How about second order discretization scheme?



"Artificial" converged solution after 340 iterations, with convergence criteria for continuity set as 1e-3

"Intermediate" converged solution after 1000 iterations, with convergence criteria for continuity set as 1e-4

"Final" converged solution after 6000 iterations, with convergence criteria for continuity set as 4e-5, after monitored point reaching steady value

↑
2nd order accurate

Checklist for Ventilation Simulation (Brief)

Checklist Items	Description/Selection	Response & Criteria
• Submission Details	Client, architect, ESD consultants	Info on NV design
• Building Type	industrial facilities etc	Describe building functionality
• Problem Statement	Objective & Work Scope	Describe NV challenges, proposed solution, desired outcome and work scopes from the simulation model.
• Site Information	3 times development length	Surrounding buildings, terrain, greenery
• CFD Approach	Simulation Methodology	Choose well validated software with latest version,
• CFD Approach	Assumption & Simplification	Describe simulation model assumption, limitations and geometrical simplification
• CFD Domain	Computational Domain	Criteria on extent of CFD domain - blockage ratio and domain extension
• CFD Meshing	Mesh independence	Mesh independence study at the functional unit NV space
• CFD Meshing	Mesh size, distribution and quality	Proper domain decomposition. To use hexahedral cells if possible; 0.1 to 0.5 m within the functional space of interest

Checklist for Ventilation Simulation

Checklist Items	Description/Selection	Response & Criteria
• CFD Model	Atmospheric Boundary Layer	CFD simulation with recommended ABL criteria
	Buoyancy	To use Boussinesq or variable density and check gravity direction, if thermal simulation is performed
	Turbulence model	To use the steady Reynolds-Average Navier Stokes (RANS), with minimum requirement k-ε turbulence model equation
	Wall treatment	To ensure wall functions producing y+ values that is satisfying the log-law fully turbulent zone: 40 < y+
• CFD Boundary Conditions	Inlet wind profile	The vertical profile for wind velocity and turbulence in the ABL should be modelled by assuming constant shear stress with height
	Top & lateral sides of domain	To use zero velocity gradients and zero normal gradients of all variables
	Outlet plane	To use zero static pressure
	Window modelling	To use actual window opening size. Attach window schedule and drawing for verification
	Mechanical fan modelling	To use fan input with appropriate swirl components

Checklist for Ventilation Simulation

Checklist Items	Description/Selection	Response & Criteria
• CFD Numerical	Discretization scheme	To use 2nd order for momentum equations
	Convergence criteria	To ensure solution is converged and monitored points at functional space reach steady values
• User		Training, experience, ask expert
• Documentation		Full documentation of parameters, readable scale
• Design Iteration	Baseline case Modified case Optimal case	Baseline case – fixed building massing and layout Modified case – to highlight improvement on NV design Optimal case – final design with incorporation of NV features or passive innovative idea

Concluding Remarks

- Improvements/modification have been made to current GM Ventilation Simulation Methodology
  - Domain size specification – follow the standard guideline
  - Mesh quality – good mesh and cell size requirement
  - ABL simulation with the near ground surface fine mesh distribution
  - Tree canopy modelling
  - Device simulation specification, including mechanical fans, heat source & louver
  - Check list to ensure CFD quality & track NV design changes
- Ongoing process to gather feedback from stakeholder and task force committees
- Can a truly integrated BPS platform be constructed that is able to represent real world complexity? Or will the problems derived from over-simplified tools continue to be pushed on to unsuspecting users? Only time will tell. [Clarke & Hensen, Building & Environment (91), 2015, 294 – 306]

CFD software

General Purposed CFD software

1. ANSYS-FLUENT (http://www.ansys.com/Products/Fluids/ANSYS-Fluent)
2. NUMECA (http://www.numeca.com/)
3. STAR-CCM+ (http://www.cd-adapco.com/)
4. CFD-ACE+ (http://www.cfdrc.com/index.html)
5. FLOW3D (http://www.flow3d.com/)
6. AutoDesk CFD (http://www.autodesk.com/products/cfd/overview)
7. FLUIDYN (http://www.fluidyn.com/)
8. ACUSIM (http://www.acusim.com/)
9. COMSOL (http://www.comsol.com/)
10. UrbaWind (http://www.meteodyn.com)
11. Envi-Met (http://www.envi-met.com/)

CFD Ventilation software

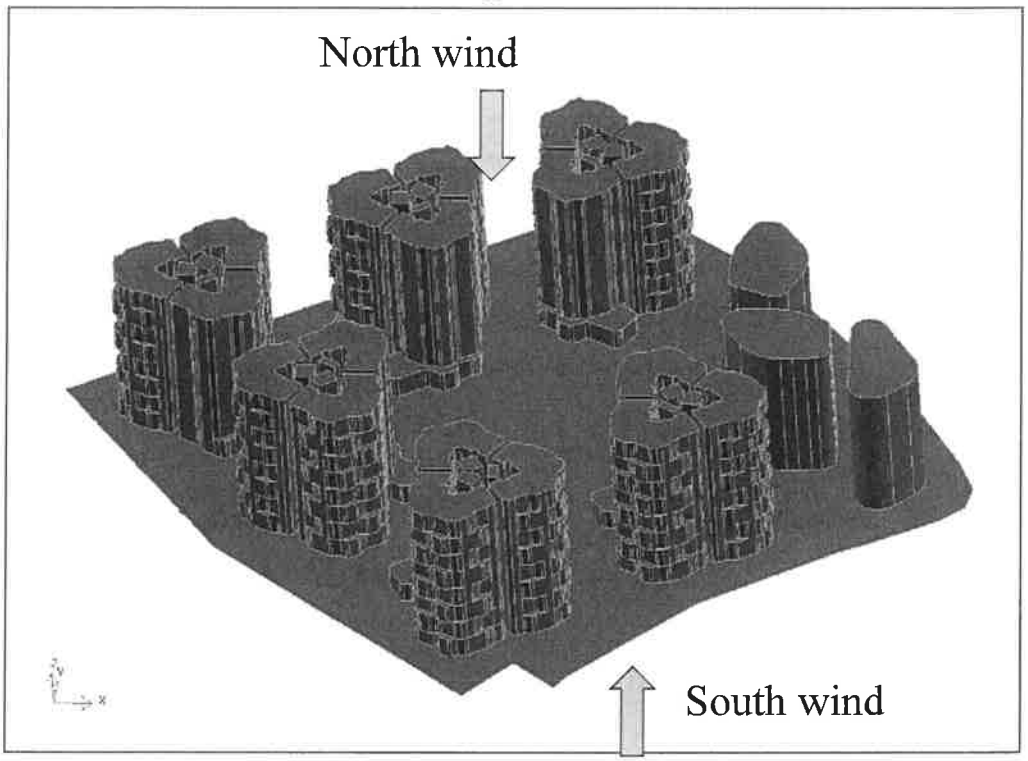
- VORTEX
- AIRPAK
- FLOVENT
- GrBEST

CFD Fire software

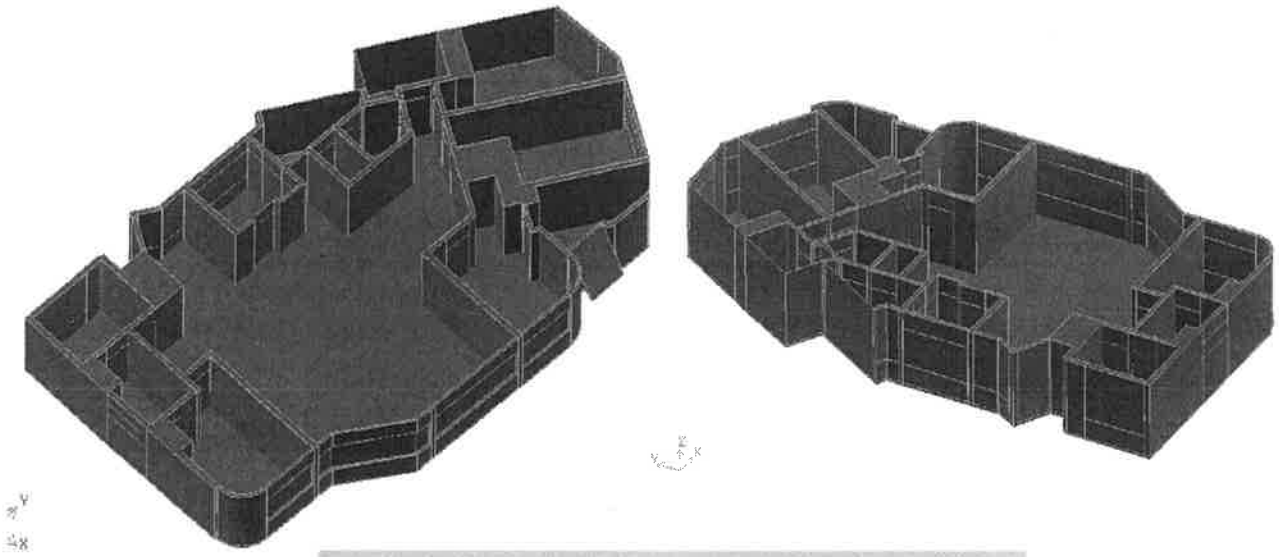
- Smart-Fire (http://fseg.gre.ac.uk/smartfire/)
- JASMINE
- FDS (http://www.fire.nist.gov/fds/)
- SOFIE
  http://wwwlegacy.cranfield.ac.uk/sme/sofie/

Showcases about CFD application in Green Mark building design

CFD simulation for RB building – Example A

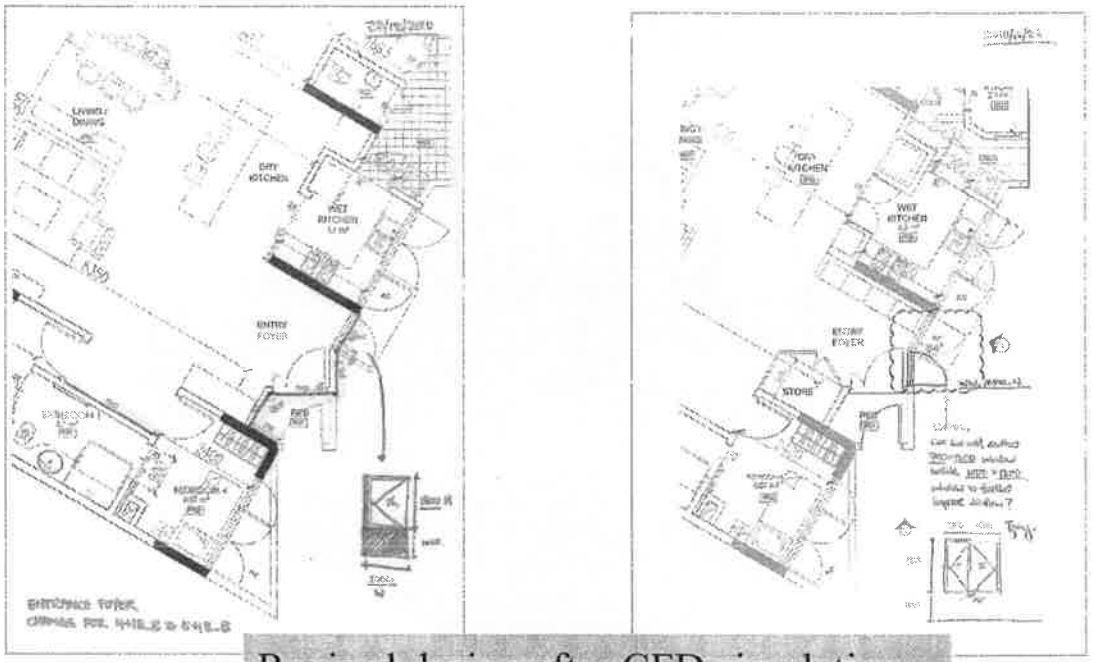


CFD simulation for RB building – Example A



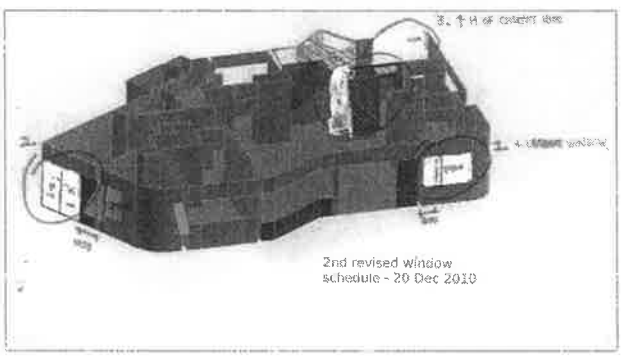
Original design after CFD simulation

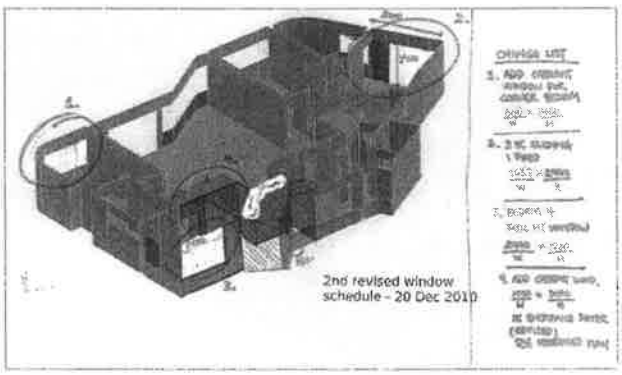
CFD simulation for RB building – Example A



Revised design after CFD simulation

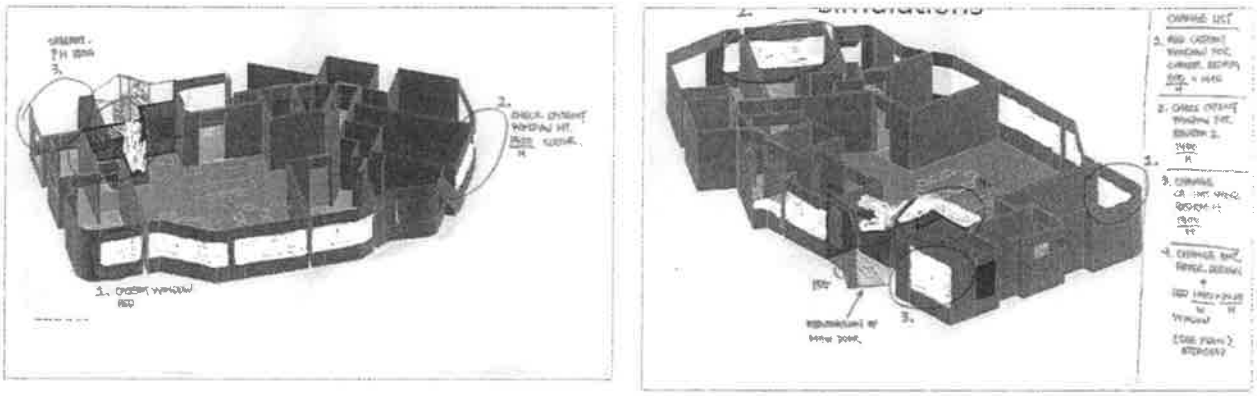
CFD simulation for RB building – Example A





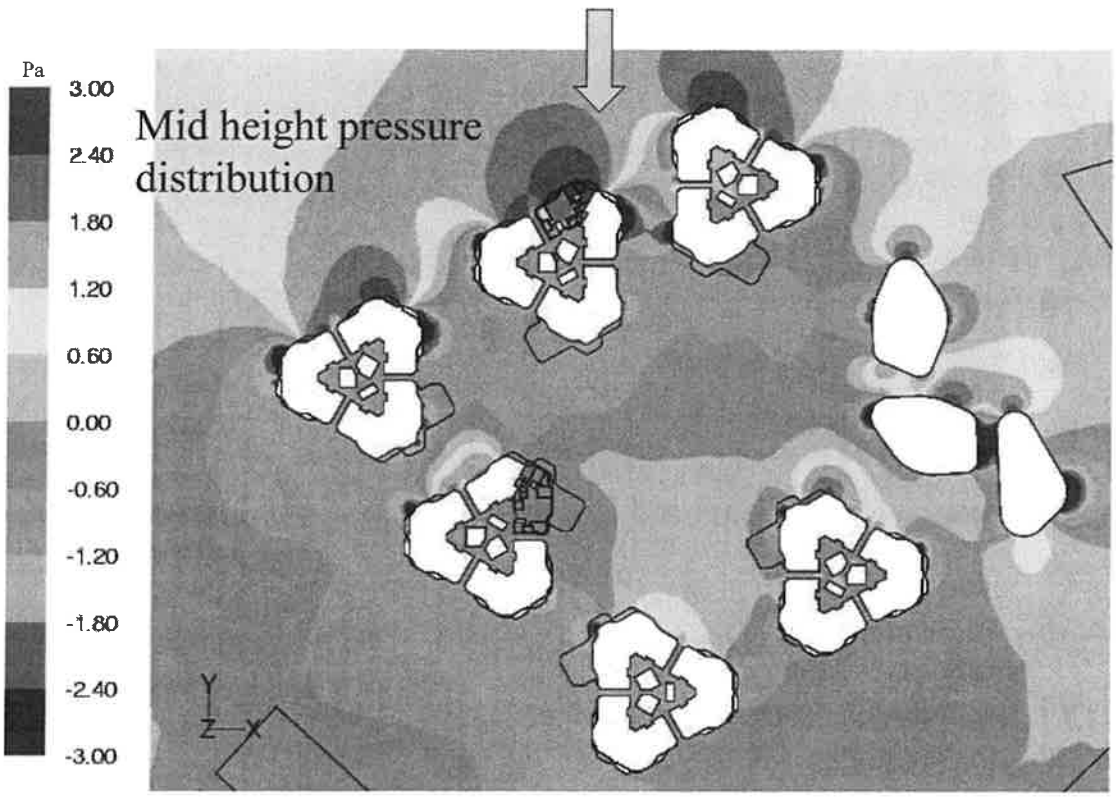
Revised design after CFD simulation

CFD simulation for RB building – Example A

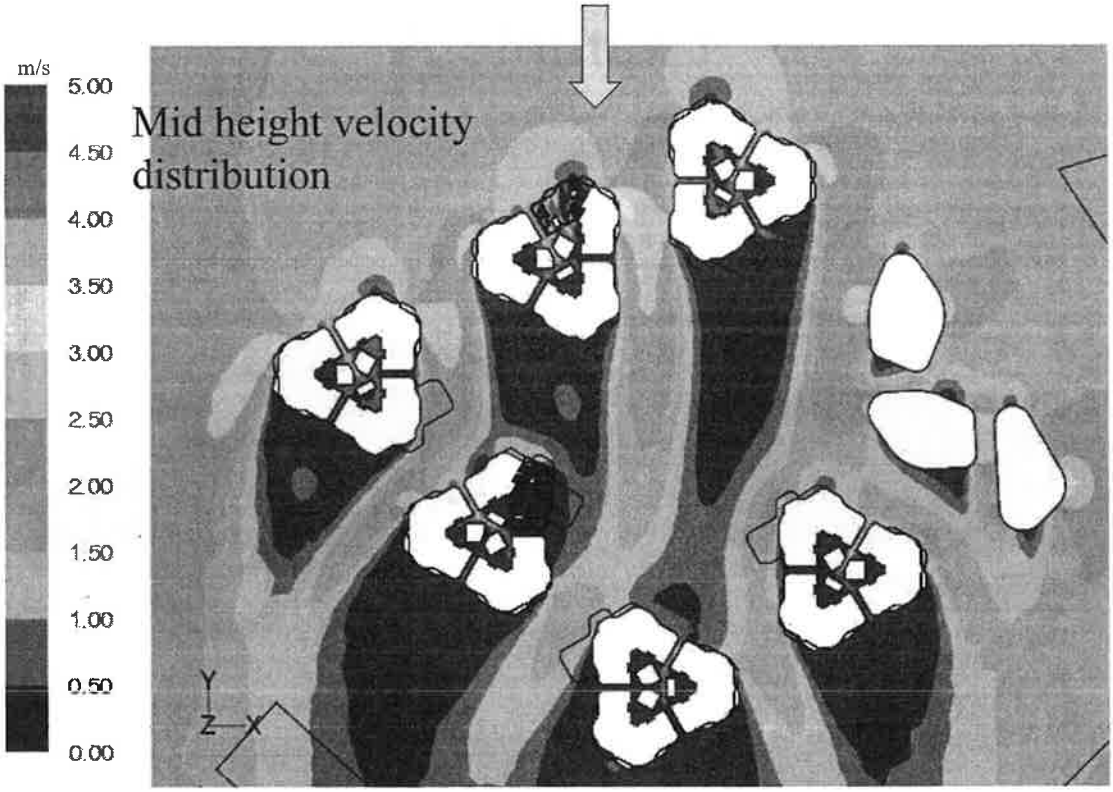


Revised design after CFD simulation

North wind

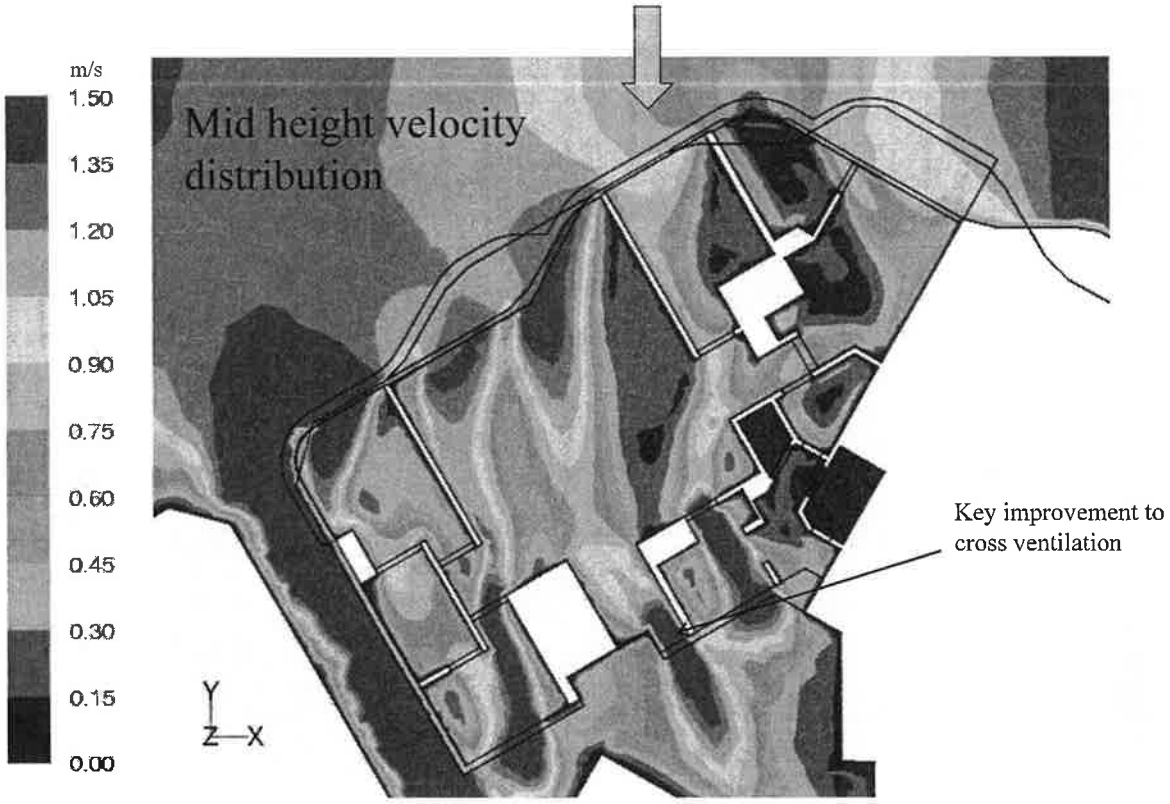


North wind



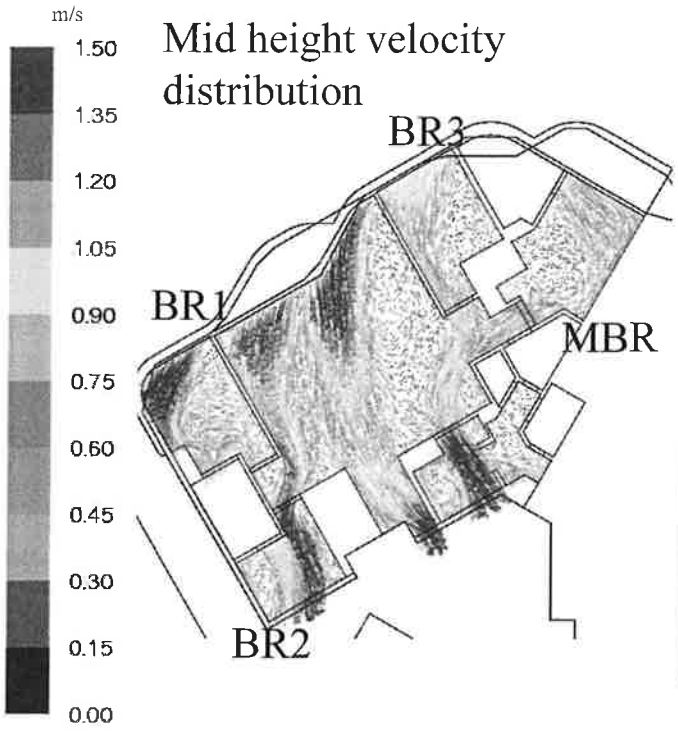
144

North wind



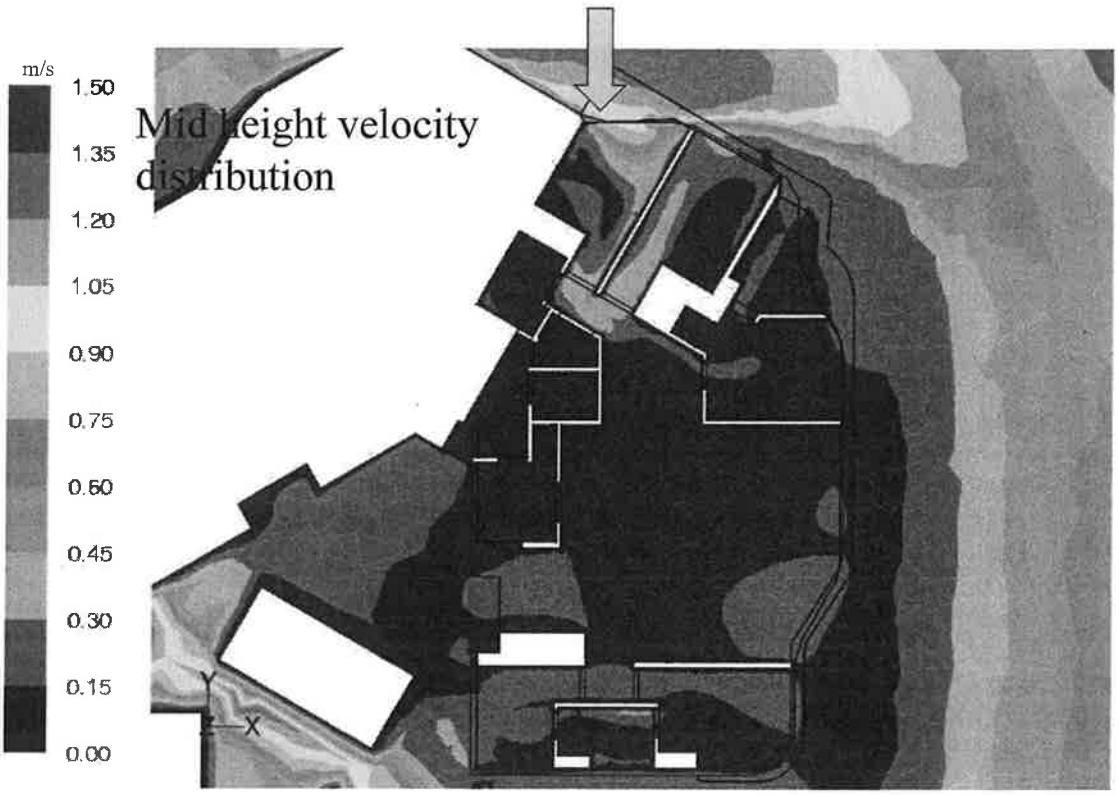
145

North wind

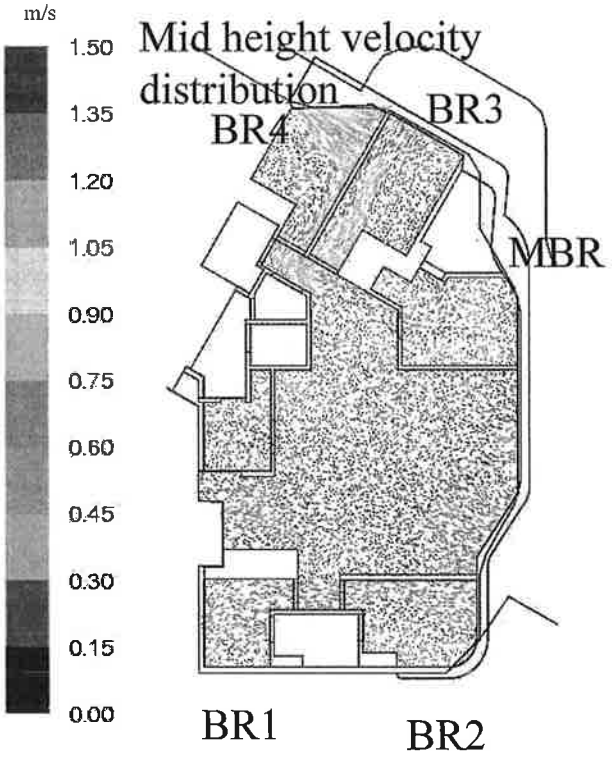


	Average velocity (m/s)
Kitchen	0.72
Living/Dining	0.76
Bedroom 1	0.75
Bedroom 2	0.98
Bedroom 3	0.55
Bedroom MBR	0.41
Average	0.70

North wind

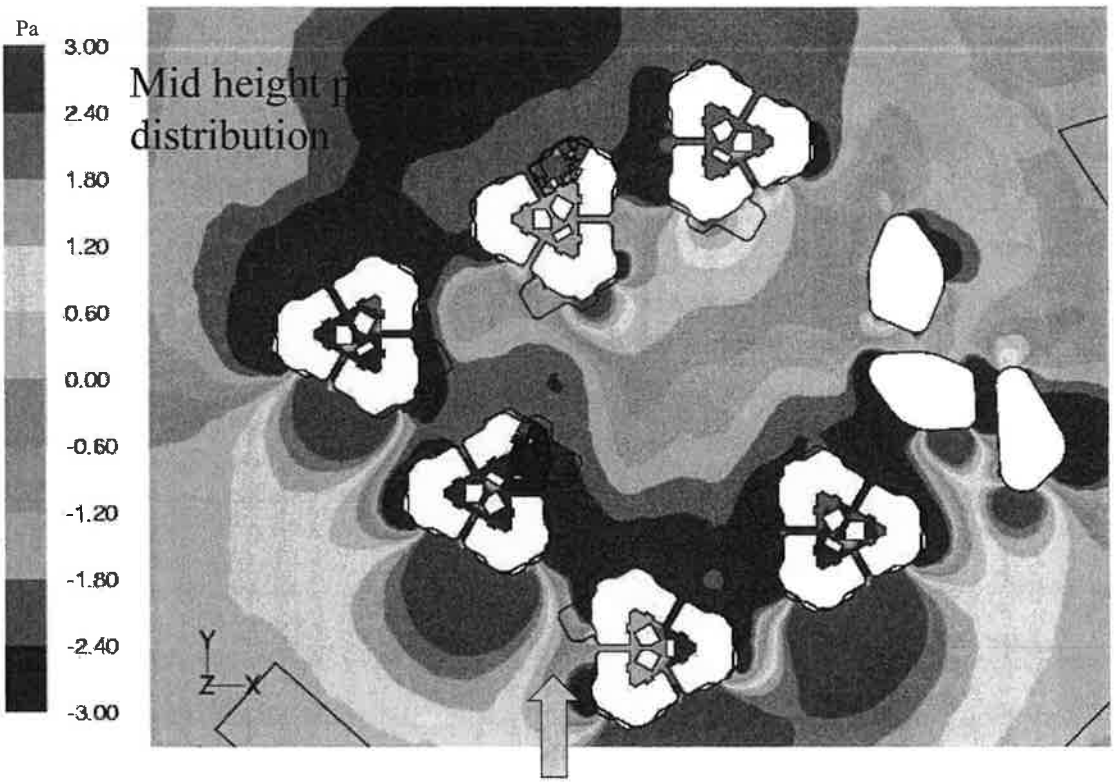


North wind

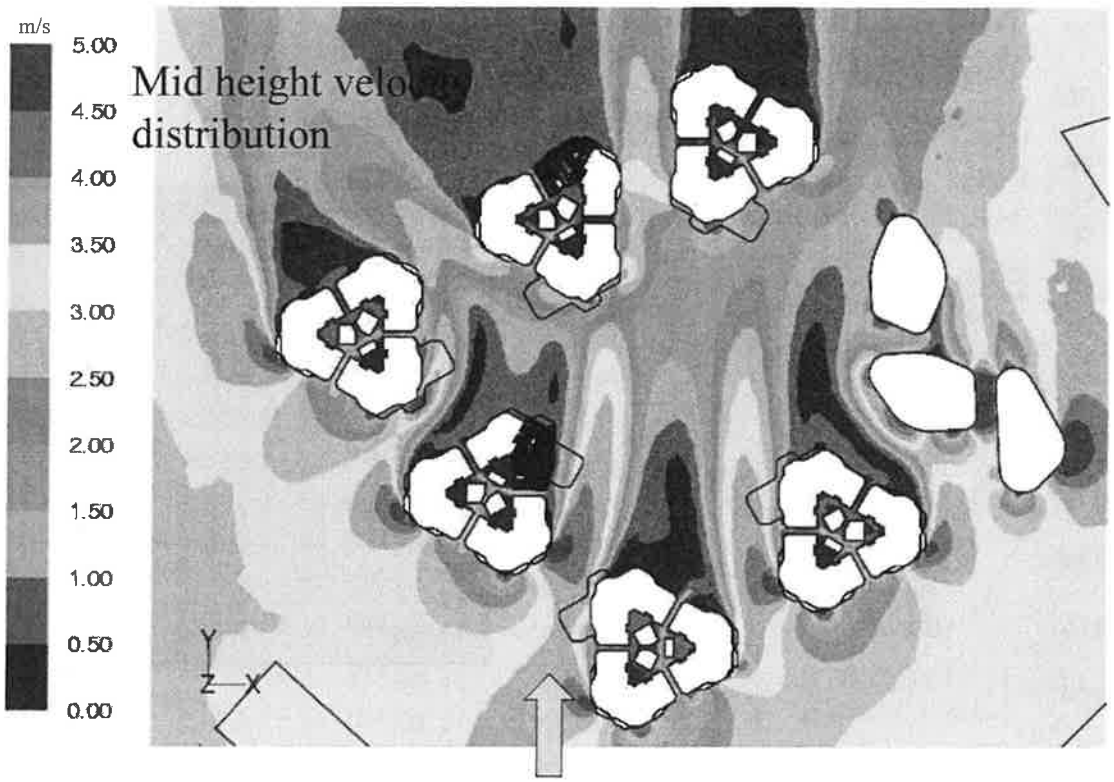


	Average velocity (m/s)
Kitchen	0.08
Living/Dining	0.12
Bedroom 1	0.17
Bedroom 2	0.12
Bedroom 3	0.21
Bedroom 4	0.32
Bedroom MBR	0.08
Average	0.14

South wind

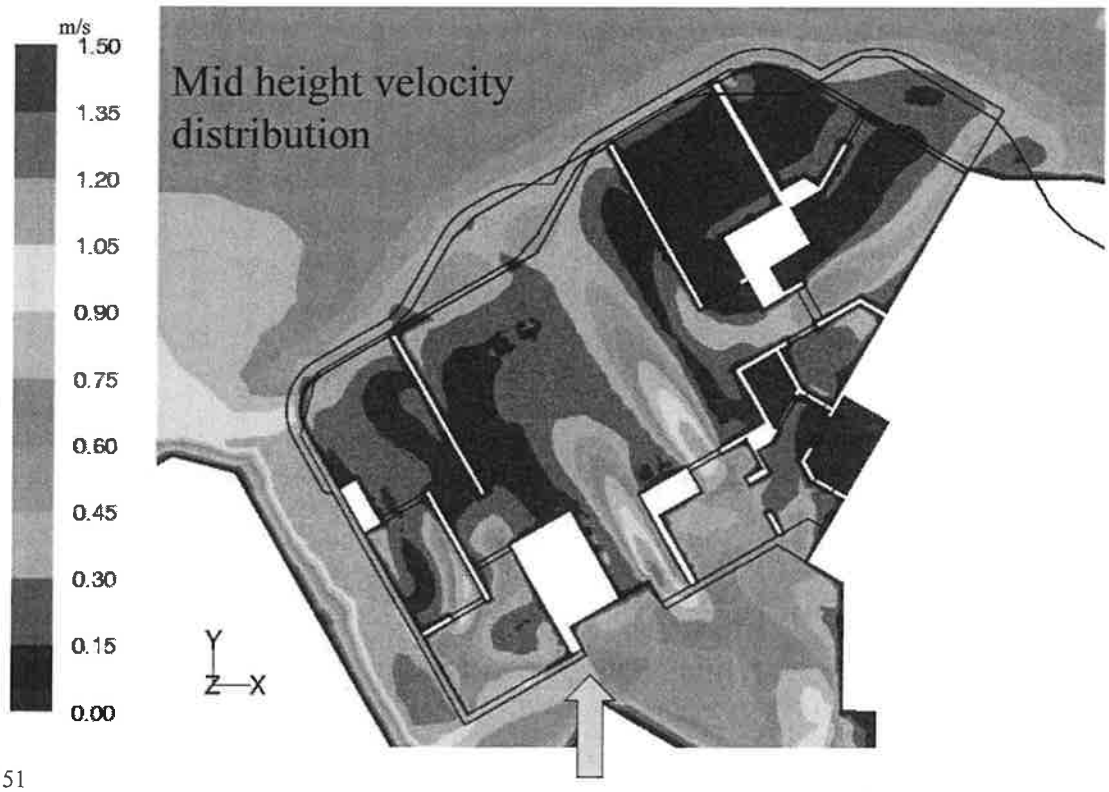


South wind



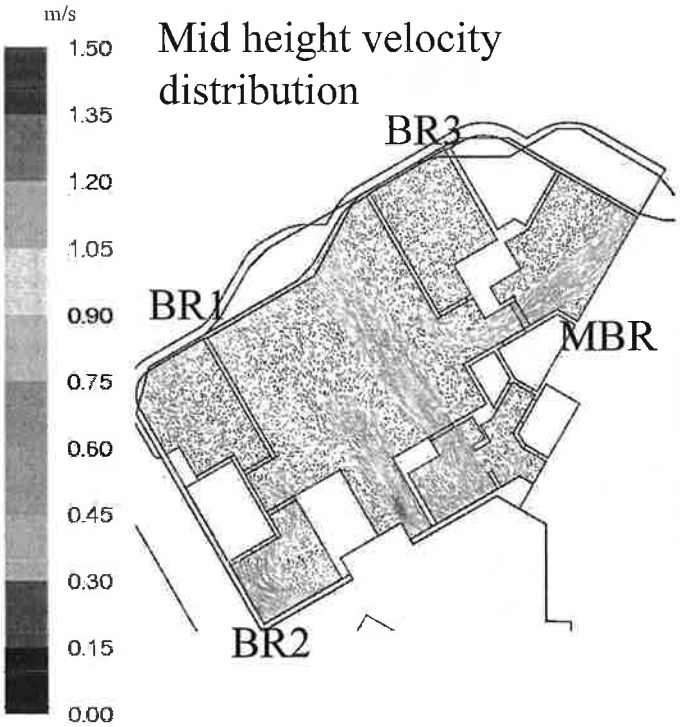
150

South wind



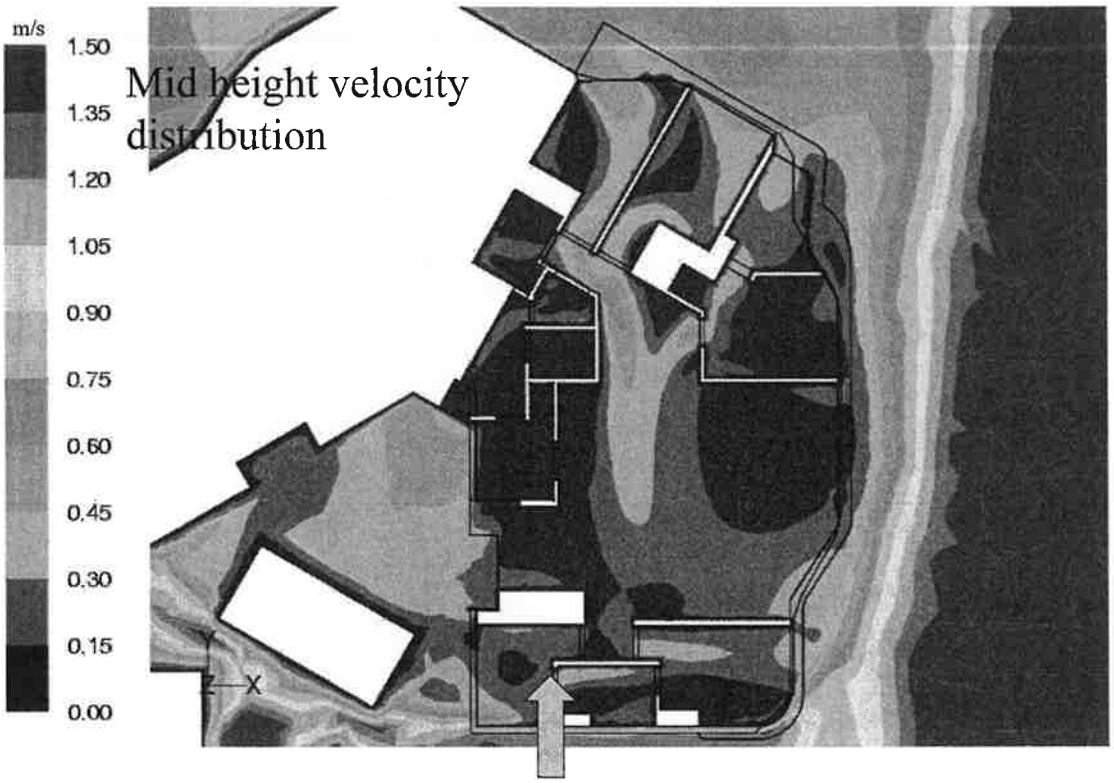
151

South wind

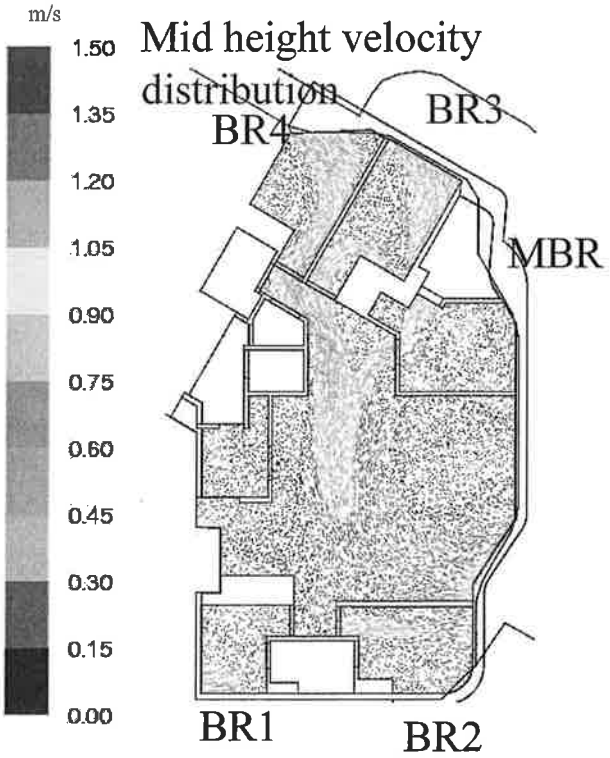


	Average velocity (m/s)
Kitchen	0.43
Living/Dining	0.32
Bedroom 1	0.19
Bedroom 2	0.52
Bedroom 3	0.09
Bedroom MBR	0.27
Average	0.30

South wind



South wind



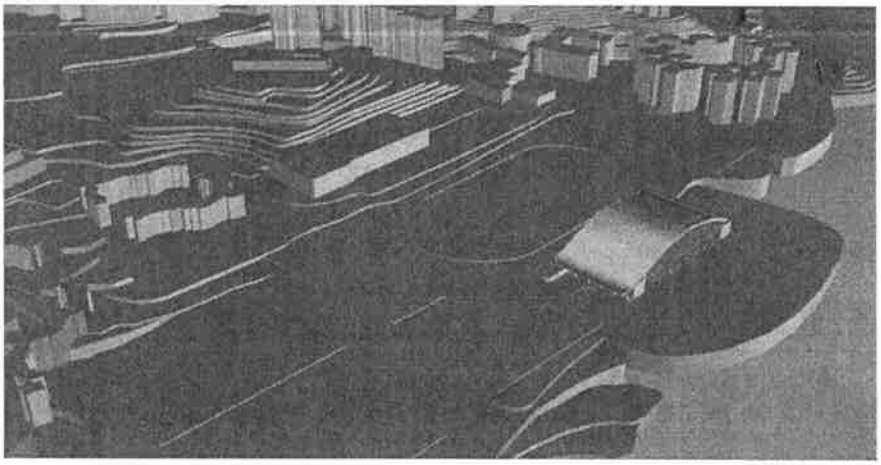
	Average velocity (m/s)
Kitchen	0.08
Living/Dining	0.19
Bedroom 1	0.25
Bedroom 2	0.19
Bedroom 3	0.28
Bedroom 4	0.29
Bedroom MBR	0.13
Average	0.20

CFD simulation for NRB building – Example B – Sports Hall

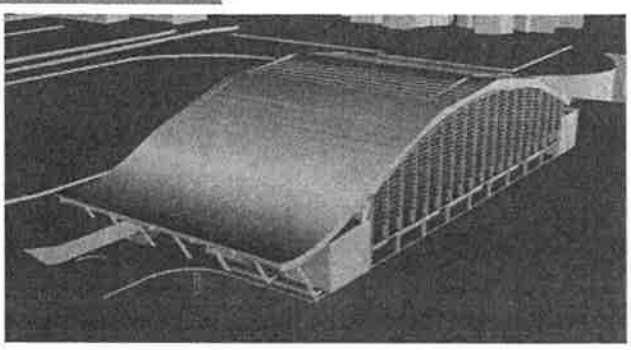
CFD NV simulation for Sport Hall

NTU sport hall, GM2015 methodology and passing criteria

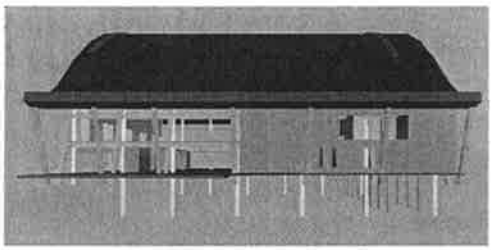
Original Model



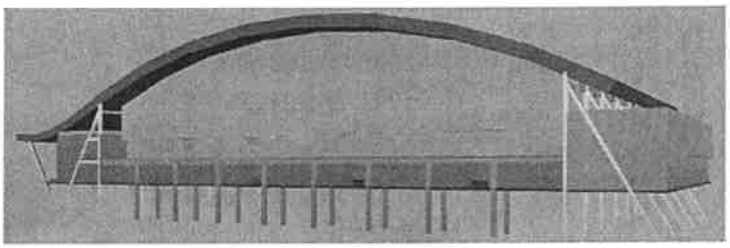
Original STL model



Simplified Geometry



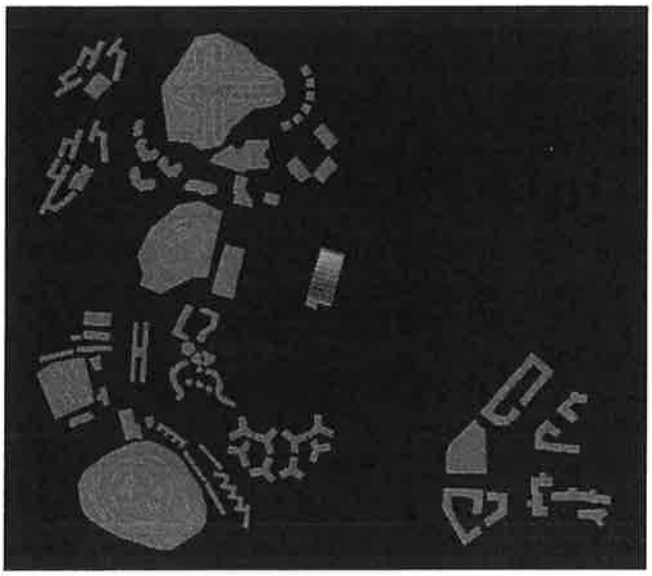
Front View

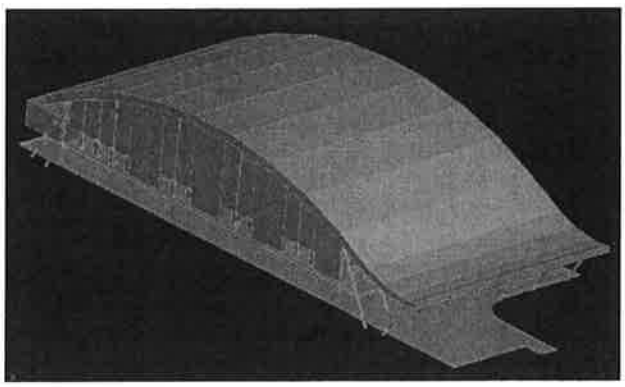


Side View

Geometry was cleaned and simplified in Rhino to ensure compatibility with Gambit

Gambit Models



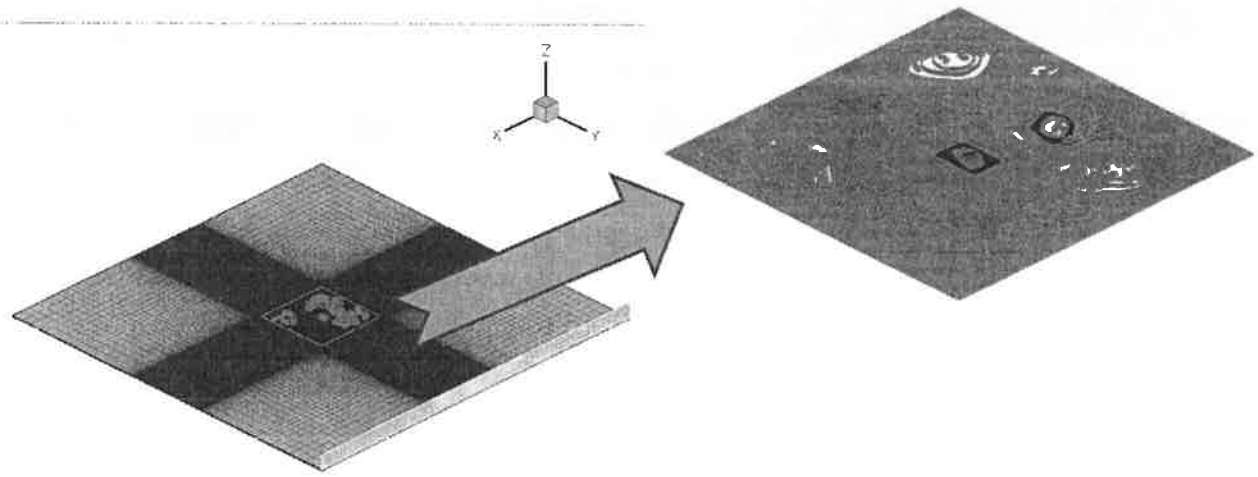


1. Surrounding buildings and actual sport hall were generated in Gambit.
2. Terrain effects were included.

Mesh

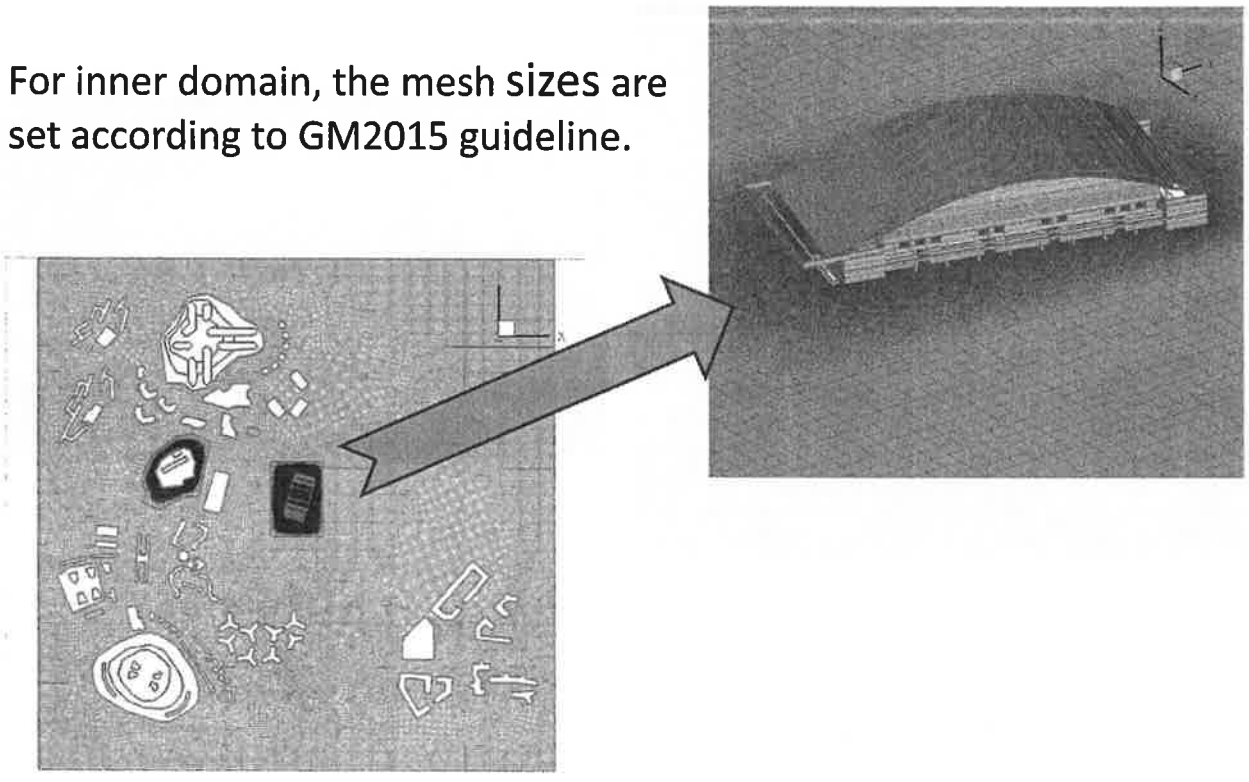
Domain size : 6.2km X 6.2km X 250m

Explicitly modelled domain : 1.2km X 1.2km X 250m

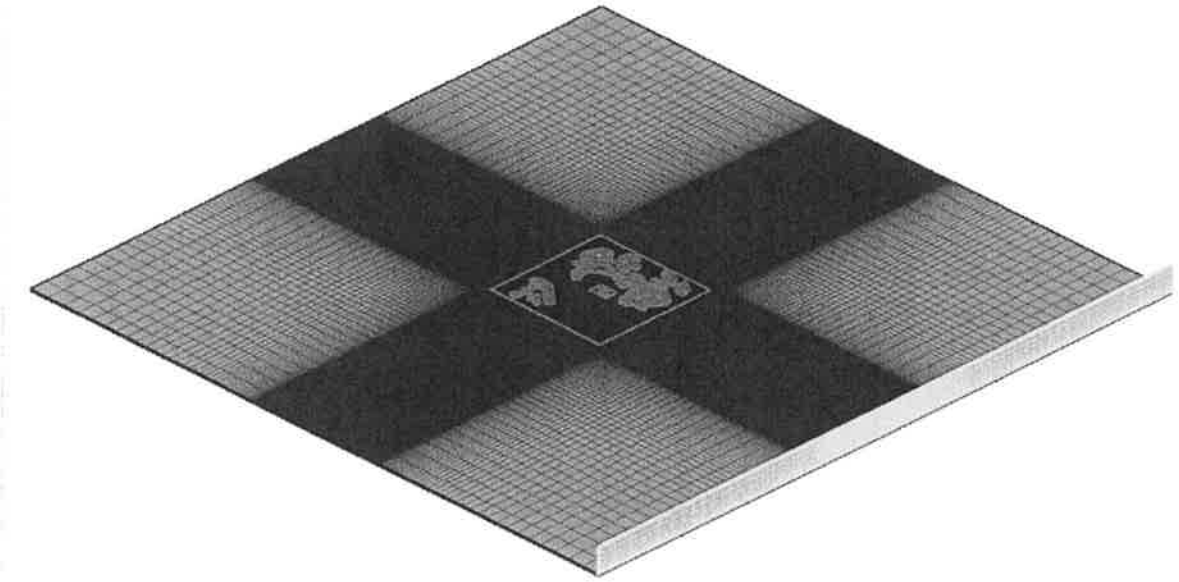


Mesh

For inner domain, the mesh sizes are set according to GM2015 guideline.



Mesh



For outer domain, the first-cell height is set as 2m, to satisfy the requirement of applying 1m aerodynamic roughness height.

Ventilation simulation

- Both north-wind and south-wind scenarios are investigated for the present ventilation simulations.

NTU Sports Hall Simulation

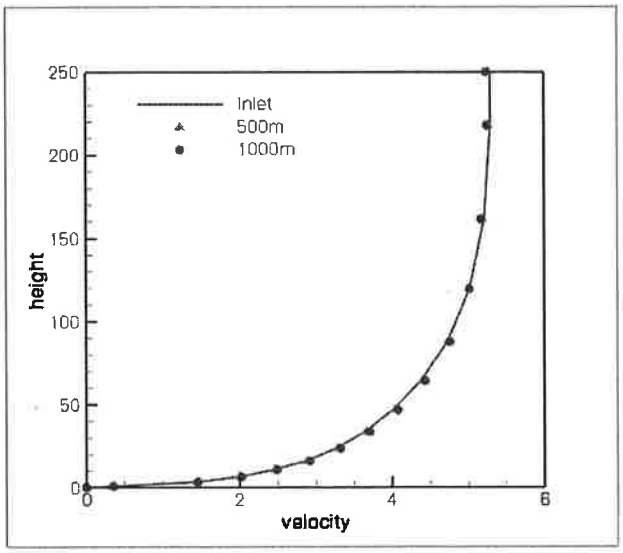
	Green Mark 2015		Green Mark 4.1	
	North Wind	South Wind	North Wind	South Wind
V_{ave} (m/s) (h=1.2m)	0.178	0.340	0.2	0.5
V_{ave} (m/s) (h=4.8m)	0.186	0.412	0.19	1.2 (?)
ACH	6.891	10.798	15.3	32.4

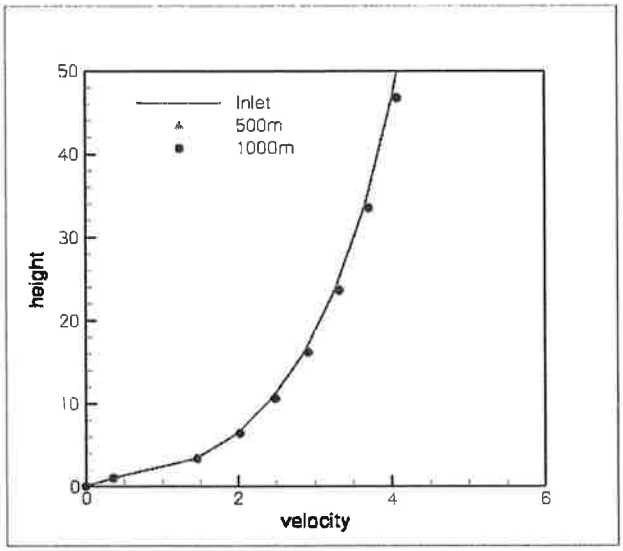
- Geometry simulated for sports hall with open doors
  - We further simplified the geometry in our simulation
- Previous simulation still had relatively high residuals

A*STAR IHPC

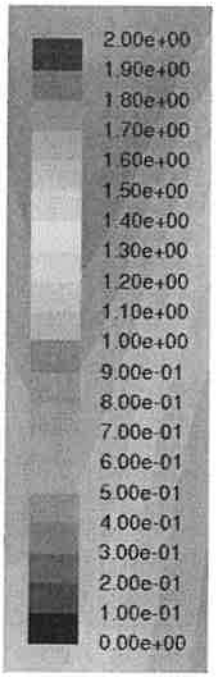
ABL : horizontal homogeneity

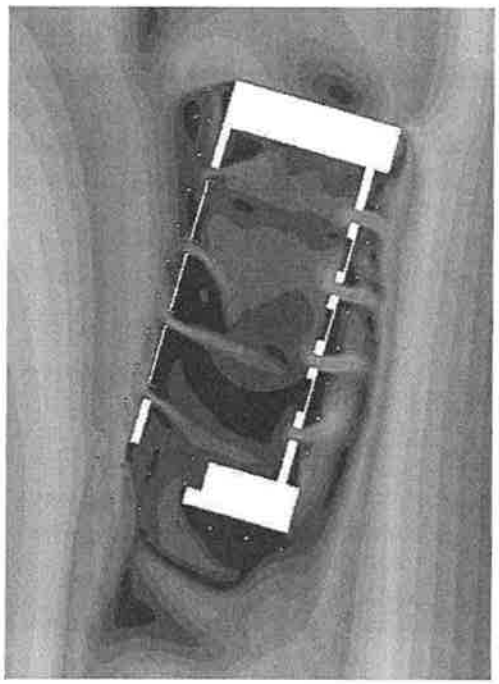
- North-wind case as a example
- Mid-plane Z-cut velocity profile at different downstream



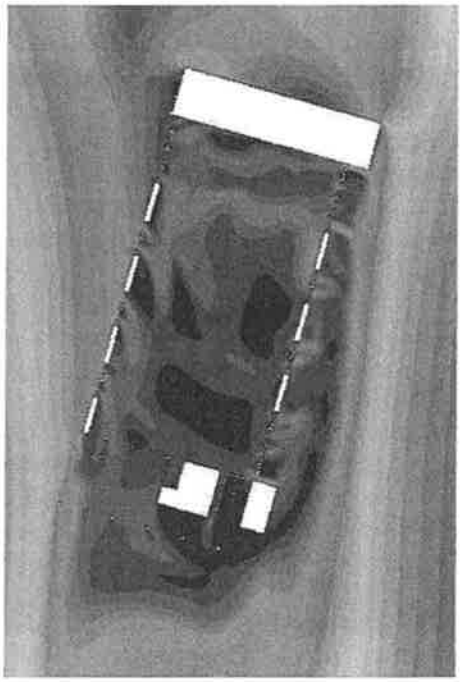


North Wind Simulation with Green Mark 2015



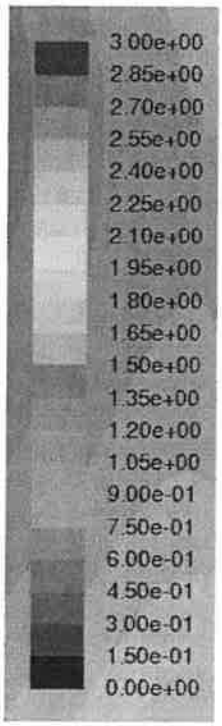


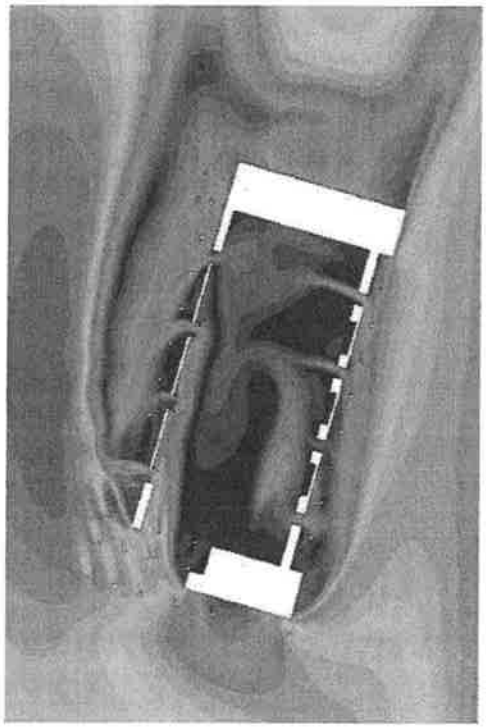
1st Floor



2nd Floor

South Wind Simulation with Green Mark 2015



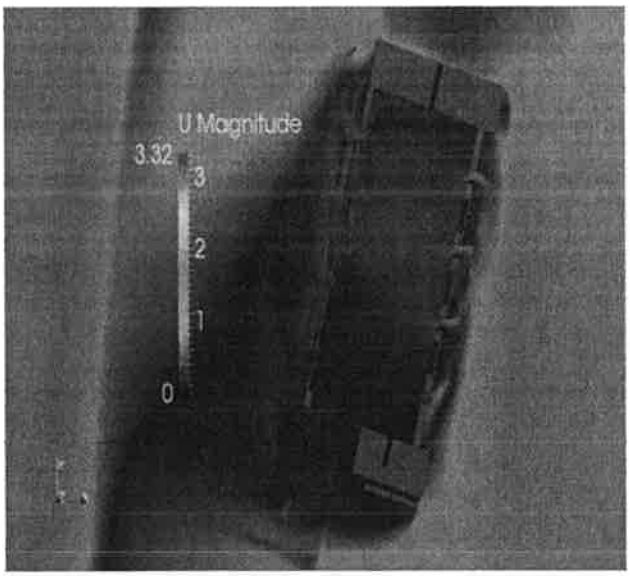


1st Floor

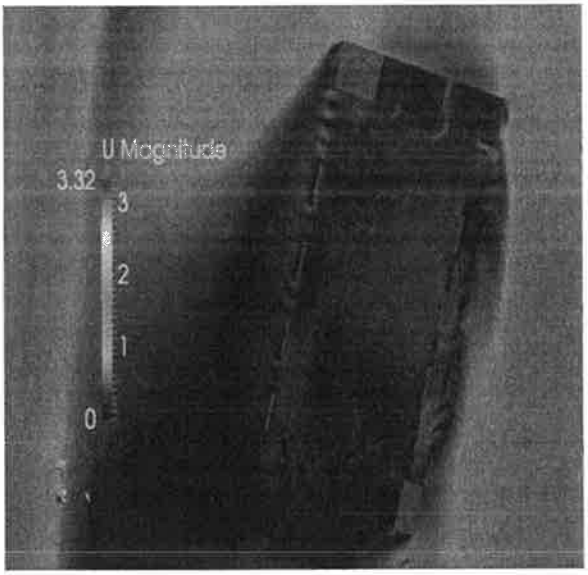


2nd Floor

North Wind with Green Mark 4.1

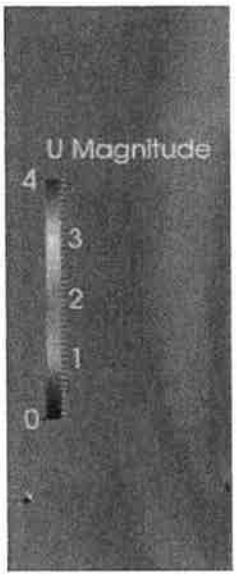


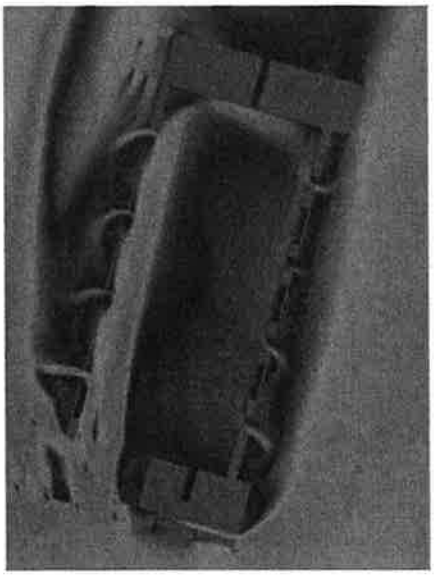
1st Floor



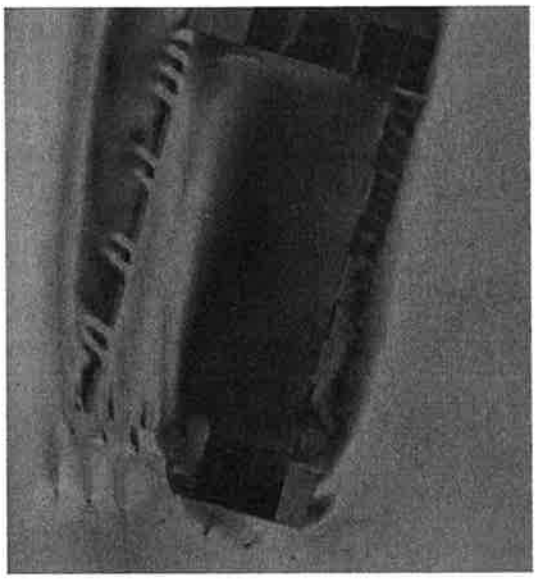
2nd Floor

South Wind with Green Mark 4.1



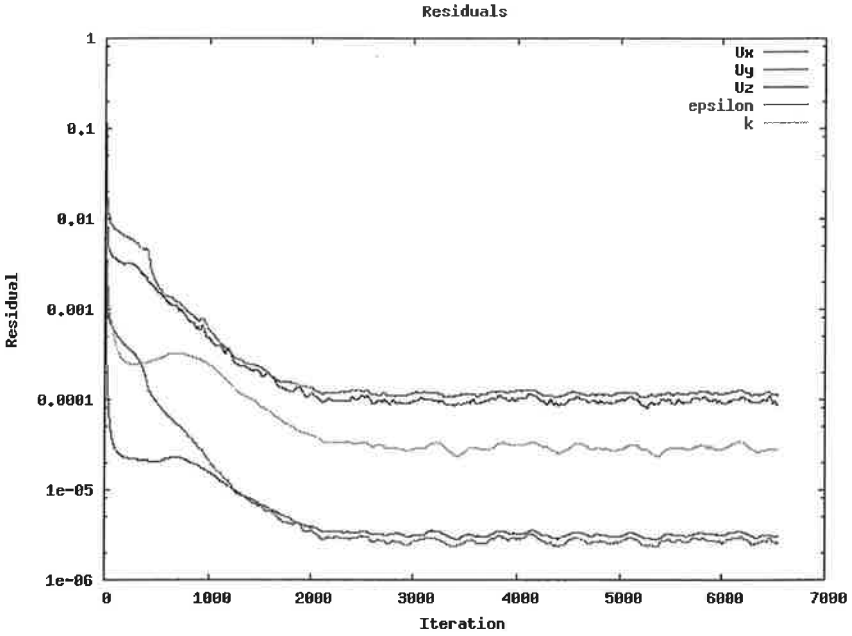


1st Floor



2nd Floor

OpenFOAM Results – South Wind (Green Mark 4.1)



Residuals for GM 4.1 OpenFOAM simulations remain high in case files

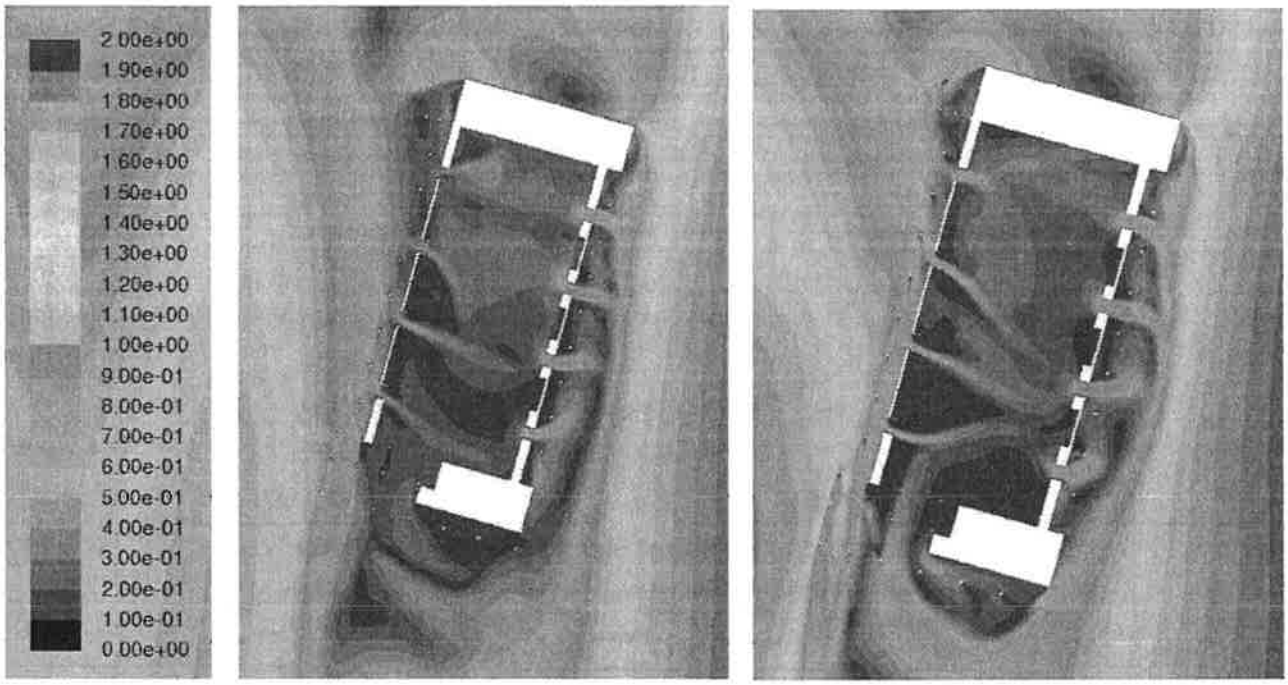
Different Meshing Results

	Green Mark 2015			
	Hex Mesh		CutCell Mesh	
	North Wind	South Wind	North Wind	South Wind
V_{ave} (m/s) (h=1.2m)	0.178	0.340	0.176	0.322
V_{ave} (m/s) (h=4.8m)	0.186	0.412	0.173	0.401
Cell Number (x 10^6)	7.2		29.9	
Residuals	~ 2×10^{-5}	~ 3×10^{-5}	~ 6×10^{-5}	~ 1×10^{-5}
External domain ground cell height	2m	2m	0.9m	0.9m

- Velocity results are relatively similar
- CutCell mesh cell count is significantly higher
- Note that for CutCell mesh, the requirement for aerodynamic roughness height (1m) is violated.

IHPC
A*STAR

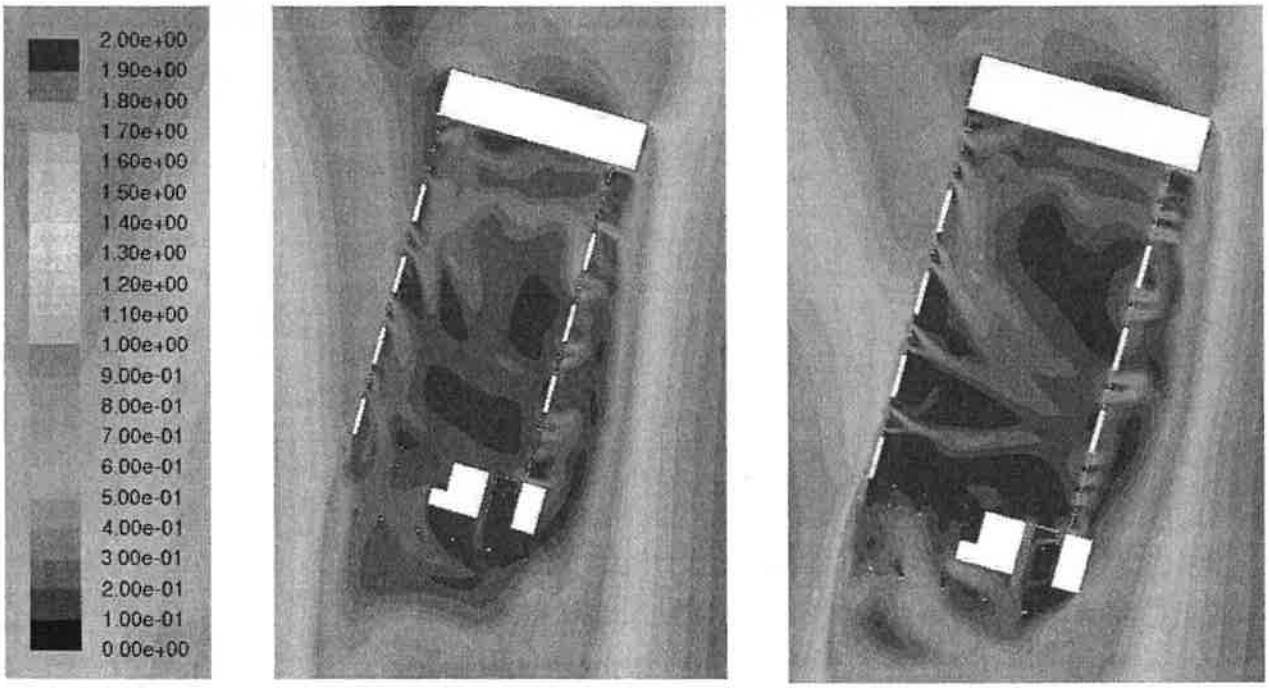
Hex vs CutCell Mesh – 1st Floor



Hex Mesh

CutCell Mesh

Hex vs CutCell Mesh – 2nd Floor



Hex Mesh

CutCell Mesh

Different Inlet k-ε Specifications

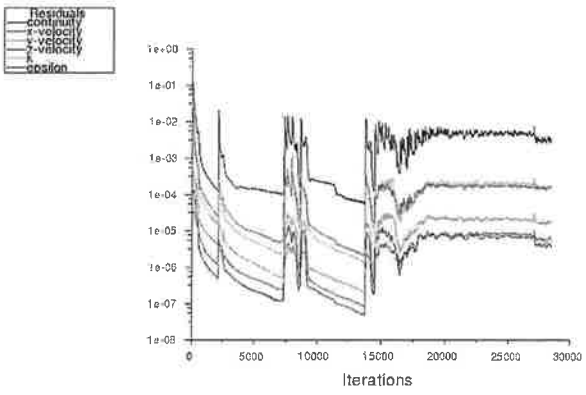
	Turbulent Viscosity Ratio at Fluent's Default Limit ($< 10^5$)			
	Inlet k-ε at Default Intensity and Length Scale		Inlet k-ε according to Blocken et al.	
	North Wind	South Wind	North Wind	South Wind
V_{ave} (m/s) (h=1.2m)	0.122	0.272	0.134	0.284
V_{ave} (m/s) (h=4.8m)	0.120	0.324	0.126	0.339
Residuals (for 2nd order)	~ 3×10^{-3}	~ 3×10^{-3}	~ 4×10^{-5}	~ 4×10^{-4}

- High residuals are observed when solving for 2nd order
- Vel. values are lower than when limit on turbulent viscosity ratio is removed

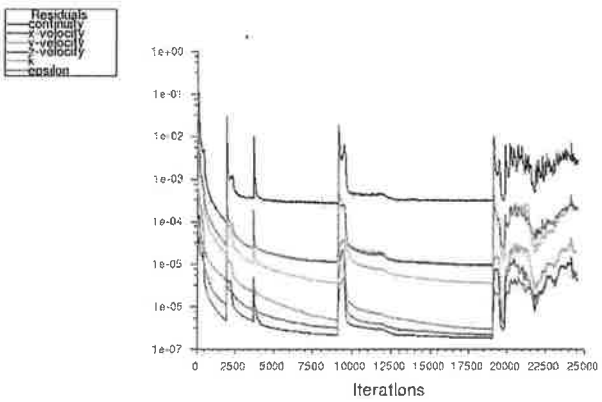
IHPC A*STAR

Residual Plots – Default k-ε

North Wind

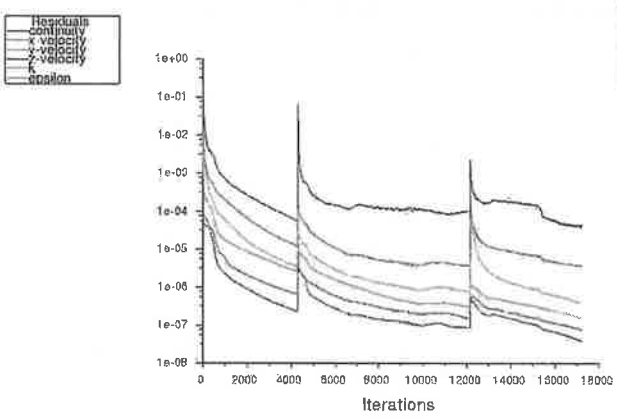


South Wind

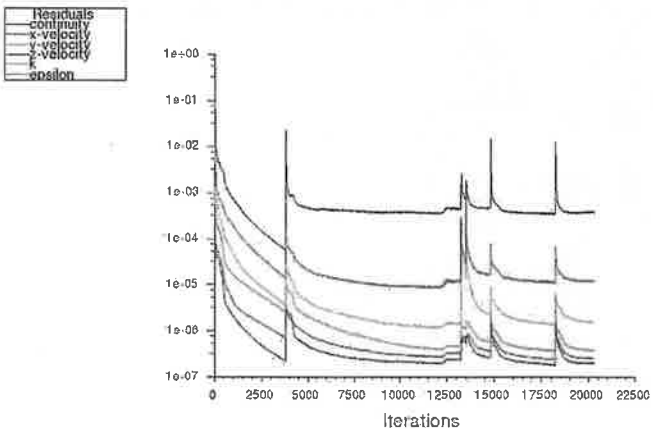


Residual Plots – UDF k-ε

North Wind



South Wind

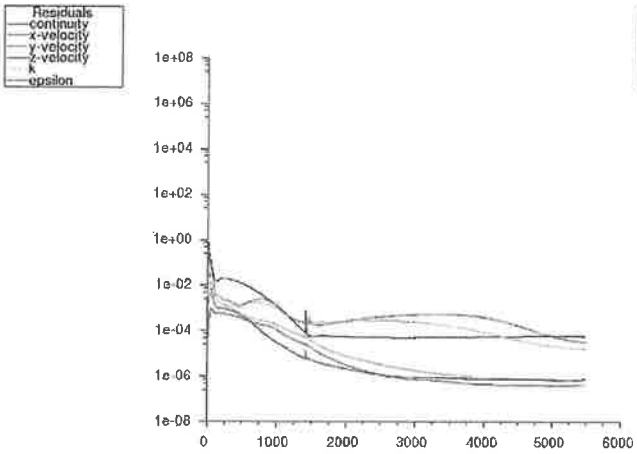


Impact of Meshing Differences

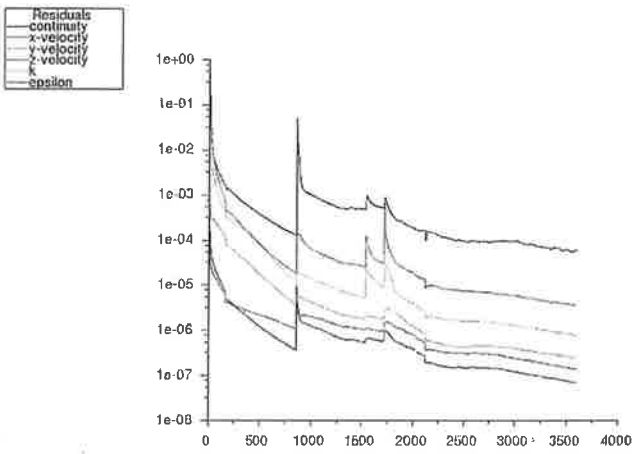
	Green Mark 2015 w/ Consistent External Domain Aerodynamics Roughness	
	Hex Mesh	**CutCell Mesh**
	North Wind	North Wind
V_{ave} (m/s) (h=1.2m)	0.234	0.194
V_{ave} (m/s) (h=4.8m)	0.241	0.207
Residuals	~ 2 x 10^{-5}	~ 6 x 10^{-5}

IHPC
A*STAR

Residual Plots – CutCell Mesh

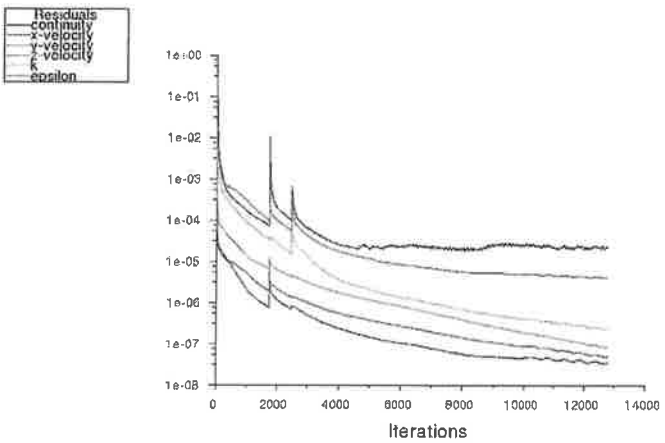


External Domain z_0 = 1m

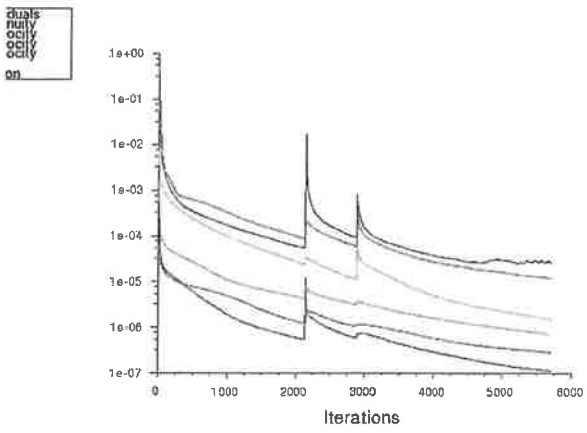


External Domain z_0 = 0.4m

Residual Plots – Hex Mesh

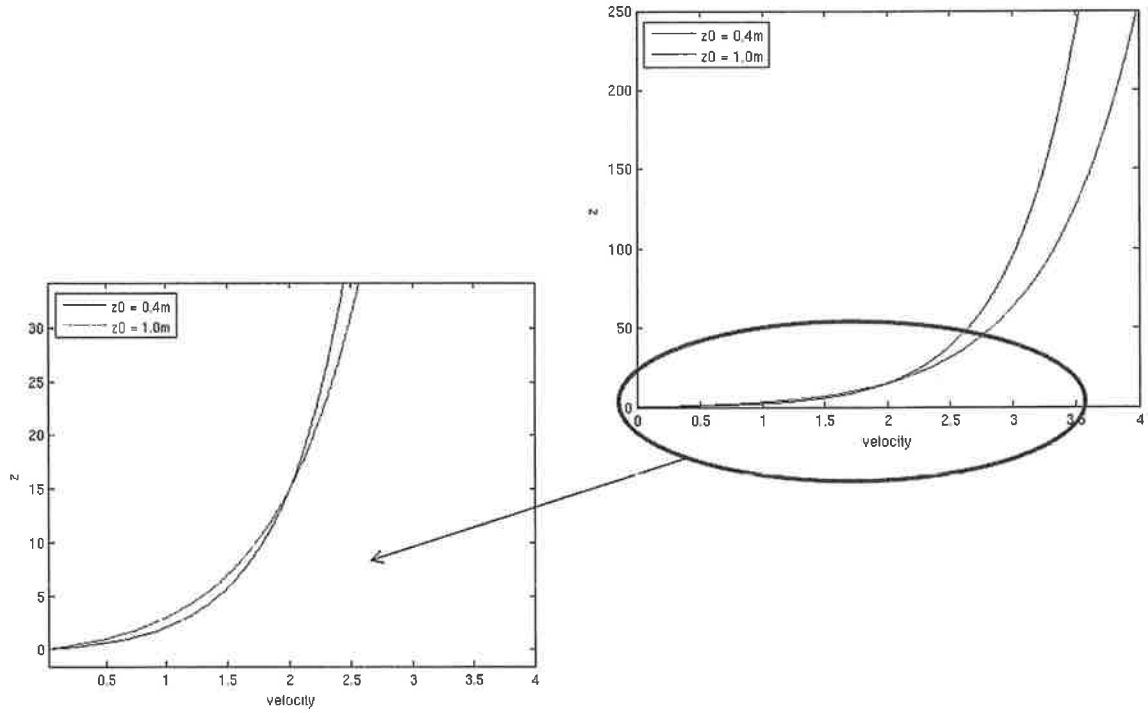


External Domain z_0 = 1m



External Domain z_0 = 0.4m

Velocity profile V.S. roughness height



Summary

- CutCell mesh yields generally similar ventilation results
- Specifying inlet profile as per Blocken et al. helps with convergence
- NV Simulations for NTU Sports Hall is performed. GM2015 criteria give lower velocity prediction, about 10 – 40% smaller

Day 2 (1:30pm – 3:00pm)

- Overview of CFD process
- Demo of CFD software

Overview of CFD Process

What is CFD?

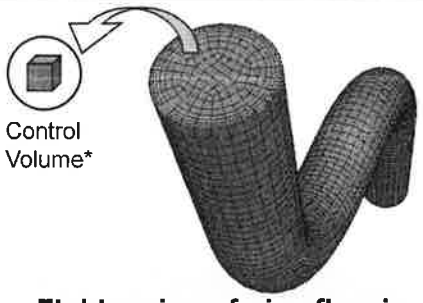
- Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations
  - Conservation of mass
  - Conservation of momentum
  - Conservation of energy
  - Conservation of species
  - Effects of body forces
  - Etc.

- The results of CFD analyses are relevant in:
  - Conceptual studies of new designs
  - Detailed product development
  - Troubleshooting
  - Redesign

- CFD analysis complements testing and experimentation by reducing total effort and cost required for experimentation and data acquisition.

How Does CFD Work?

- ANSYS CFD solvers are based on the finite volume method
  - Domain is discretized into a finite set of control volumes
  - General conservation (transport) equations for mass, momentum, energy, species, etc. are solved on this set of control volumes



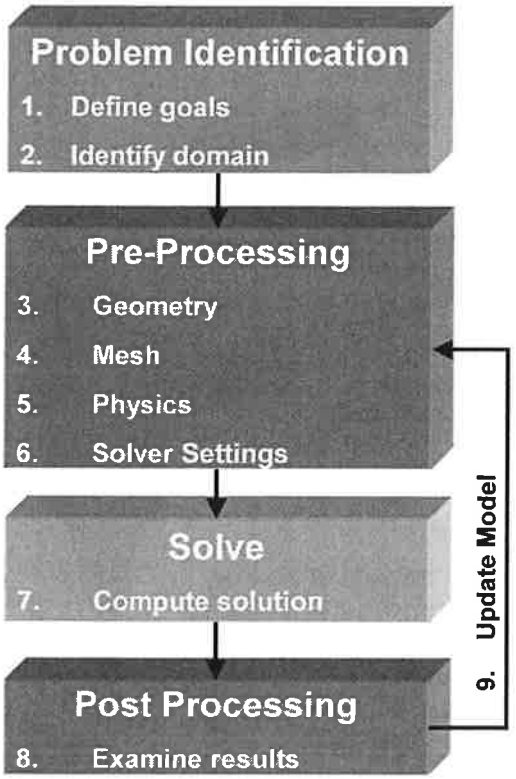
Fluid region of pipe flow is discretized into a finite set of control volumes.

$$\underbrace{\frac{\partial}{\partial t}\int_V \rho\phi\, dV}_{\text{Unsteady}} + \underbrace{\oint_A \rho\phi\mathbf{V}\cdot d\mathbf{A}}_{\text{Convection}} = \underbrace{\oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A}}_{\text{Diffusion}} + \underbrace{\int_V S_\phi\, dV}_{\text{Generation}}$$

  - Partial differential equations are discretized into a system of algebraic equations
  - All algebraic equations are then solved numerically to render the solution field

Equation	Variable
Continuity	1
X momentum	u
Y momentum	v
Z momentum	w
Energy	h

CFD Modeling Overview



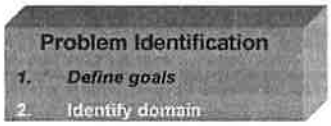
- **Problem Identification**
  1. Define your modeling goals
  2. Identify the domain you will model
- **PreProcessing and Solver Execution**
  3. Create a solid model to represent the domain
  4. Design and create the mesh (grid)
  5. Set up the physics (physical models, material properties, domain properties, boundary conditions, ...)
  6. Define solver settings (numerical schemes, convergence controls, ...)
  7. Compute and monitor the solution
- **Post-Processing**
  8. Examine the results.
  9. Consider revisions to the model.

Define Your Modeling Goals

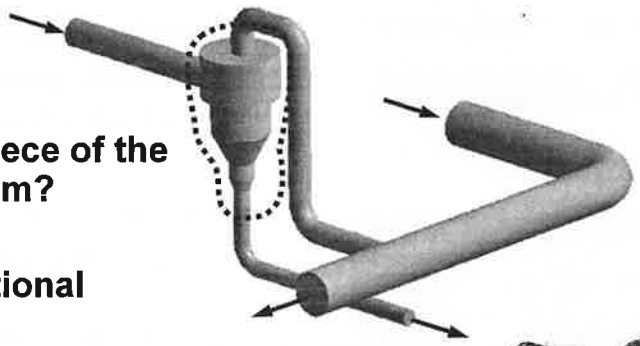


- **What results are you looking for (i.e. pressure drop, mass flow rate), and how will they be used?**
  - **What are your modeling options?**
    - **What physical models will need to be included in your analysis (i.e. turbulence, compressibility, radiation)?**
    - **What simplifying assumptions do you have to make?**
    - **What simplifying assumptions can you make (i.e. symmetry, periodicity)?**
    - **Do you require a unique modeling capability?**
      - **User-defined functions (written in C) in FLUENT or User FORTRAN functions in CFX**
- **What degree of accuracy is required?**
- **How quickly do you need the results?**
- **Is CFD an appropriate tool?**

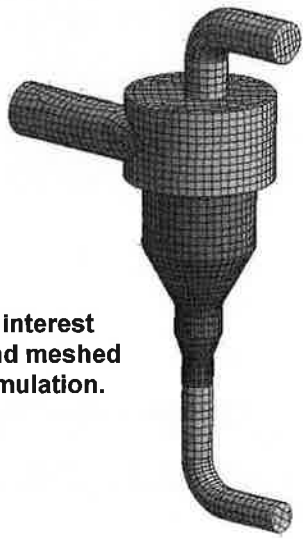
Identify the Domain You Will Model



- How will you isolate a piece of the complete physical system?
- Where will the computational domain begin and end?
  - Do you have boundary condition information at these boundaries?
  - Can the boundary condition types accommodate that information?
  - Can you extend the domain to a point where reasonable data exists?
- Can it be simplified or approximated as a 2D or axisymmetric problem?

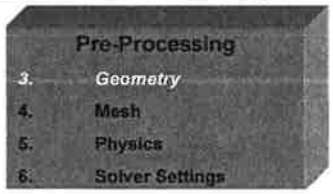


Domain of Interest as Part of a Larger System (not modeled)

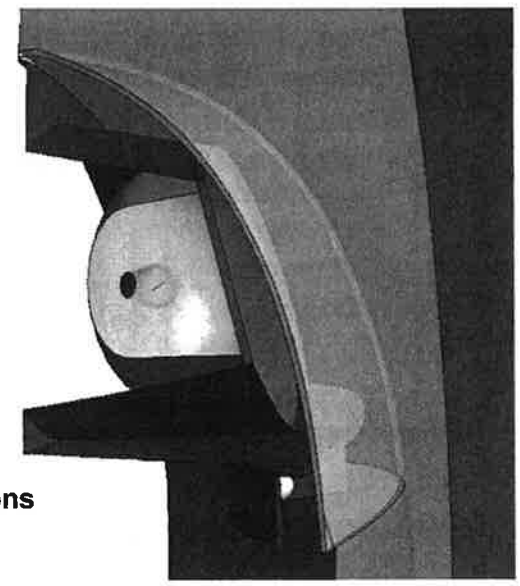


Domain of interest isolated and meshed for CFD simulation.

Create a Solid Model of the Domain

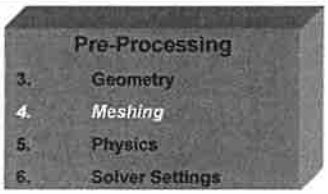


- How will you obtain a solid model of the *fluid* region?
  - Make use of existing CAD models?
    - Extract the fluid region from a solid part?
  - Create from scratch?
- Can you simplify the geometry?
  - Remove unnecessary features that would complicate meshing (fillets, bolts…)?
  - Make use of symmetry or periodicity?
    - Are both the solution and boundary conditions symmetric / periodic?
- Do you need to split the model so that boundary conditions or domains can be created?

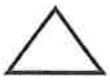


Solid model of a Headlight Assembly

Design and Create the Mesh

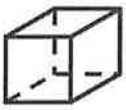


A mesh divides a geometry into many elements. These are used by the CFD solver to construct control volumes


Triangle


Quadrilateral


Tetrahedron


Hexahedron

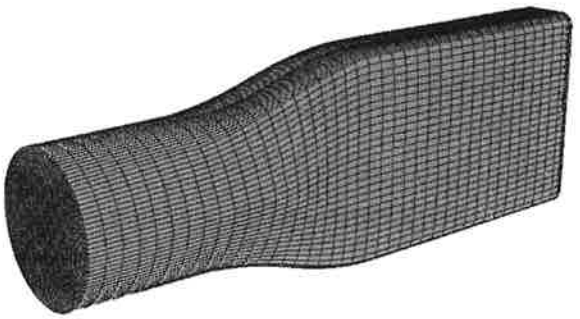

Pyramid

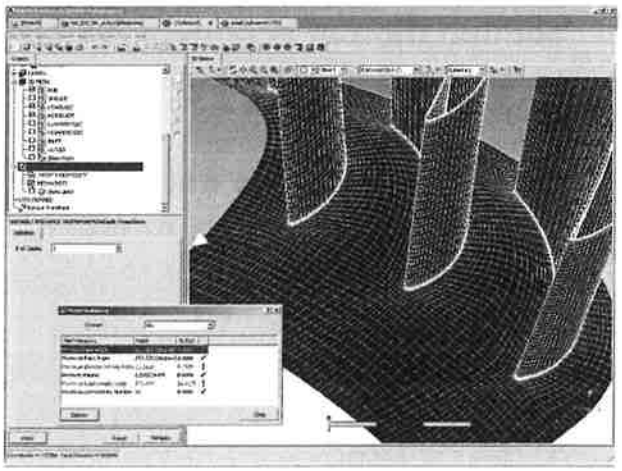

Prism/Wedge

- **What degree of mesh resolution is required in each region of the domain?**
  - **The mesh must resolve geometric features of interest and capture gradients of concern, e.g. velocity, pressure, temperature gradients**
  - **Can you predict regions of high gradients?**
  - **Will you use adaption to add resolution?**
- **What type of mesh is most appropriate?**
  - **How complex is the geometry?**
  - **Can you use a quad/hex mesh or is a tri/tet or hybrid mesh suitable?**
  - **Are non-conformal interfaces needed?**
- **Do you have sufficient computer resources?**
  - **How many cells/nodes are required?**
  - **How many physical models will be used?**

Tri/Tet vs. Quad/Hex Meshes

- **For flow-aligned geometries, quad/hex meshes can provide higher-quality solutions with fewer cells/nodes than a comparable tri/tet mesh**
  - **Quad/Hex meshes show reduced numerical diffusion when the mesh is aligned with the flow.**
  - **It does require more effort to generate a quad/hex mesh**

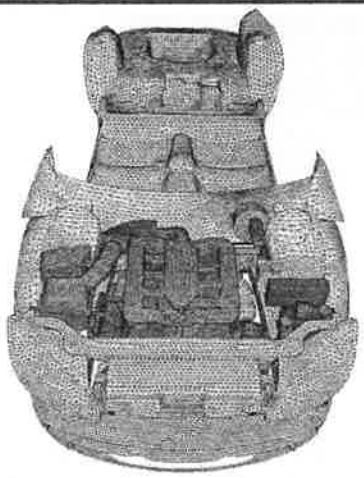




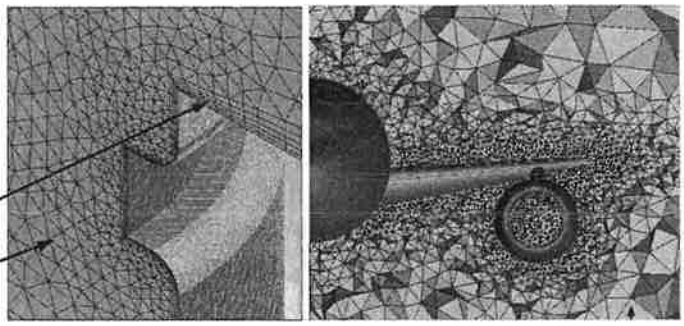
- **Meshing tools designed for a specific application can streamline the process of creating a quad/hex mesh for some geometries.**

Tri/Tet vs. Quad/Hex Meshes

- **For complex geometries, quad/hex meshes show no numerical advantage, and you can save meshing effort by using a tri/tet mesh or hybrid mesh**
  - **Quick to generate**
  - **Flow is generally not aligned with the mesh**



- **Hybrid meshes typically combine tri/tet elements with other elements in selected regions**
  - **For example, use wedge/ prism elements to resolve boundary layers.**
  - **More efficient and accurate than tri/tet alone.**



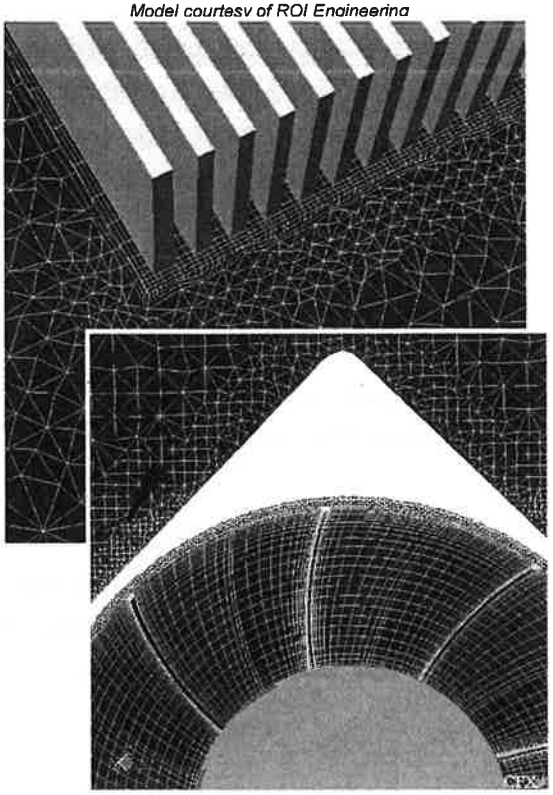
Multizone (or Hybrid) Meshes

- **A multizone or hybrid mesh uses different meshing methods in different regions. For example,**
  - **Hex mesh for fan and heat sink**
  - **Tet/prism mesh elsewhere**

- **Multizone meshes yield a good combination of accuracy, efficient calculation time and meshing effort.**

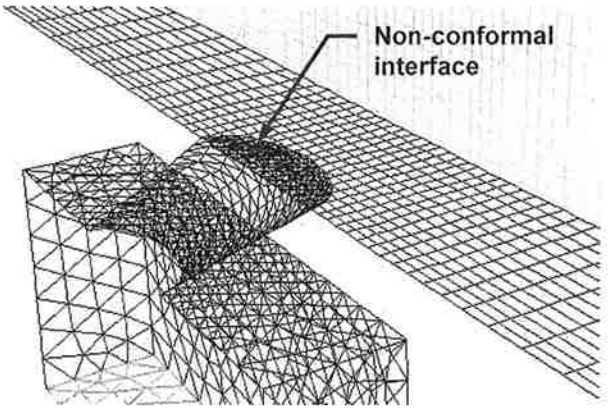
- **When the nodes do not match across the regions, a non-conformal interface can be used.**

Model courtesy of ROI Engineering

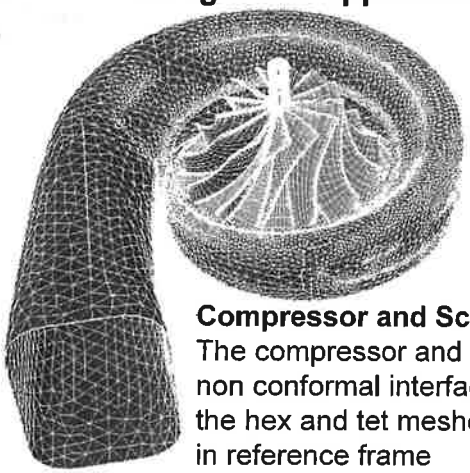


Non-Conformal Meshes

- **Non conformal meshes are useful for meshing complex geometries**
  - **Mesh each part then join together**
- **Non conformal interfaces are also used in other situations**
  - **Change in reference frames**
  - **Moving mesh applications**

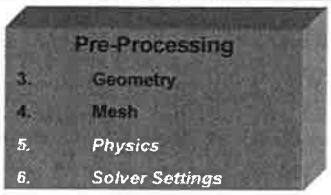


3D Film Cooling
Coolant is injected into a duct from a plenum. The plenum is meshed with tetrahedral cells while the duct is meshed with hexahedral cells



Compressor and Scroll
The compressor and scroll are joined through a non conformal interface. This serves to connect the hex and tet meshes and also allows a change in reference frame

Set Up the Physics and Solver Settings



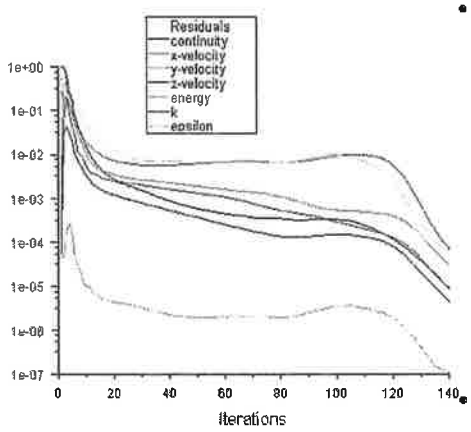
For complex problems solving a simplified or 2D problem will provide valuable experience with the models and solver settings for your problem in a short amount of time.

- **For a given problem, you will need to:**
  - **Define material properties**
    - **Fluid**
    - **Solid**
    - **Mixture**
  - **Select appropriate physical models**
    - **Turbulence, combustion, multiphase, etc.**
  - **Prescribe operating conditions**
  - **Prescribe boundary conditions at all boundary zones**
  - **Provide initial values or a previous solution**
  - **Set up solver controls**
  - **Set up convergence monitors**

Mass flow rate

Compute the Solution



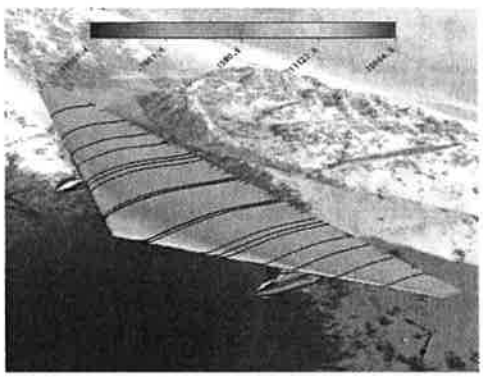


A converged and mesh-independent solution on a well-posed problem will provide useful engineering results!

- **The discretized conservation equations are solved iteratively until convergence.**
- **Convergence is reached when:**
  - **Changes in solution variables from one iteration to the next are negligible.**
    - **Residuals provide a mechanism to help monitor this trend.**
  - **Overall property conservation is achieved**
    - **Imbalances measure global conservation**
  - **Quantities of interest (e.g. drag, pressure drop) have reach steady values.**
    - **Monitor points track quantities of interest.**
- **The accuracy of a *converged* solution is dependent upon:**
  - **Appropriateness and accuracy of physical models.**
  - **Mesh resolution and independence**
  - **Numerical errors**

Examine the Results

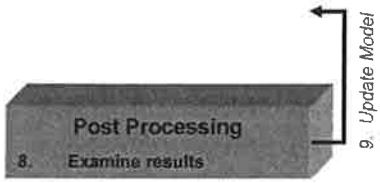




Examine results to ensure property conservation and correct physical behavior. High residuals may be caused by just a few poor quality cells.

- **Examine the results to review solution and extract useful data**
  - **Visualization Tools can be used to answer such questions as:**
    - **What is the overall flow pattern?**
    - **Is there separation?**
    - **Where do shocks, shear layers, etc. form?**
    - **Are key flow features being resolved?**
  - **Numerical Reporting Tools can be used to calculate quantitative results:**
    - **Forces and Moments**
    - **Average heat transfer coefficients**
    - **Surface and Volume integrated quantities**
    - **Flux Balances**

Consider Revisions to the Model



- **Are the physical models appropriate?**
  - Is the flow turbulent?
  - Is the flow unsteady?
  - Are there compressibility effects?
  - Are there 3D effects?
- **Are the boundary conditions correct?**
  - Is the computational domain large enough?
  - Are boundary conditions appropriate?
  - Are boundary values reasonable?
- **Is the mesh adequate?**
  - Can the mesh be refined to improve results?
  - Does the solution change significantly with a refined mesh, or is the solution mesh independent?
  - Does the mesh resolution of the geometry need to be improved?

CFD Demo

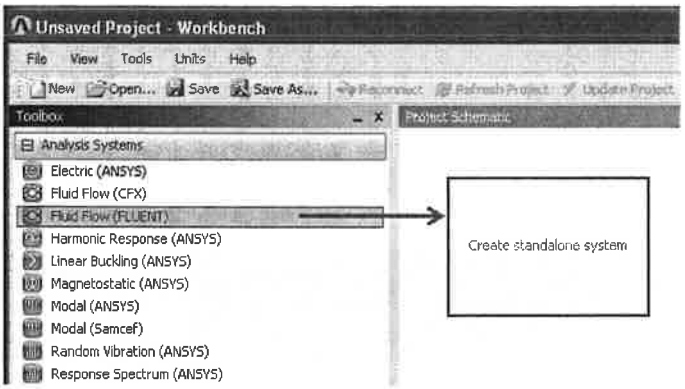
ANSYS Workbench & FLUENT

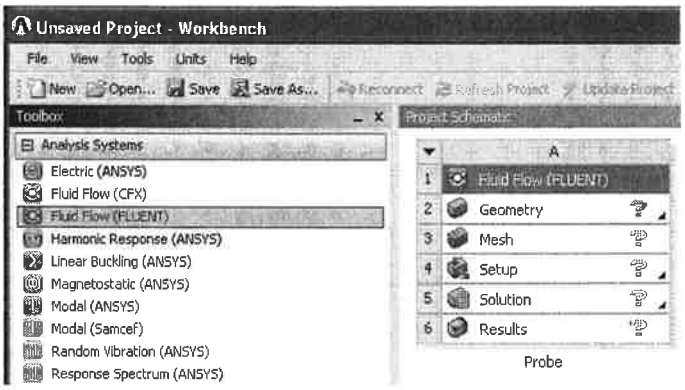
- Launching CFD software
- Understanding User Interface
- Reading, Writing and Managing Files

1. Design modeler — Geometry creation
2. Ansys meshing — mesh generation
3. Ansys Fluent — solver
   — post processing

Workflow

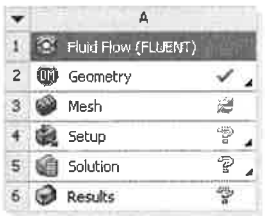
- Start ANSYS Workbench
- Drag the **Fluid Flow (FLUENT)** system from Analysis Systems group in the Toolbox onto preview drop target shown in the **Project Schematic**.





Import the Geometry

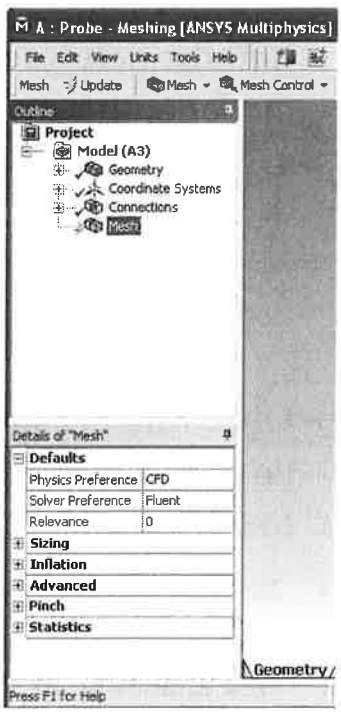
- Right-click on Geometry cell A2 and select Import Geometry
- Import the geometry file (CAD model or DesignModeler .agdb file)
- You can also link the FLUENT simulation to an existing DesignModeler session.



Probe

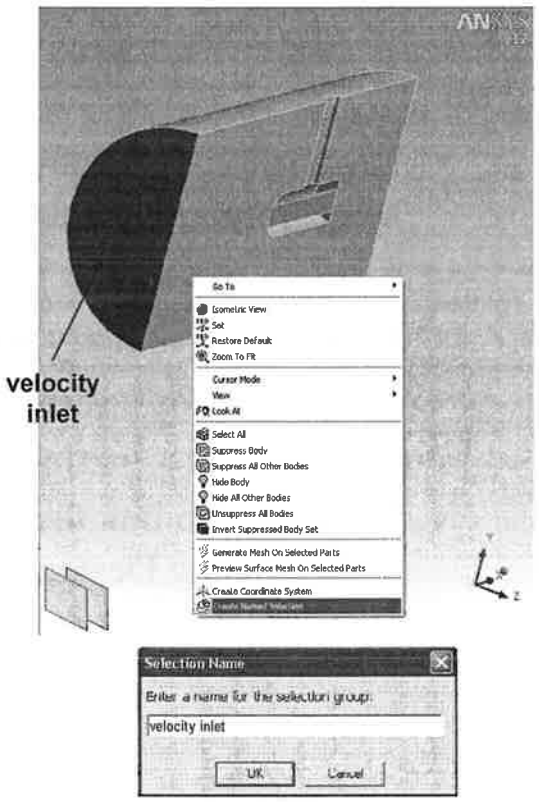
Generate a Mesh

- Right-click on Mesh cell and select Edit.
  - Meshing opens and loads geometry
- Select Mesh under Model in Outline
  - Note that Preferences are automatically set for FLUENT, because Meshing was opened from a FLUENT system.



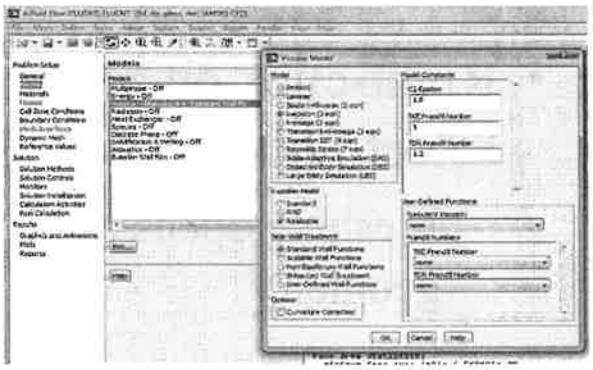
Define Boundary and Cell Zones

- Create boundary zones using Named selections.
  - Select the surface which will represent the boundary you wish to set.
  - Right-click the selection and select Create Named Selection.
  - Name the selection and click OK.
- You will also need to define the regions of the flow containing fluid and solid (if any).
  - Solids are required for conjugate heat transfer calculations only.
  - More details will be presented later.

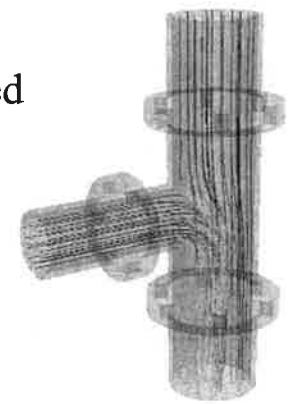


Set Up and Run FLUENT

- Edit the Setup cell to set up the model options
  - Boundary conditions
  - Solver settings
  - Solution
  - Post processing



- Once run, the solution can then be either post processed in FLUENT or data exported to CFD-Post for post processing
  - Contour and vector plots
  - Profile plots
  - Calculation of forces and moments
  - Animation of unsteady flow results



Day 2 (3:30pm – 5:30pm)

- Tutorial for Practice 1 – Fluid Flow and Heat Transfer in a Mixing Elbow (step by step guideline about using ANSYS Fluent in ANSYS Workbench)

Day 3 (9:00am – 10:30am)

- Assignment briefing
- Introduction to CFD
  - Mathematical Background
  - Turbulence Model
  - Wind Effect on Building Structure
  - Verification and Validation in CFD
- Creating 3D geometry model

Assignment Briefing

Introduction to CFD

Mathematical Background

- General 3-dimentional advection-diffusion equation

$$\frac{\partial(\rho\Phi)}{\partial t}+div\left(\rho\vec{V}\Phi\right)=div\left(\Gamma_\Phi \overrightarrow{grad}\Phi\right)+S_\Phi$$

The dependent variables (Φ), effective diffusion coefficients (Γ_Φ) and source terms (S_Φ) in the transport equations

Equation	Φ	Γ_Φ	S_Φ
Continuity	1	0	0
X momentum	u	μ	$-\frac{\partial P}{\partial x}$
Y momentum	v	μ	$-\frac{\partial P}{\partial y}-\rho g$
Z momentum	w	μ	$-\frac{\partial P}{\partial z}$
Energy	c_pT	k	Q
Species	c/ρ	d	Q_m

velocity of fluid. (vector quantity)

Pressure (differential)

Mathematical Background

- Reynolds decomposition to the 2D, Newtonian, incompressible instantaneous flow equations

1. Continuity

$$\frac{\partial\overline{u}}{\partial x}+\frac{\partial\overline{v}}{\partial y}=0$$

2. X momentum

$$\frac{\partial\overline{u}}{\partial t}+\overline{u}\frac{\partial\overline{u}}{\partial x}+\overline{v}\frac{\partial\overline{u}}{\partial y}=-\frac{1}{\rho}\frac{\partial\overline{p}}{\partial x}+\frac{\partial}{\partial x}\left(\nu\frac{\partial\overline{u}}{\partial x}-\overline{u'u'}\right)+\frac{\partial}{\partial y}\left(\nu\frac{\partial\overline{u}}{\partial y}-\overline{u'v'}\right)$$

3. Y momentum

$$\frac{\partial\overline{v}}{\partial t}+\overline{u}\frac{\partial\overline{v}}{\partial x}+\overline{v}\frac{\partial\overline{v}}{\partial y}=-\frac{1}{\rho}\frac{\partial\overline{p}}{\partial y}+\frac{\partial}{\partial x}\left(\nu\frac{\partial\overline{v}}{\partial x}-\overline{u'v'}\right)+\frac{\partial}{\partial y}\left(\nu\frac{\partial\overline{v}}{\partial y}-\overline{v'v'}\right)+g\beta_T\left(\overline{T}-T_o\right)$$

4. Energy

$$\frac{\partial\overline{T}}{\partial t}+\overline{u}\frac{\partial\overline{T}}{\partial x}+\overline{v}\frac{\partial\overline{T}}{\partial y}=\frac{k}{\rho c_p}\left(\frac{\partial^2\overline{T}}{\partial x^2}+\frac{\partial^2\overline{T}}{\partial y^2}\right)+\frac{\partial}{\partial x}\left(-\overline{u'T'}\right)+\frac{\partial}{\partial y}\left(-\overline{v'T'}\right)$$

Solve for primary variables, $\overline{u},\overline{v},\overline{p},\overline{T}$ turbulent stresses $\overline{u'u'},\overline{u'v'},\overline{v'v'}$ and turbulent heat transfers $\overline{u'T'},\overline{v'T'}$

Mathematical Background

- Analogy between laminar and turbulent stresses

$$\tau_{ij} = \nu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad -\overline{u_i' u_j'} = \nu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$$

- Analogy between laminar and turbulent heat fluxes

$$\varphi_j = k\left(\frac{\partial T}{\partial x_j}\right) \qquad -\overline{u_j' T'} = k_t\left(\frac{\partial \bar{T}}{\partial x_j}\right)$$

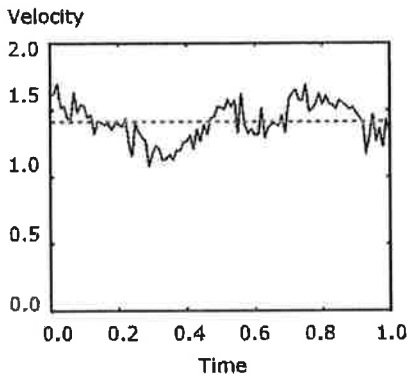
- Laminar kinematic viscosity (ν) and thermal conductivity (k) are the intrinsic properties of fluid, while turbulent viscosity (ν_t) and turbulent thermal conductivity (k_t) and are the local properties of flow
- Both analogies can reduce number of unknowns, but ν_t and k_t must be calculated everywhere in the flow

Turbulent → mixing, eddy, rate of transfer

Turbulence Models

Laminar flow/turbulent flow
2D steady state

$$\rho u\frac{\partial c}{\partial x} + \rho v\frac{\partial c}{\partial y} = \Gamma_c\frac{\partial^2 c}{\partial y^2} + S_c$$



A discussion of different turbulence models as e.g. the *k-ε* model, the *k-ω* model, the *SST* model and the Reynolds Stress model

The Large Eddy Simulation is also discussed

Direct numerical solution.

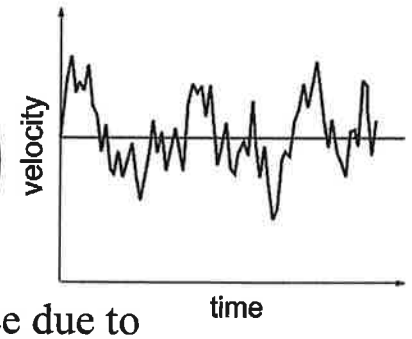
Turbulence Models

- Laminar flow – momentum conservation in laminar boundary layer

$$\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}=-\frac{dp}{dx}+\mu\frac{\partial^2 u}{\partial y^2}$$

- Turbulent flow – momentum conservation in turbulent boundary layer

$$\rho\bar{u}\frac{\partial \bar{u}}{\partial x}+\rho\bar{v}\frac{\partial \bar{u}}{\partial y}=-\frac{d\bar{p}}{dx}+\mu\frac{\partial^2 \bar{u}}{\partial y^2}-\rho\frac{\partial \overline{u'v'}}{\partial y}$$



Additional force due to turbulent fluctuations

In boundary layer, the turbulent shear stress, $\rho\overline{u'v'}$ and the velocity gradient $\partial\bar{u}/\partial y$ always have opposite signs

Turbulence Models

1. Turbulent viscosity models
   - Turbulent shear stress can be modeled as $\rho\overline{u_i'u_j'}=-\mu_t\frac{\partial \bar{u}}{\partial y}$
   - Momentum equation in boundary layer can be written as

$$\rho\bar{u}\frac{\partial \bar{u}}{\partial x}+\rho\bar{v}\frac{\partial \bar{u}}{\partial y}=-\frac{d\bar{p}}{dx}+\frac{\partial}{\partial y}\left[(\mu+\mu_t)\frac{\partial \bar{u}}{\partial y}\right]$$

   - κ-ε is the most popular turbulence model
   - Two additional transport equations are solved for the turbulence kinetic energy κ (m^2/s^2) and turbulence dissipation rate ε (m^2/s^3)
   - Eddy viscosity is given as $\mu_t=c_\mu\rho\frac{\kappa^2}{\varepsilon}$
   - κ-ω turbulence model, $\mu_t=\rho\frac{\kappa}{\omega}$

Turbulence Models

2. Shear Stress Transport models
   - Combination of κ-ω model (in the inner boundary layer) and κ-ε model (in the outer region of and outside boundary layer)
   - κ-ε model has weakness of over-predicting shear stress in adverse pressure gradient flows
   - Menter (1994) propose SST turbulence model as it predicts smaller shear stress for adverse pressure gradient flows

$$-\overline{u'v'} = k \frac{1}{\max\left(0.31\omega, \partial\overline{u}/\partial y\right)}$$

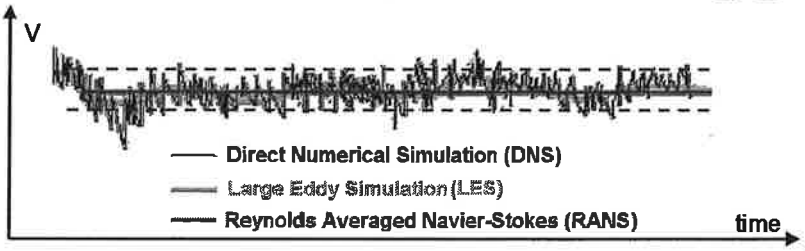
Turbulence Models

3. Reynolds stress models
   - Transport equations are established for the six Reynolds shear stress components (in 3D), $\overline{u'u'}, \overline{v'v'}, \overline{w'w'}, \overline{u'v'}, \overline{u'w'}, \overline{v'w'}$
   - Computationally more expensive than κ-ε model
   - More details on RSM can be found in Hanjalic (1999)
- For eddy viscosity, SST & RSM, they are considered as RANS (Reynolds Averaged Navier Stokes) approach, in which the dependent variables (e.g. *u, v, p, T*) are decomposed into time-averaged and turbulence fluctuations, the later is then modeled by appropriate turbulence model

Turbulence Models

4. Large Eddy Simulation (LES)
   - Sub-grid scale (SGS) turbulence fluctuations (smaller than control volume) are modeled
   - Grid scale (GS) turbulence fluctuations (larger than control volume) are resolved directly from unsteady NS equations
   - Dependent variables are decomposed into volume-averaging and fluctuation part
   - Unsteady simulation
   - Much more expensive than RANS, but also more accurate
   - In ventilation - capture low Reynolds number effects
   - More details on LES can be found in Davidson et al (1996, 1998, 2000) and Sagaut (2002)

Flow Modelling – CFD Approach



$$\frac{\partial(\rho\tilde{u}_i)}{\partial t}+\frac{\partial(\rho\tilde{u}_i\tilde{u}_j)}{\partial x_j}=-\frac{\partial\tilde{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial\tilde{u}_i}{\partial x_j}-\rho\left(\widetilde{u_iu_j}-\tilde{u}_i\tilde{u}_j\right)\right)$$ **LES**

Spatial Filtering

$$\frac{\partial(\rho u_i)}{\partial t}+\frac{\partial(\rho u_iu_j)}{\partial x_j}=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial u_i}{\partial x_j}\right)$$ **DNS**

Time Averaging

$$\frac{\partial(\rho\bar{u}_i)}{\partial t}+\frac{\partial(\rho\bar{u}_i\bar{u}_j)}{\partial x_j}=-\frac{\partial\bar{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\mu\frac{\partial\bar{u}_i}{\partial x_j}-\rho\overline{u_i'u_j'}\right)$$ **RANS**

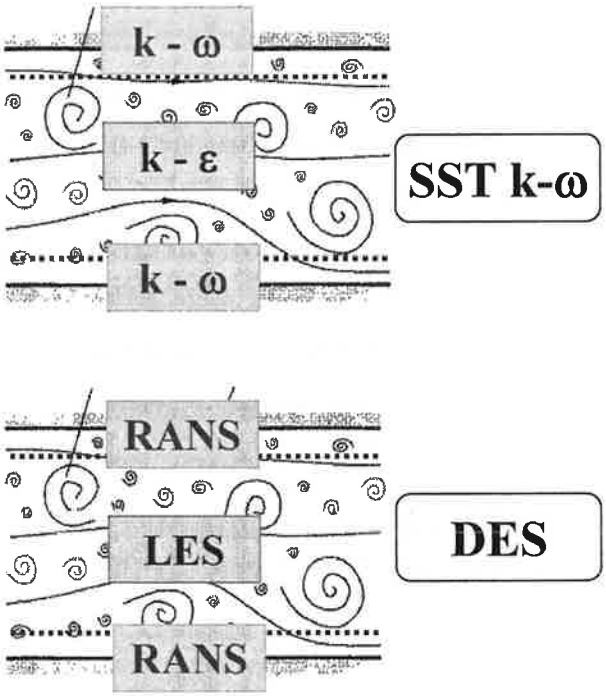
model

Different Turbulence Model

Indoor Environment Turbulence Models

- RANS
  - EVM
    - 0-eq.: mixinglength; Indoor model
    - 1-eq. S-A
    - 2-eq.: k-ε (Standard; Zonal; LRN); k-ω; k-kl
    - 3-eq.: k-ε-$\overline{\theta'^2}$; k-ε-$\overline{v'^2}$ (v2f)
    - n-eq. n ≥ 4
  - RSM: ASM; DSM
- LES — Smagorinsky, SL, WALE, Dynamic
- DNS

Coupling between models



Courtesy of Prof. Chen Qingyan

Different Turbulence Model

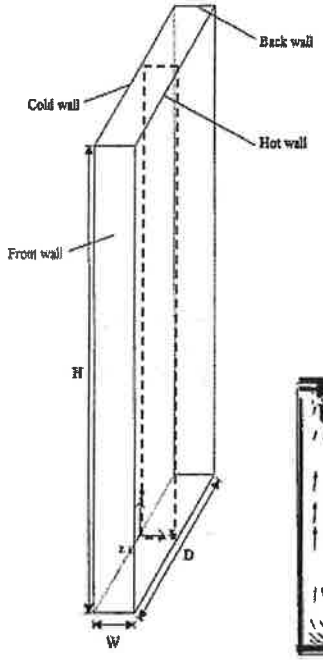
Indoor Environment Modeling: Models Tested

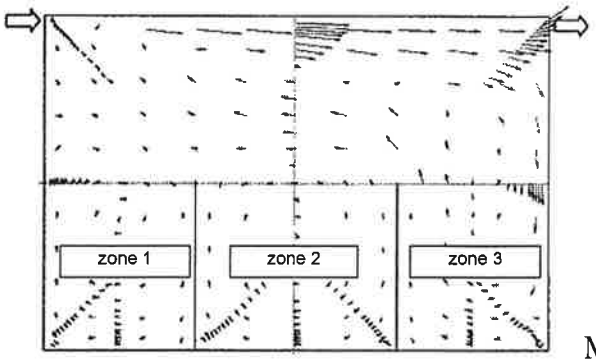
Group		Tested for indoor airflows	Selected for comparison
EVM	Zero eqn	Indoor zero-eq. (Chen and Xu, 1999)	Indoor zero eq.
	Two eqns	Standard k-ε (Launder and Spalding, 1974) RNG k- ε (Yakhot and Orszag, 1986) realizable k- ε (Shih et al., 1995)	RNG k- ε
		Launder-Sharma (1974) LRN k- ε Lam-Bremhorst (1981) LRN k- ε AKN LRN k- ε (Abe et al., 1994)	Launder-Sharma
		Standard k-ω (Wilcox, 1988) SST k-ω (Menter, 1994)	SST
	Three eqns	v2f (Durbin 1995) v2f variant I (Kalizin 1999) v2f variant II (Davidson et al. 2003)	Variant II
RSM		IP (Gibson and Launder, 1978) SSG (Speziale et al., 1991)	IP
LES		Smagorinsky SGS (Smagorinsky, 1963) Dynamic SGS (Zhang and Chen, 2000)	Smagorinsky SGS
DES		Spalart-Allmaras (1992) one equation EVM and LES (Deardorff 1970)	DES (S-A)

Different Turbulence Model

Indoor Environment Modeling: Benchmark Cases

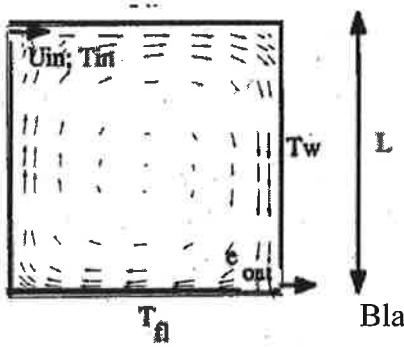
Betts and Bokhari (2000)

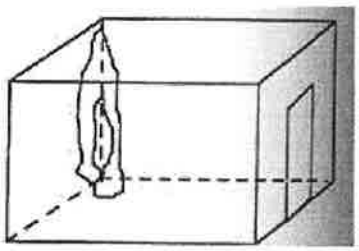




Ito et al (2000)

Murakami et al. (1995)





Blay et al (1992)

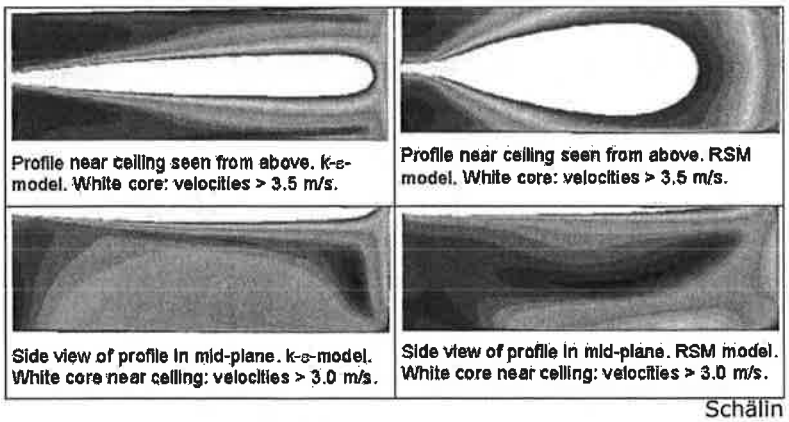
Different Turbulence Model

Indoor Environment Modeling: Model Performance

Cases	Compared items	Turbulence models							
		0-eq.	RNG k-e	SST k-w	LRN -LS	V2f-dav	RSM-IP	DES	LES
Natural convec-tion	Mean Temp.	B	A	A	C	A	A	C	A
	Mean Velo.	D	B	A	B	A	B	D	B
	Turbulence	n/a	C	C	C	A	C	C	A
Forced convec-tion	Mean Velo.	C	A	C	A	A	B	C	A
	Turbulence	n/a	B	C	B	B	B	C	B
Mixed convec-tion	Mean Temp.	A	A	A	A	A	B	B	A
	Mean Velo.	A	B	B	B	A	A	B	B
	Turbulence	n/a	A	D	B	A	A	B	B
Strong buo-yancy flow	Mean Temp.	A	A	A	A	A	n/c	n/a	B
	Mean Velo.	B	A	A	A	A	n/c	n/a	A
	Turbulence	n/a	C	A	B	B	n/c	n/a	B
Computing time (unit)		1	2 - 4		4 - 8		10	10^2 - 10^3	

Different Turbulence Model

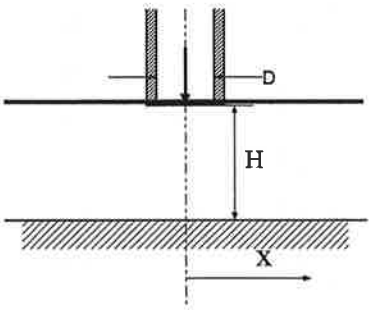
- 3D ceiling jet in a room simulated by a k-epsilon Model and a Reynolds Stress Model
- With k-ε, the ceiling jet becomes too thick. However, with RSM the RSM remains thin due to strong damping of turbulent fluctuations perpendicular to ceiling
- RSM provide better prediction on spreading rate of ceiling jet due to anisotropy fluctuation model associated with it

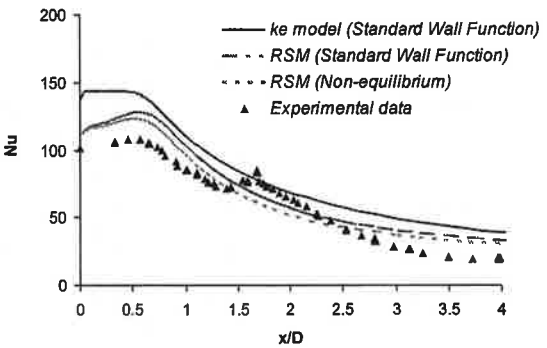


Schälin and Nielsen

Different Turbulence Model

- Turbulent impinging jet heat transfer with different turbulence model
- Experimental data from S. Ashforth-Frost and K. Jambunathan (*International Journal for Numerical Methods in Fluids, Vol.23, 295-306, 1996*)
- Circular jet (D=10mm, H/D=2, Re=23000)
- Standard k-ε model tends to over-predict heat transfer; as the viscous-sublayer at the stagnation zone in the impinging jet governs heat transfer process; but this is not captured by k-ε model which is only suitable for fully turbulent flow modeling

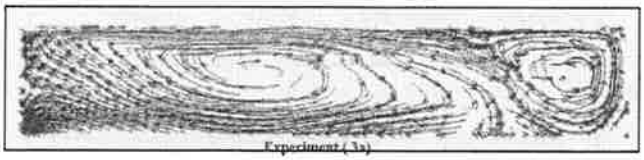




231

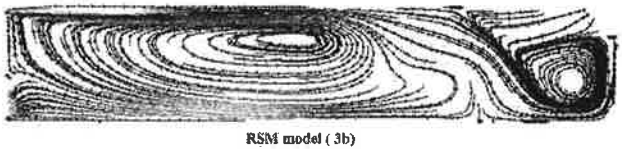
Different Turbulence Model

Turbulence Models



Experiment (3a)

LDV Measurements



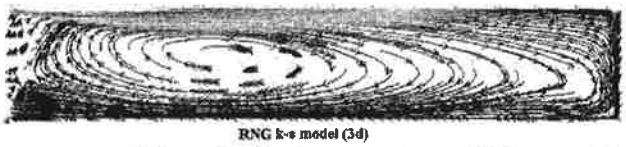
RSM model (3b)

RSM Fine grid: 624,000 cells



k-e model (3c)

Standard κ - ε



RNG k-e model (3d)

RNG κ - ε



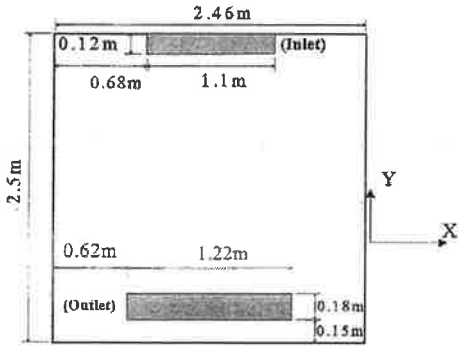
RSM model with a coarse grid (3e)

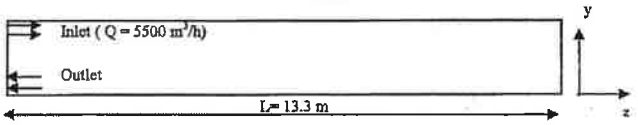
RSM Coarse grid: 240 000 cells

Different Turbulence Model

Comparison between Turbulence Models : Refrigerated Transport cabin

Container
scale 1: 3.3





Moureh & Flick (2002), Centre de Thermique, France

Different Turbulence Model

Comparison between Turbulence Models/...

Turbulence model	*Standard k-ε*	*Low-Re k-ε*	*RNG*	*RSM*	*LES*
Internal flows with weak streamline curvature (e.g. duct flow)	1	1	1	1	1
Flows with strong streamline curvature (e.g. external flow, separation, etc.)	3, 4	3, 4		1	1
Jets: Normal **Swirl** **Impinging**	1 3, 4 2, 3	1 3, 4 2, 3	1 1, 2	1 1 1	1 1 1
Cavity flow (non-isothermal): **Weak stratification** **Strong stratification (e.g. negatively buoyant)**	 1 3, 4	 1 3, 4	 1 3, 4	 1 1, 2	 1 1
Forced convection	2, 3	1, 2	2, 3	2, 3	1, 2
Mixed convection	2, 3	1, 2	1	2, 3	2, 3
Low Reynolds number flow	3, 4	2	2	3, 4	1
Unsteady flow (e.g. vortex shedding)	4	2		2	1

Note: 1 (excellent), 2 (good), 3 (fair) and 4 (poor/unacceptable) but where no score is given there is little or no information available

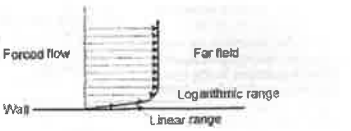
RNG – Isotropic

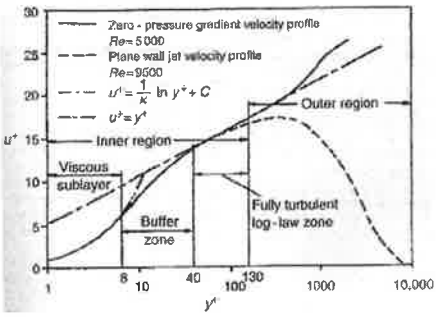
Wall Function in Turbulent BL

- Character of near Wall Turbulence
- Implication for the modeling of near wall turbulent flow
- The Wall Function Strategy
- Limitation of Conventional Wall Function

Wall Functions

- Laminar sub-layer: $y+ < 8$
- Buffer zone: $8 < y+ < 40$
- Log-law fully turbulent zone (inner): $40 < y+ < 130$
- Outer turbulent zone: $y+ > 130$ (wake zone)
- Numerical values of these constants vary from one report to another. In view of al assumptions that preceded law of the wall, slight adjustments are not crucial

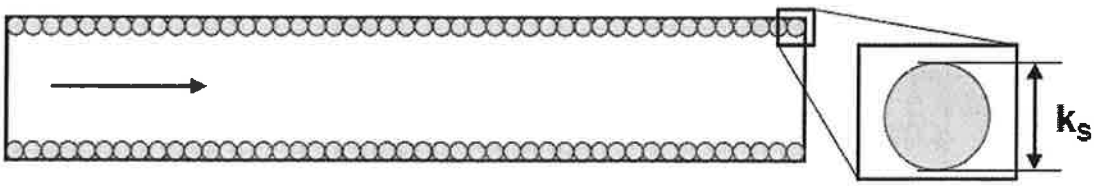




ignore viscos sublayer

Surface Roughness

- Most commercial CFD codes were established for **mechanical engineering problems** rather than for **atmospheric flow problems**
- Wall function roughness modifications based on semi-empirical information for **sand-grain roughened pipes and channels (Nikuradse 1933).**



- Small-scale roughness (expressed by equivalent sand-grain roughness height k_S)

$$u = \frac{u^*}{\kappa} \ln\left(\frac{y}{k_s^+}\right) + 8.5$$

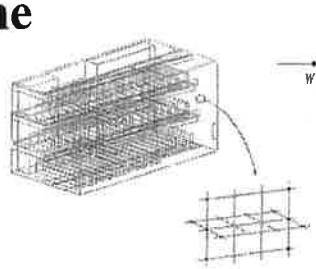
- Atmospheric flows: large-scale roughness (expressed by aerodynamic roughness length y_0)
- Many commercial CFD codes (Fluent, Ansys-CFX, …) provide only the first type of roughness modification.

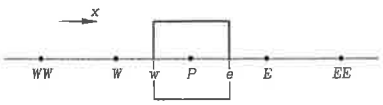
Blocken B, Stathopoulos T, Carmeliet J. 2007. CFD simulation of the Atmospheric Boundary Layer – wall function problems. ***Atmospheric Environment 41(2): 238-252.***

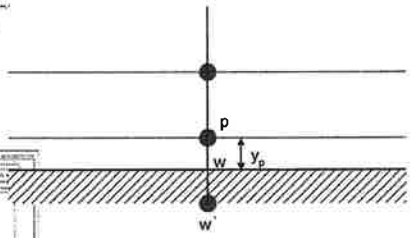
Accuracy of CFD Simulation

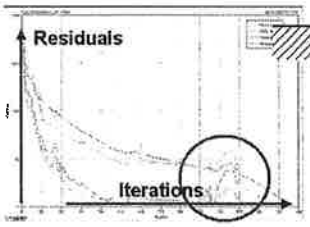
The accuracy of a CFD solution depends on a number of factors and in particular the following:

- **Discretization scheme**
- **Computational grid**
- **Near-wall boundary conditions**
- **Convergence criteria**









Verification and Validation in CFD

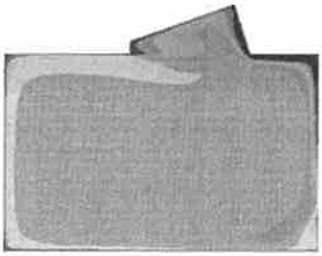
- Definition given in AIAA guide (1998) & Roache (1998)
  - Verification
    - Procedure to ensure the program solves the equations correctly
  - Validation
    - Procedure to test the extent to which the model accurately represent reality
  - Calibration
    - Procedure to assess CFD code to predict global quantities of interest for specific geometry of engineering design interest

Verification and Validation in CFD

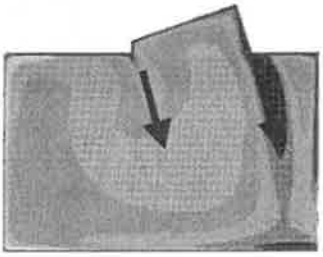


Various reasons for draught with different influences (entrance, façade, stack effect through various floor, wind....





2D steady state



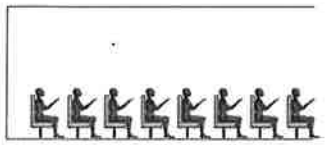
3D transient

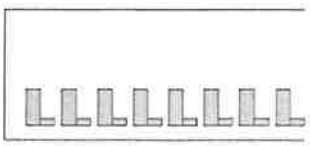
Strong downdraft under shed roof can only be captured with transient 3D modeling

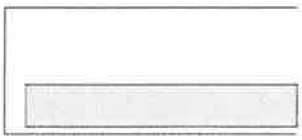
Verification and Validation in CFD

Sources of Errors and Uncertainties

Simplification, modelling level





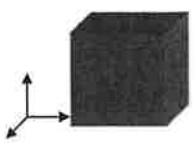


Creating 3D geometry model

Geometry

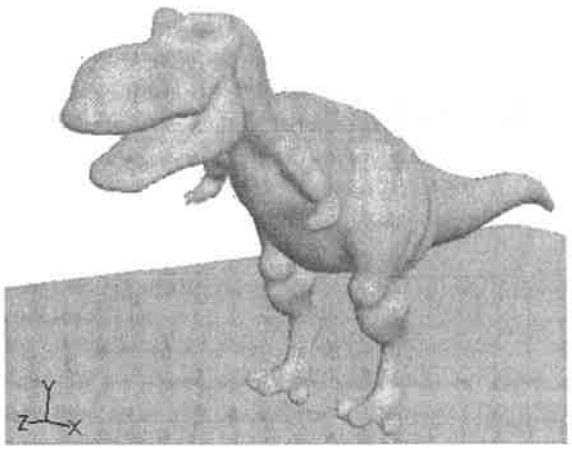
- The starting point for all problems is a "geometry."
- The geometry describes the shape of the problem to be analyzed.
- Can consist of volumes, faces (surfaces), edges (curves) and vertices (points).

Geometry can be very simple...



geometry for a "cube"

... or more complex



RSM - 1 lecture

Geometry creation

- Geometries can be created top-down or bottom-up.
  - **Top-down** refers to an approach where the computational domain is created by performing logical operations on primitive shapes such as cylinders, bricks, and spheres.
  - **Bottom-up** refers to an approach where one first creates vertices (points), connects those to form edges (lines), connects the edges to create faces, and combines the faces to create volumes.
- Importing third party geometry data (Building Information Model, BIM)
  - Created using other programs (e.g. CAD, graphics).

Creating 3D geometry model

- Factors to be concerned
  - Simplifying architecture data
    - The most time consuming part for CFD work – typically 70% of time spent on this part, especially geometry clean up
    - Good practice to build a model which does not have any unnecessary small gaps between objects
      - Retains desired connectivity
      - Avoids possible meshing problems
  - Creating water tight model
  - Preparing model for meshing

Creating 3D geometry model

- Model Building Considerations
  - Always start from a simple model
  - Add more complexities gradually
  - Reduce the size of your model
    - Neglect small details
    - Replace very thin objects by planar objects
    - Use symmetry walls if possible
  - Align very closely spaced faces

BIM-based Geometry

BIM & CFD Analysis

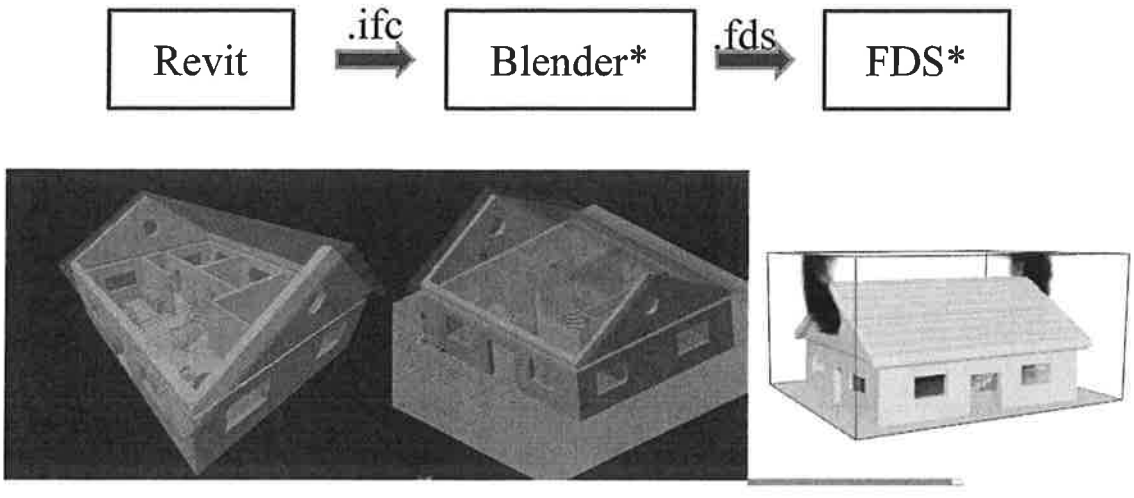
- BIM (Building Information Model) is a digital representation of the physical and functional characteristics of a building/facility.
- Building geometry and space information needed for CFD simulation can be obtained from a BIM model
- IFC (Industry Foundation Classes) is an open BIM standard developed by buildingSMART, which is an international alliance for interoperability to promote sharing of building information
- Commercial BIM-enabled design tools like Autodesk Revit, Graphisoft ArchiCAD and Bentley MicroStation all have the capability to export their BIM models into IFC format
- It is desirable for CFD tools to read in IFC files for direct CFD input on building geometry data and space information
- Design in BIM and sharing of BIM models would allow CFD consultants to establish the airflow simulation model much faster and shorten the design and analysis cycles significantly

BIM-based Geometry

- Cases studies
  - Fire modeling in a house
  - Ventilation in an operating theatre
  - Ventilation around a private house
- Efforts in geometry cleanup
- Sources of problems
- Preliminary remark

BIM-based Geometry - Case studies

- Operational procedure

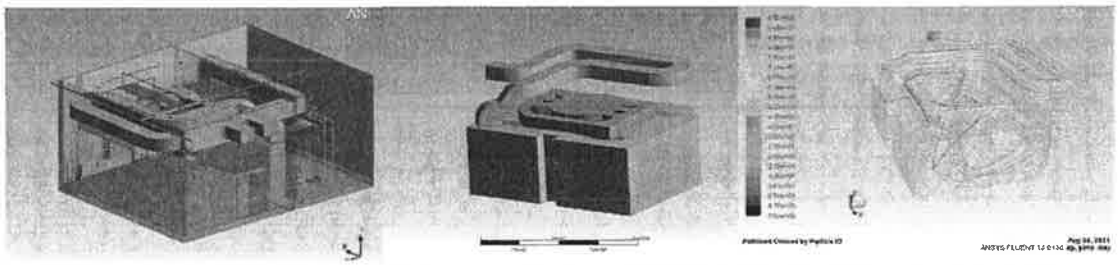


* Open-source codes, toll free

BIM-based Geometry - Case studies

- Operational procedure

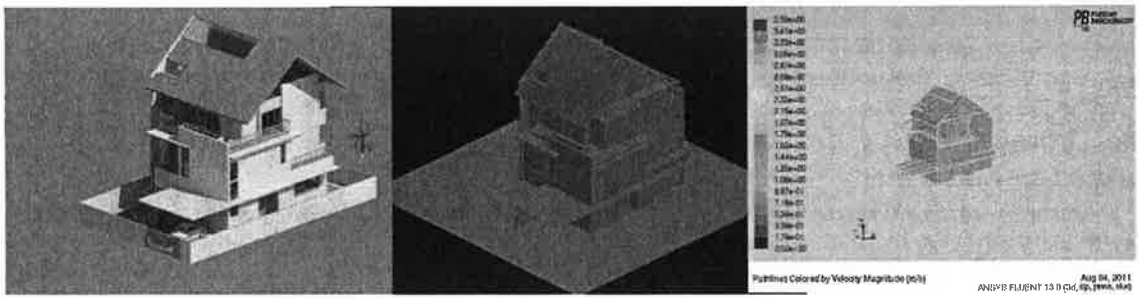
Revit → .sat → Workbench → .msh → Fluent



BIM-based Geometry - Case studies

- Operational procedure

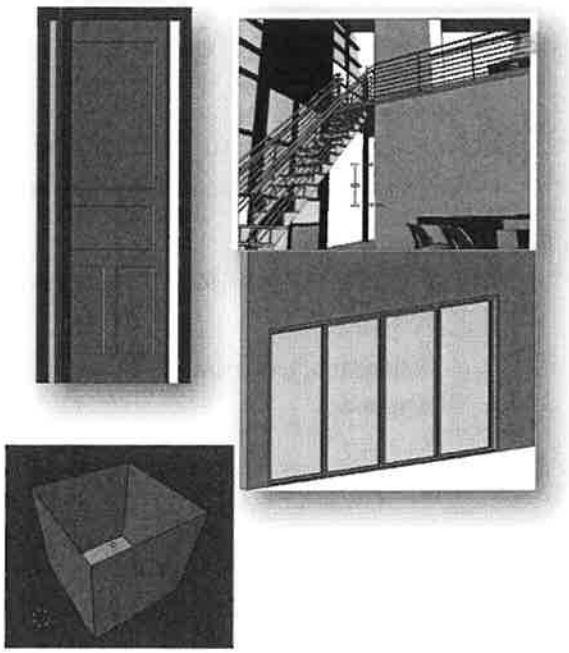
Revit → .sat → Gambit → .msh → FLUENT



BIM-based Geometry

Geometry cleanup

- Ignore small objects
  - Door handler, railings, …
- Simplify objects
  - Windows, doors, …
- Detect and FIX clashing objects
  - Beam, slab, floor, wall, …
- Identify and FIX gaps
  - Untouched objects
  - Templates for windows and doors
  - …
- Fix non-manifold objects



BIM-based Geometry

Sources of problems

- Human errors
  - Judging the integrity based on 'looking'
  - Unawareness of simulation requirements
- "Defects" due to templates available in BIM software
  - Windows, doors, …
- Requirements in CFD packages
  - FDS, ICEMCFD → allowable for small gaps
  - GAMBIT, DM → not allowed for small gaps
  - …

BIM-based Geometry

Some Remarks

- BIM model can speed up the CFD simulation process
  - Substantially avoid the efforts to create the 3D model from scratch or 2D drawings
- Guidelines in simulations requirements for geometry should be in place for BIM modelers
  - Collaboration between BIM modeler and simulation specialist in order to streamline the procedure

Day 3 (11:00am – 12:30pm)

- Meshing computational model
- Applying appropriate boundary condition and physical assumption
- Quality control
- Analyzing CFD result for design and improvement

Meshing Computational Domain

Outline

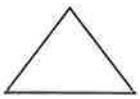
- Why is a grid needed?
- Element types.
- Grid types.
- Grid design guidelines.
- Geometry.
- Solution adaption.
- Grid import.

Why is a grid needed?

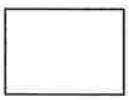
- The grid:
  - Designates the cells or elements on which the flow is solved.
  - Is a discrete representation of the geometry of the problem.
  - Has cells grouped into boundary zones where b.c.'s are applied.
- The grid has a significant impact on:
  - Rate of convergence (or even lack of convergence).
  - Solution accuracy.
  - CPU time required.
- Importance of mesh quality for good solutions.
  - Grid density.
  - Adjacent cell length/volume ratios.
  - Skewness.
  - Tet vs. hex.
  - Boundary layer mesh.
  - Mesh refinement through adaption.

Typical cell shapes

- Many different cell/element and grid types are available. Choice depends on the problem and the solver capabilities.
- Cell or element types:
  - 2D:

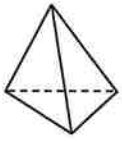


triangle ("**tri**")

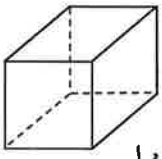


2D prism (**quadrilateral** or "**quad**")

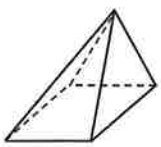
  - 3D:



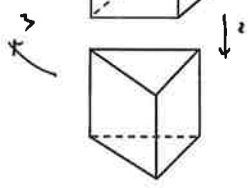
tetrahedron ("**tet**")



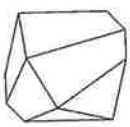
prism with quadrilateral base (**hexahedron** or "**hex**")



pyramid



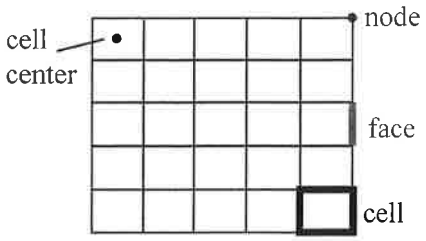
prism with triangular base (**wedge**)



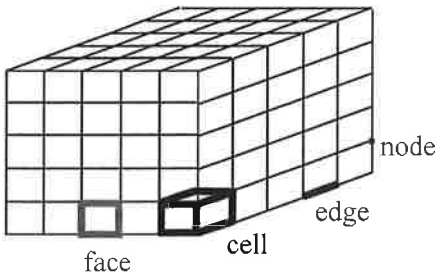
arbitrary polyhedron

Terminology

- Cell = control volume into which domain is broken up.
- Node = grid point.
- Cell center = center of a cell.
- Edge = boundary of a face.
- Face = boundary of a cell.
- Zone = grouping of nodes, faces, and cells:
  - Wall boundary zone.
  - Fluid cell zone.
- Domain = group of node, face and cell zones.



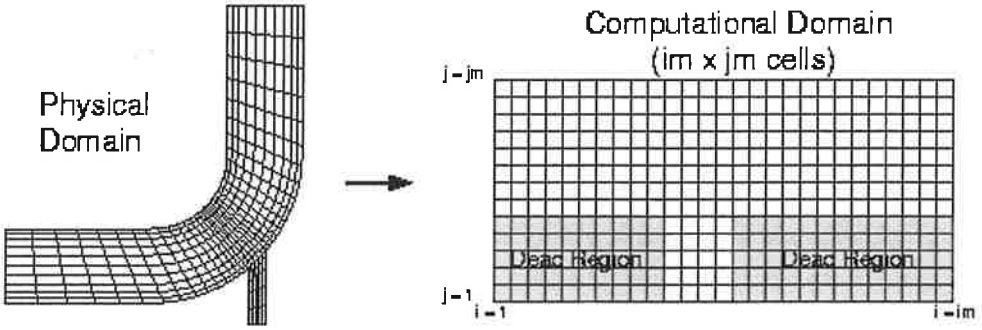
2D computational grid



3D computational grid

Grid types: structured grid

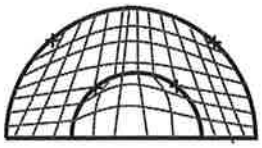
- Single-block, structured grid.
  - i,j,k indexing to locate neighboring cells.
  - Grid lines must pass all through domain.
- Obviously can't be used for very complicated geometries.



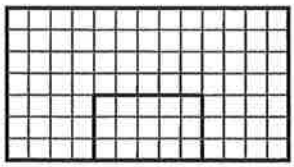
Face meshing: structured grids

- Different types of hexahedral grids.
- Single-block.
  - The mesh has to be represented in a single block.
  - Connectivity information (identifying cell neighbors) for entire mesh is accessed by three index variables: i, j, k.

Single-block geometry



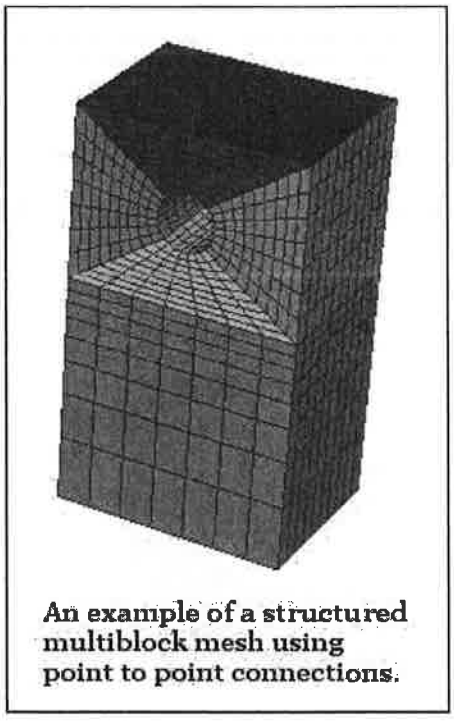
Logical representation.



- Single-block meshes may include 180 degree corners.

Grid types: multiblock

- Multi-block, structured grid.
  - Uses i,j,k indexing within each mesh block.
  - The grid can be made up of (somewhat) arbitrarily-connected blocks.
- More flexible than single block, but still limited.



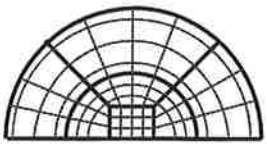
An example of a structured multiblock mesh using point to point connections.

Source: www.cfdreview.com

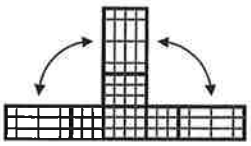
Face meshing: multiblock

- Different types of hexahedral grids.
  - Multi-block.
  - The mesh can be represented in multiple blocks.

Multi-block geometry



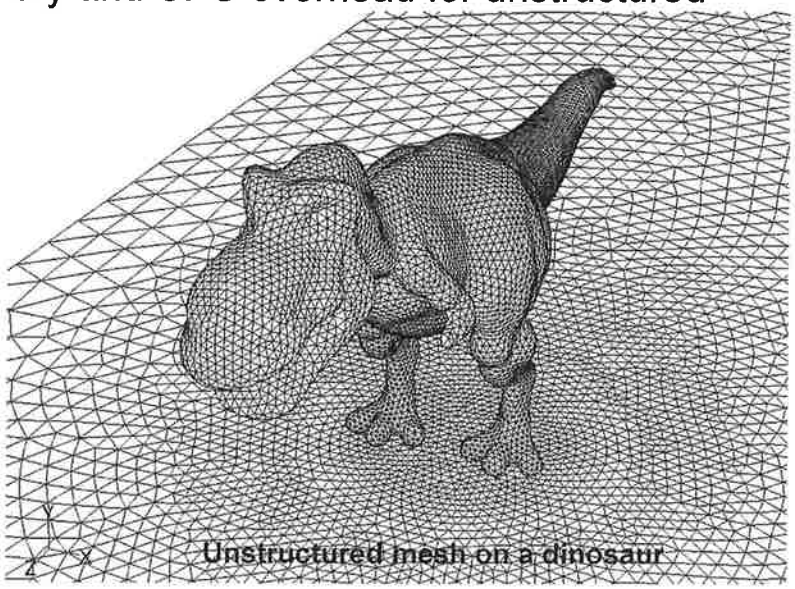
Logical representation.



  - This structure gives full control of the mesh grading, using edge meshing, with high-quality elements.
  - Manual creation of multi-block structures is usually more time-consuming compared to unstructured meshes.

Grid types: unstructured

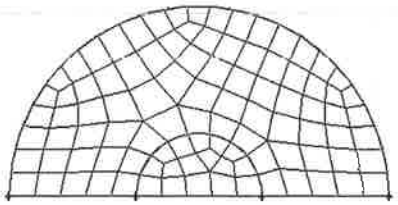
- Unstructured grid.
  - The cells are arranged in an arbitrary fashion.
  - No i,j,k grid index, no constraints on cell layout.
- There is some memory and CPU overhead for unstructured referencing.



Unstructured mesh on a dinosaur

Face meshing: unstructured grids

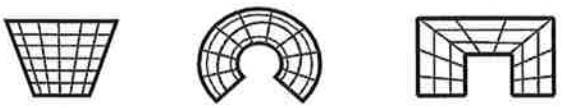
- Different types of hexahedral grids.
  - Unstructured.
  - The mesh has no logical representation.



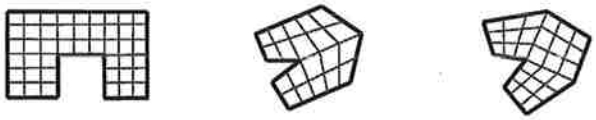
Unstructured Grid

Face meshing: quad examples

- Quad: Map.



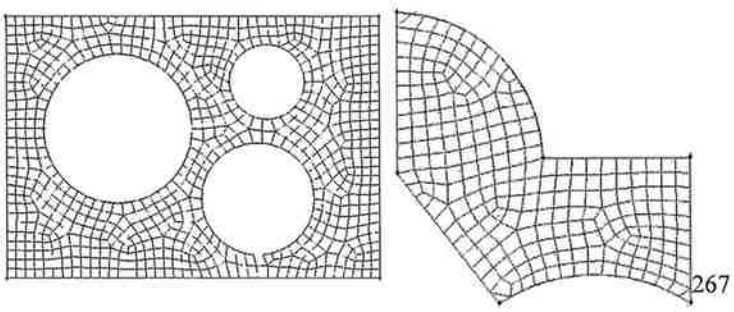
- Quad: Submap.



- Quad: Tri-Primitive.

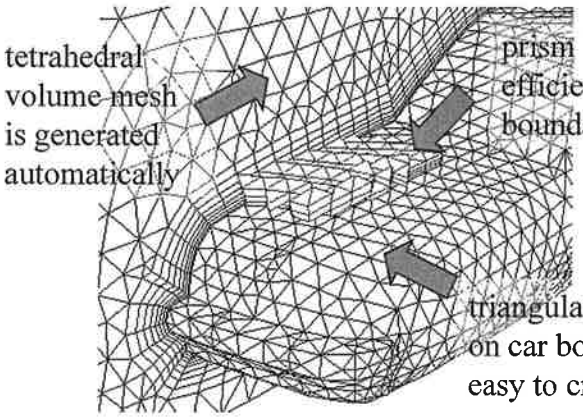


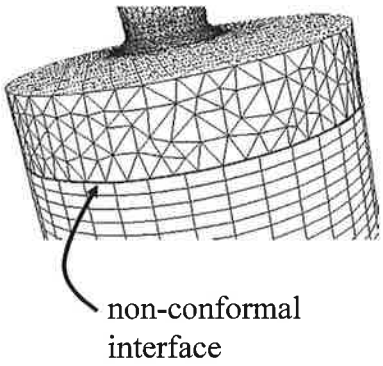
- Quad: Pave and Tri-Pave.



Grid types: hybrid

- Hybrid grid.
  - Use the most appropriate cell type in any combination.
    - Triangles and quadrilaterals in 2D.
    - Tetrahedra, prisms and pyramids in 3D.
  - Can be non-conformal: grids lines don't need to match at block boundaries.

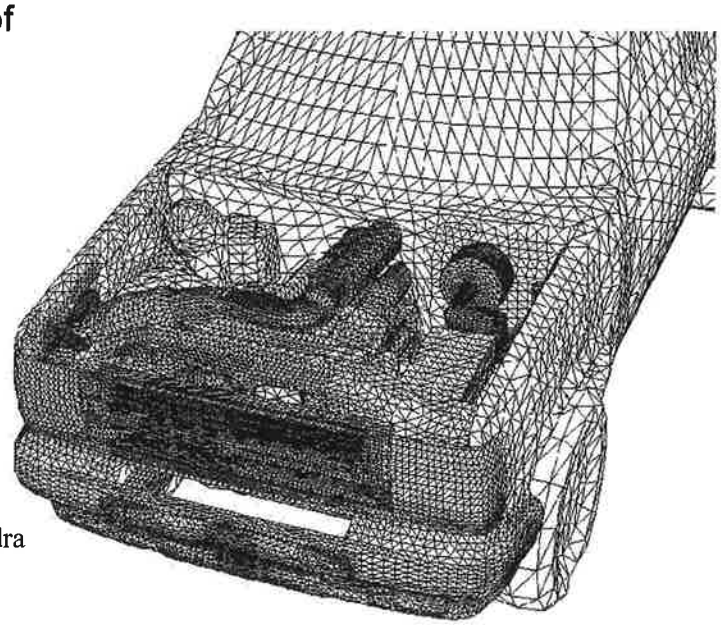




Tetrahedral mesh

- Start from 3D boundary mesh containing only triangular faces.
- Generate mesh consisting of tetrahedra.

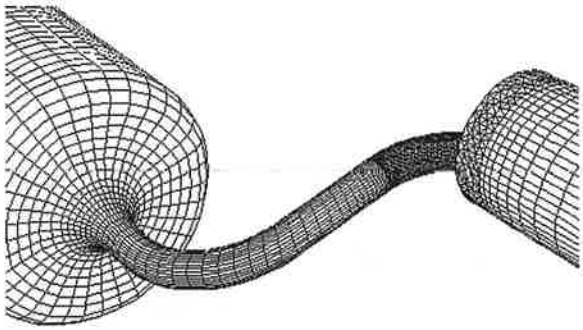
Complex Geometries



Surface mesh for a grid containing only tetrahedra

Zonal hybrid mesh

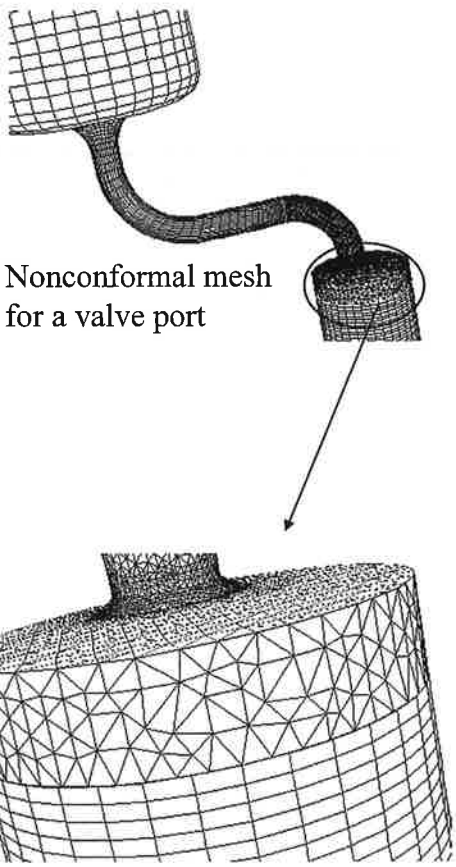
- Flow alignment well defined in specific regions.
- Start from 3D boundary and volume mesh:
  - Triangular and quadrilateral faces.
  - Hexahedral cells.
- Generate zonal hybrid mesh, using:
  - Tetrahedra.
  - Existing hexahedra.
  - Transition elements: pyramids.



Surface mesh for a grid containing hexahedra, pyramids, and tetrahedra (and prisms)

Nonconformal mesh

- Parametric study of complex geometries.
- Nonconformal capability allows you to replace portion of mesh being changed.
- Start from 3D boundary mesh or volume mesh.
- Add or replace certain parts of mesh.
- Remesh volume if necessary.



Mesh naming conventions - topology

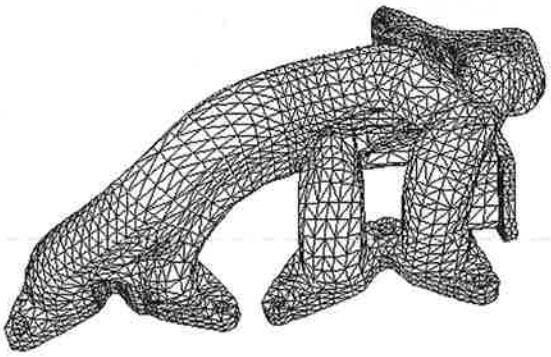
- Structured mesh: the mesh follows a structured i,j,k convention.
- Unstructured mesh: no regularity to the mesh.
- Multiblock: the mesh consists of multiple blocks, each of which can be either structured or unstructured.

Mesh naming conventions – cell type

- Tri mesh: mesh consisting entirely of triangular elements.
- Quad mesh: consists entirely of quadrilateral elements.
- Hex mesh: consists entirely of hexahedral elements.
- Tet mesh: mesh with only tetrahedral elements.
- Hybrid mesh: mesh with one of the following:
  - Triangles and quadrilaterals in 2D.
  - Any combination of tetrahedra, prisms, pyramids in 3D.
  - Boundary layer mesh: prizms at walls and tetrahedra everywhere else.
  - Hexcore: hexahedra in center and other cell types at walls.
- Polyhedral mesh: consists of arbitrary polyhedra.
- Nonconformal mesh: mesh in which grid nodes do not match up along an interface.

Mesh generation process

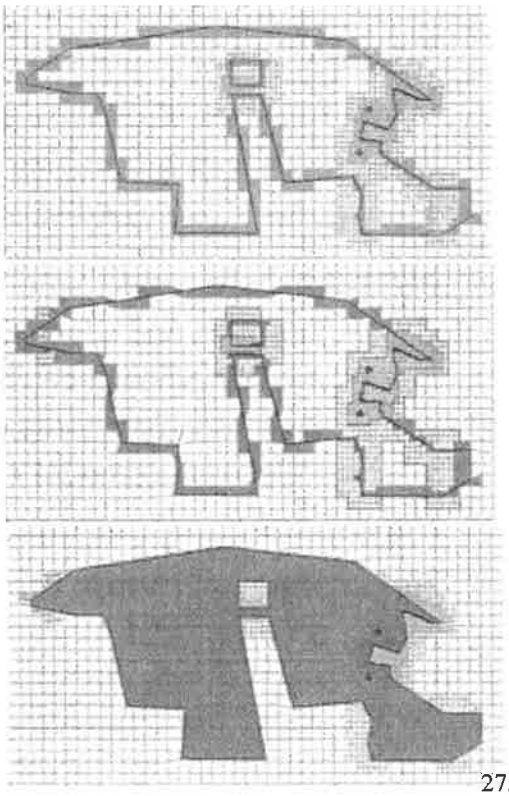
1. Create, read (or import) boundary mesh(es).
2. Check quality of boundary mesh.
3. Improve and repair boundary mesh.
4. Generate volume mesh.
5. Perform further refinement if required.
6. Inspect quality of volume mesh.
7. Remove sliver and degenerate cells.
8. Save volume mesh.



Surface mesh for a grid containing only tetrahedra

Cutcell mesh generation process

1. Volume Objects created
2. Uniform Cartesian mesh is generated
3. Cells intersecting the geometry are marked
4. Refinement is carried out based on curvature, proximity, etc.
5. Nodes are projected onto the geometry and if quality is degraded cells are cut. Result is layer of hex, tet, prism and pyramid on the boundary.
6. Cells are separated into different cell zones according to the definition of the Volume Objects
7. New Boundaries are extracted corresponding to the input geometry



Tri/tet grid generation process

- Two phases:
  - **Initial mesh generation**: Triangulate boundary mesh.
  - **Refinement on initial mesh**: Insert new nodes.



Initial mesh



Boundary refinement



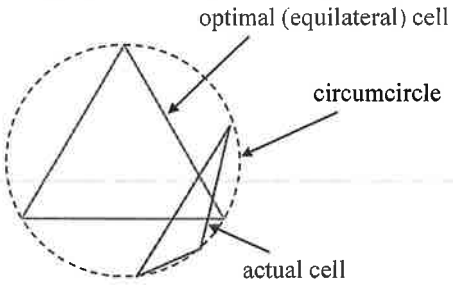
Cell zone refinement

Mesh quality

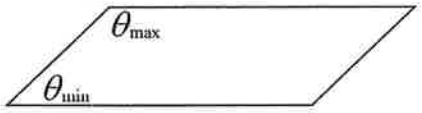
- For the same cell count, hexahedral meshes will give more accurate solutions, especially if the grid lines are aligned with the flow.
- The mesh density should be high enough to capture all relevant flow features.
- The mesh adjacent to the wall should be fine enough to resolve the boundary layer flow. In boundary layers, quad, hex, and prism/wedge cells are preferred over tri's, tets, or pyramids.
- Three measures of quality:
  - Skewness.
  - Smoothness (change in size).
  - Aspect ratio.

Mesh quality: skewness

- Two methods for determining skewness:
  1. Based on the equilateral volume:
     - Skewness = $\dfrac{\text{optimal cell size} - \text{cell size}}{\text{optimal cell size}}$
     - Applies only to triangles and tetrahedra.
     - Default method for tris and tets.



  2. Based on the deviation from a normalized equilateral angle:
     - Skewness (for a quad) = $\max\left[\dfrac{\theta_{max} - 90}{90}, \dfrac{90 - \theta_{min}}{90}\right]$
     - Applies to all cell and face shapes.
     - Always used for prisms and pyramids.



0.7x 40 OK

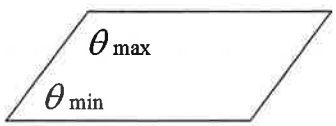
Equiangle skewness

- Common measure of quality is based on equiangle skew.
- Definition of equiangle skew:

$$\max\left[\frac{\theta_{max} - \theta_e}{180 - \theta_e}, \frac{\theta_e - \theta_{min}}{\theta_e}\right]$$

where:

- θ_{max} = largest angle in face or cell.
- θ_{min} = smallest angle in face or cell.
- θ_e = angle for equiangular face or cell.
  - e.g., 60 for triangle, 90 for square.

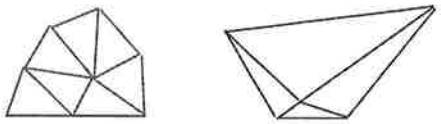


- Range of skewness:



Mesh quality: smoothness and aspect ratio

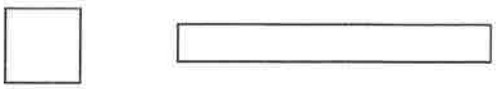
- Change in size should be gradual (smooth).



smooth change in cell size

large jump in cell size

- Aspect ratio is ratio of longest edge length to shortest edge length. Equal to 1 (ideal) for an equilateral triangle or a square.



aspect ratio = 1

high-aspect-ratio quad



aspect ratio = 1

high-aspect-ratio triangle

not less than 15°

not more than 20%. e.g. x 1.2

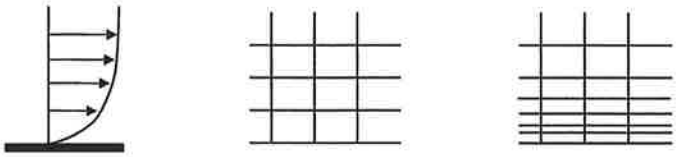
Striving for quality

- A poor quality grid will cause inaccurate solutions and/or slow convergence.
- Minimize equiangle skew:
  - Hex and quad cells: skewness should not exceed 0.85.
  - Tri's: skewness should not exceed 0.85.
  - Tets: skewness should not exceed 0.9.
- Minimize local variations in cell size:
  - E.g. adjacent cells should not have 'size ratio' greater than 20%.
- If such violations exist: delete mesh, perform necessary decomposition and/or pre-mesh edges and faces, and remesh.

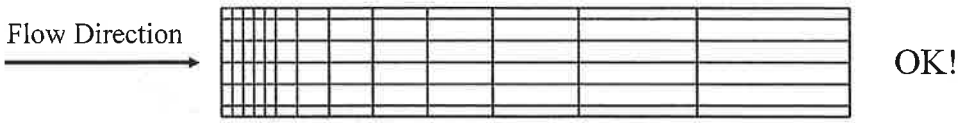
Value of Skewness	0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.99	0.99-1.00
Cell Quality	excellent	good	acceptable	poor	sliver	degenerate

Grid design guidelines: resolution

- Pertinent flow features should be adequately resolved.



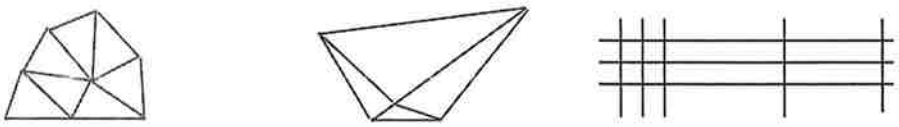
- Cell aspect ratio (width/height) should be near one where flow is multi-dimensional.
- Quad/hex cells can be stretched where flow is fully-developed and essentially one-dimensional.



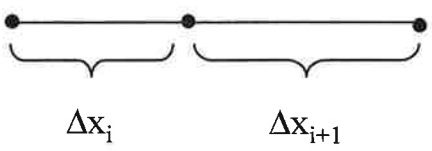
Flow mesh for abrupt changes

Grid design guidelines: smoothness

- Change in cell/element size should be gradual (smooth).



- Ideally, the maximum change in grid spacing should be <20%:



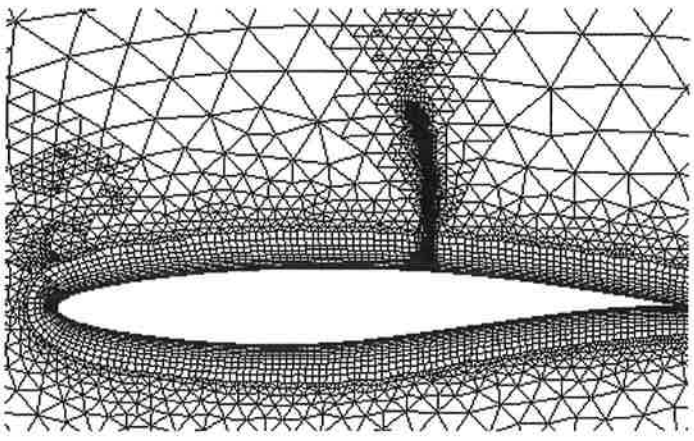
$$\frac{\Delta x_{i+1}}{\Delta x_i} \leq 1.2$$

Grid design guidelines: total cell count

- More cells *can* give higher accuracy. The downside is increased memory and CPU time.
- To keep cell count down:
  - Use a non-uniform grid to cluster cells only where they are needed.
  - Use solution adaption to further refine only selected areas.
- Cell counts of the order:
  - 1E4 are relatively small problems.
  - 1E5 are intermediate size problems.
  - 1E6 are large. Such problems can be efficiently run using multiple CPUs, but mesh generation and post-processing may become slow.
  - 1E7 are huge and should be avoided if possible. However, they are common in aerospace and automotive applications.
  - 1E8 and more are department of defense style applications.

Solution adaption

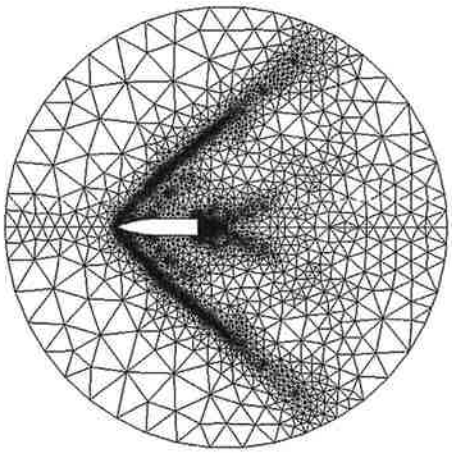
- How do you ensure adequate grid resolution, when you don't necessarily know the flow features? Solution-based grid adaption!
- The grid can be refined or coarsened by the solver based on the developing flow:
  - Solution values.
  - Gradients.
  - Along a boundary.
  - Inside a certain region.



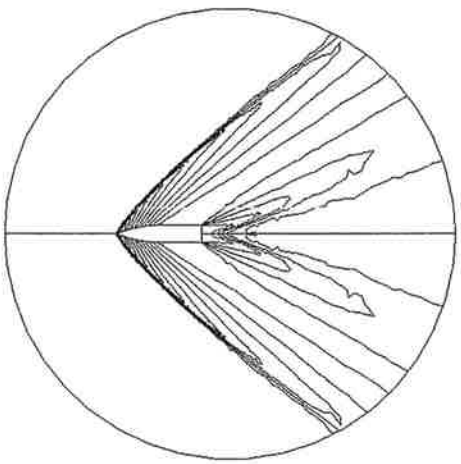
Grid adaption

- Grid adaption adds more cells where needed to resolve the flow field.
- Fluent adapts on cells listed in register. Registers can be defined based on:
  - Gradients of flow or user-defined variables.
  - Isovalues of flow or user-defined variables.
  - All cells on a boundary.
  - All cells in a region.
  - Cell volumes or volume changes.
  - y^+ in cells adjacent to walls.
- To assist adaption process, you can:
  - Combine adaption registers.
  - Draw contours of adaption function.
  - Display cells marked for adaption.
  - Limit adaption based on cell size and number of cells.

Adaption example: final grid and solution



2D planar shell - final grid



2D planar shell - contours of pressure final grid

Main sources of errors

- Mesh too coarse.
- High skewness.
- Large jumps in volume between adjacent cells.
- Large aspect ratios.
- Interpolation errors at non-conformal interfaces.
- Inappropriate boundary layer mesh.

Summary

- Design and construction of a quality grid is crucial to the success of the CFD analysis.
- Appropriate choice of grid type depends on:
  - Geometric complexity.
  - Flow field.
  - Cell and element types supported by solver.
- Hybrid meshing offers the greatest flexibility.
- Take advantage of solution adaption.

Boundary Conditions

Defining Boundary Conditions

- To define a problem that results in a unique solution, you must specify information on the flow variables at boundaries.
  - Specify fluxes of mass, momentum, energy, etc. into the domain.
- Defining boundary conditions involves:
  - Identifying the boundary locations
  - Supplying information at the boundaries
- The data required at a boundary depends upon the boundary condition type and the physical models employed.
- You must be aware of the information that is required of the boundary condition and locate the boundaries where the information on the flow variables are known or can be reasonably approximated
  - Poorly defined boundary conditions can have a significant impact on your solution

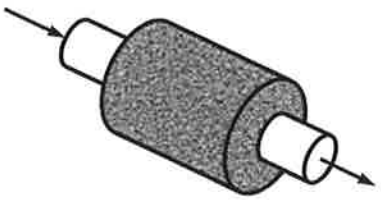
Cell Zones – Fluid

- A fluid cell zone is a group of cells for which all active equations are solved.
- Fluid material selection is required.
  - For multiple species or multiphase flows, the material is not shown. Instead, the fluid zone consists of the mixture of the phases.
- Optional inputs
  - Porous region
  - Source terms
  - Laminar region
  - Fixed Values
  - Radiation



Porous Media

- A porous zone is a special type of fluid zone.
  - Enable Porous Zone option in the Fluid panel.
  - Pressure loss in flow determined via user inputs of resistance coefficients to lumped parameter model



- Used to model flow through porous media and other uniformly distributed flow resistances.
  - Packed beds
  - Filter papers
  - Perforated plates
  - Flow distributors
  - Tube banks

$$\Delta p_x \approx \sum_{j=1}^{3} C_{2,j} \Delta n_x \tfrac{1}{2} \rho v_j |v|$$

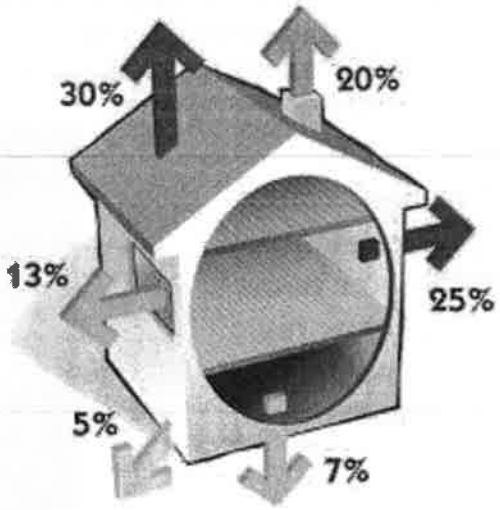
$$\Delta p_y \approx \sum_{j=1}^{3} C_{2,j} \Delta n_y \tfrac{1}{2} \rho v_j |v|$$

$$\Delta p_z \approx \sum_{j=1}^{3} C_{2,j} \Delta n_z \tfrac{1}{2} \rho v_j |v|$$

- Inputs are directional viscous and inertial resistance coefficients.

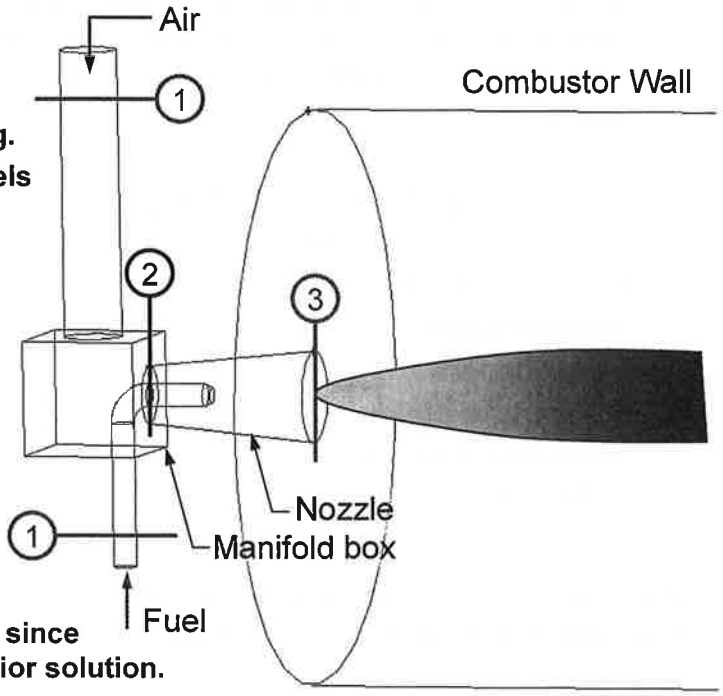
Cell Zones – Solid

- A solid zone is a group of cells for which only heat conduction problem solved. Flow equations are not solved.
- Only required input is the material name (defined in the Materials panel).
- Optional inputs allow you to set volumetric heat generation rate (heat source).
- Need to specify rotation axis if rotationally periodic boundaries adjacent to solid zone.
- Can define motion for a solid zone



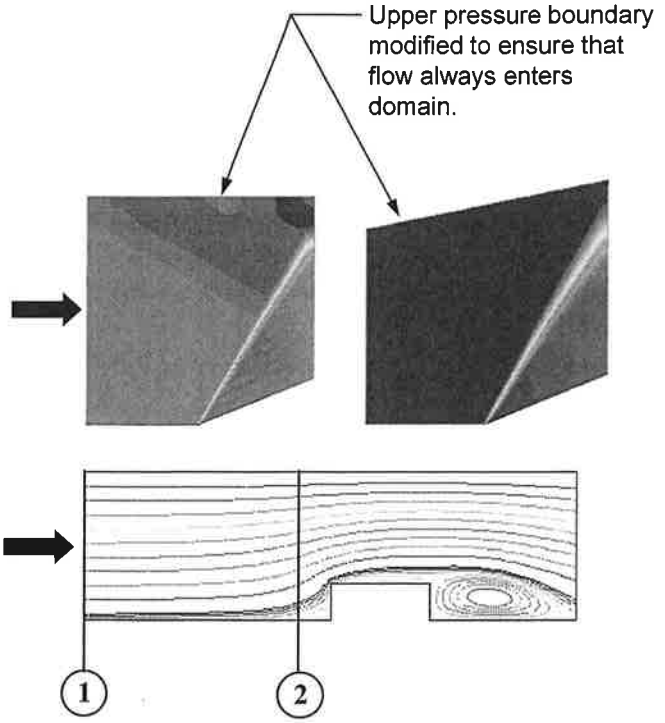
Locating Boundaries – An Example

- Three possible approaches in locating inlet boundaries for this example:
  1. Upstream of manifold
     - Can use uniform profile.
     - Properly accounts for mixing.
     - Non-premixed reaction models
     - Requires more cells.
  2. Nozzle inlet plane
     - Non-premixed reaction models.
     - Requires accurate inlet profile.
     - Flow is still non-premixed.
  3. 3 Nozzle outlet plane
     - Premixed reaction model.
     - Requires accurate profile.
     - Not generally recommended since inlet BCs may drive the interior solution.



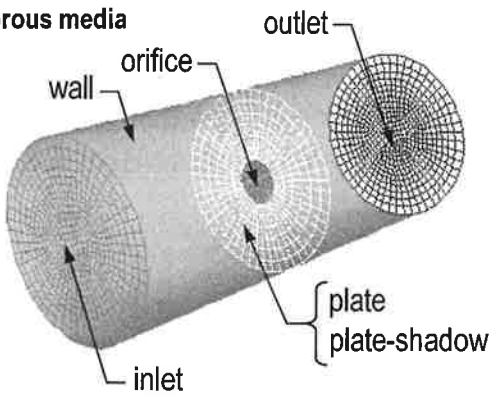
General Guidelines

- If possible, select boundary location and shape such that flow either goes in or out.
  - Not necessary, but will typically observe better convergence.
- Should not observe large gradients in direction normal to boundary.
  - Indicates incorrect set-up.
- Minimize mesh skewness near the boundary.
  - Otherwise it would introduce error early in calculation.



Generic Boundary Condition Types

- External Boundaries
  - General
    - Pressure Inlet
    - Pressure Outlet
  - Incompressible
    - Velocity Inlet
    - Outflow (not recommended)
  - Compressible
    - Mass Flow Inlet
    - Pressure Far Field
  - Other
    - Wall
    - Symmetry
    - Axis
    - Periodic
  - Special
    - Inlet / Outlet Vent
    - Intake / Exhaust Fan
- Internal Boundaries
  - Fan
  - Interior
  - Porous Jump
  - Radiator
  - Wall
- Cell (Continuum) zones
  - Fluid
  - Solid
  - Porous media

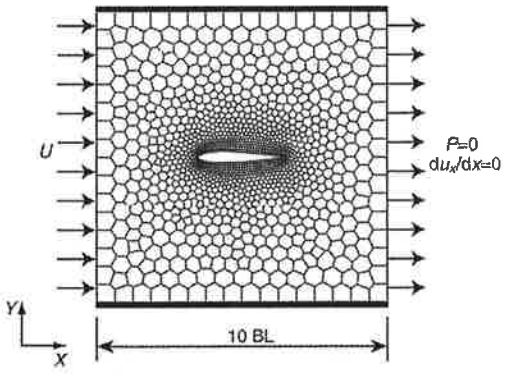


Setting Boundary Condition Data

- Explicitly assign data in BC panels.
  - To set boundary conditions for particular zone:
    - Select Boundary Conditions in the project tree.
    - Choose the boundary name in the Zone list.
    - Click the Edit... button.
  - Boundary condition data can be copied from one zone to another.

- Boundary conditions can also be defined by UDFs and profiles.

- Profiles can be generated by:
  - Writing a profile from another CFD simulation
  - Creating an appropriately formatted text file with boundary condition data.



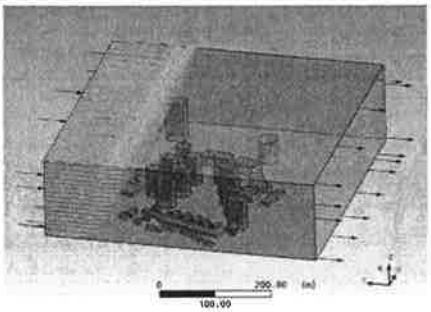
Velocity Inlet

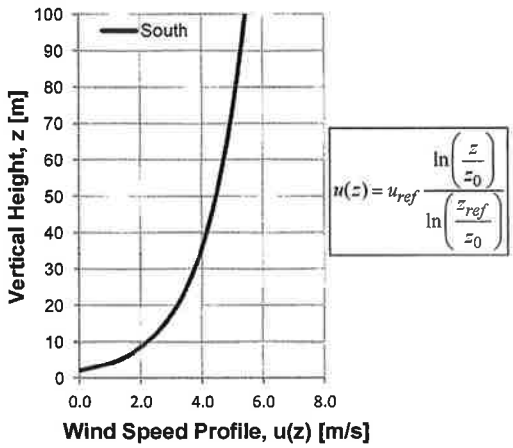
- Velocity Specification Method
  - Magnitude, Normal to Boundary
  - Components
  - Magnitude and Direction

- Applies a uniform velocity profile at the boundary, unless UDF or profile is used.

- Velocity inlets are intended for use in incompressible flows and are not recommended for compressible flows.

- Velocity Magnitude input can be negative, implying that you can prescribe the exit velocity.





Pressure Inlet

- Pressure inlets are suitable for both compressible and incompressible flows.
  - Pressure inlet boundary is treated as a loss-free transition from stagnation to inlet conditions.
  - FLUENT calculates static pressure and velocity at inlet
  - Mass flux through boundary varies depending on the interior solution and specified flow direction.

- Required inputs
  - Gauge Total Pressure
  - Supersonic / Initial Gauge Pressure
  - Inlet flow direction
  - Turbulence quantities (if applicable)
  - Total temperature (if heat transfer and/or compressible).

Incompressible:

$$p_{total} = p_{static} + \frac{\rho V^2}{2}$$

Compressible:

$$p_{total,abs} = p_{static,abs}\left(1 + \frac{k-1}{2}M^2\right)^{\frac{k}{k-1}}$$

$$T_{total,abs} = T_{static,abs}\left(1 + \frac{k-1}{2}M^2\right)$$

Mass Flow Inlet

- Mass flow inlets are intended for compressible flows; however, they can be used for incompressible flows.
  - Total pressure adjusts to accommodate mass flow inputs.
  - More difficult to converge than pressure inlet.

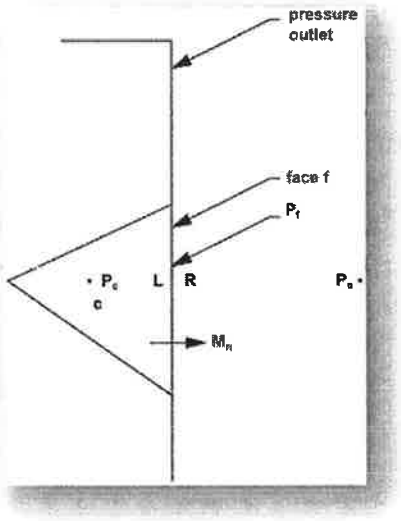
- Required information
  - Mass Flow Rate [kg/s] or Mass Flux
  - Supersonic/Initial Gauge Pressure
    - Static pressure where flow is locally supersonic; ignored if subsonic
    - Will be used if flow field is initialized from this boundary.
  - Total Temperature (on Thermal tab)
    - Used as static temperature for incompressible flow.
  - Direction Specification Method

Pressure Outlet

- Suitable for compressible and incompressible flows.
  - Specified pressure is ignored if flow is locally supersonic at the outlet.
  - Can be used as a "free" boundary in an external or unconfined flow.

- Required information
  - Gauge Pressure (static) – static pressure of the environment into which the flow exits.
  - Backflow quantities – Used as inlet conditions if/when backflow occurs (outlet acts like an inlet).

- For ideal gas (compressible) flow, non-reflecting outlet boundary conditions (NRBC) are available.



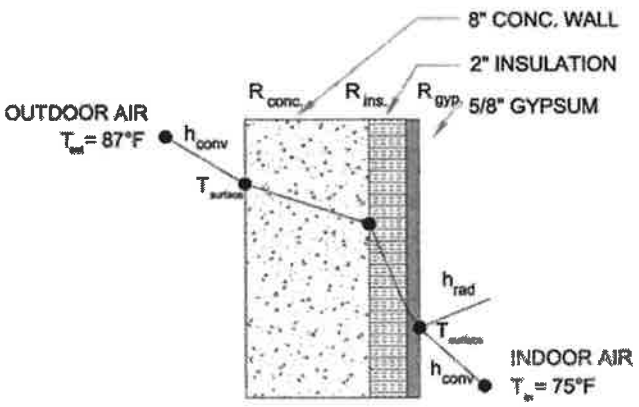
Wall Boundaries

- In viscous flows, no-slip conditions are applied at walls.
  - Shear stress can be applied.

- Thermal boundary conditions
  - Several types of thermal BCs are available.
  - Wall material and thickness can be defined for 1D or shell conduction heat transfer calculations (details will be discussed in the Heat Transfer lecture).

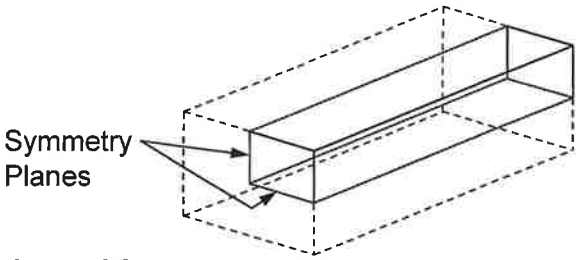
- Wall roughness can be defined for turbulent flows.
  - Wall shear stress and heat transfer based on local flow field.

- Translational or rotational velocity can be assigned to wall boundaries.

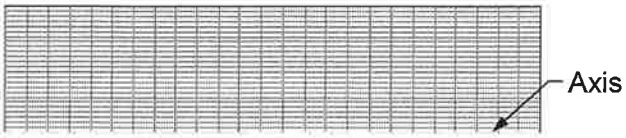


Symmetry and Axis Boundaries

- **Symmetry Boundary**
  - **No inputs are required.**
  - **Flow field and geometry must be symmetric:**
    - **Zero normal velocity at symmetry plane**
    - **Zero normal gradients of all variables at symmetry plane**
    - **Must take care to correctly define symmetry boundary locations.**



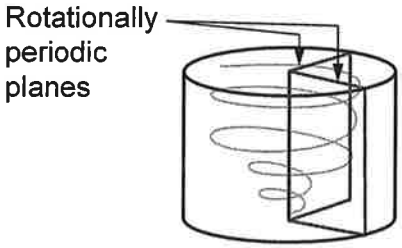
- **Axis Boundary**
  - **Used at the center line for axisymmetric problems.**
  - **No user inputs required.**
  - **Must coincide with the positive x direction!**

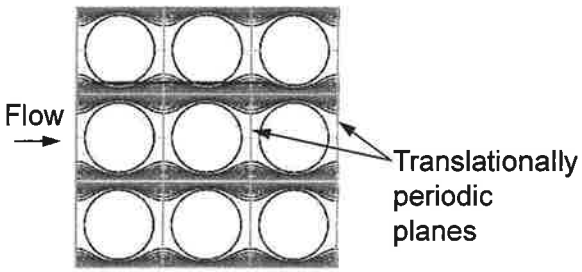


Periodic Boundaries

- **Used to reduce the overall mesh size.**

- **Flow field and geometry must contain either rotational or translational periodicity.**
  - **Rotational periodicity**
    - **ΔP = 0 across periodic planes.**
    - **Axis of rotation must be defined in fluid zone.**
  - **Translational periodicity**
    - **ΔP can be finite across periodic planes.**
    - **Models fully developed conditions.**
    - **Specify either mean ΔP per period or net mass flow rate.**





2D Tube Heat Exchanger

Internal Face Boundaries

- Defined on the cell faces only:
  - Thickness of these internal faces is zero
  - These internal faces provide means of introducing step changes in flow properties.
- Used to implement various physical models including:
  - Fans
  - Radiators
  - Porous-jump models
    - Preferable over porous media for its better convergence behavior.
  - Interior walls

Summary

- Boundary zones are used to control the solution at external and internal boundaries. Many different boundary types exist for prescribing boundary information.
- Cell zones (or continuum zones) are used to assign which fluid/solid material(s) exist in a region.
  - Options for porous media, laminar region, fixed value, etc.
- Computational effort can be reduced through use of symmetry and periodic boundaries.
- There are several other boundary condition types which were not presented.
  - Pressure Far Field
  - Exhaust Fan / Outlet Vent
  - Inlet Vent / Intake Fan
  - Outflow

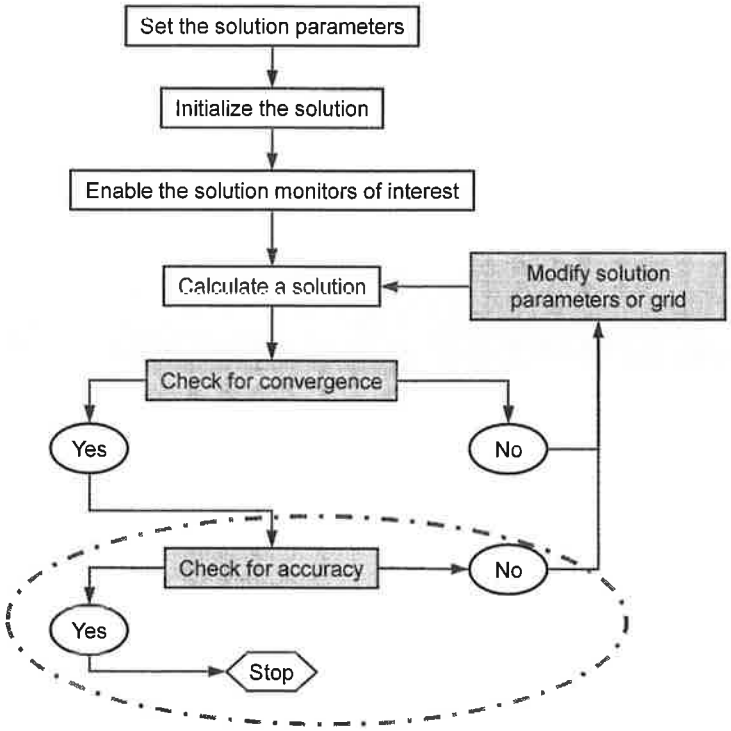
Numerical Settings

Outline

- Using the Solver (solution procedure overview)
  - Setting Solver Parameters
  - Convergence
    - Definition
    - Monitoring
    - Stability
    - Accelerating Convergence
  - Accuracy
    - Grid Independence
    - Grid Adaption
  - Unsteady Flow Modeling (covered in a later lecture)
    - Unsteady-flow problem setup
    - Unsteady flow modeling options
  - Summary
  - Appendix

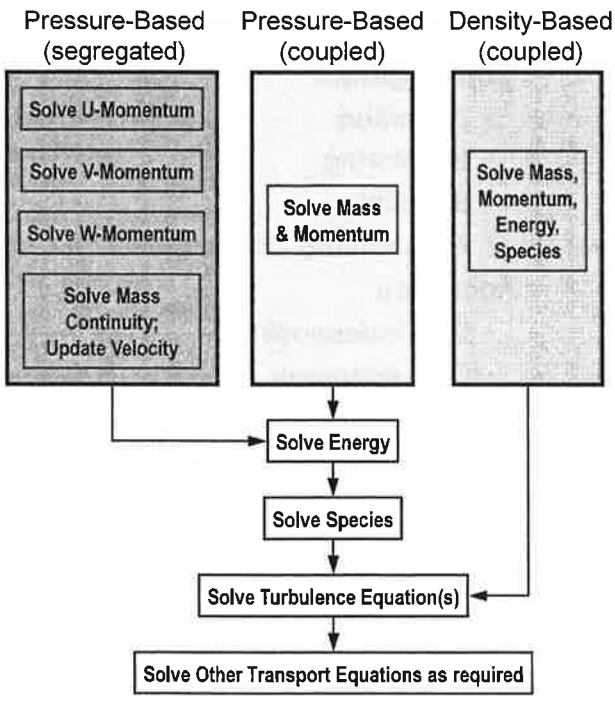
Solution Procedure Overview

- **Solution parameters**
  - **Choosing the solver**
  - **Discretization schemes**
- **Initialization**
- **Convergence**
  - **Monitoring convergence**
  - **Stability**
    - **Setting Under-relaxation**
    - **Setting Courant number**
  - **Accelerating convergence**
- **Accuracy**
  - **Grid Independence**
  - **Adaption**



Available Solvers

- **There are two kinds of conventional solvers – Pressure based and Density based.**
- **The pressure-based solvers take momentum and pressure (or pressure correction) as the primary variables.**
  - **Pressure-velocity coupling algorithms are derived by reformatting the continuity equation**
- **Two algorithms are available with the pressure-based solvers:**
  - **Segregated solver – Solves for pressure correction and momentum sequentially.**
  - **Coupled Solver (PBCS) – Solves pressure and momentum simultaneously.**



Available Solvers

- **Density-Based Coupled Solver**
  - Equations for continuity, momentum, energy and species (if required) are solved in vector form.
  - Pressure is obtained through an equation of state.
  - Additional scalar equations are solved in a segregated fashion.

- **The DBCS can be run either explicit or implicit.**
  - Implicit – Uses a point-implicit Gauss-Seidel / symmetric block Gauss-Seidel / ILU method to solve for variables.
  - Explicit: uses a multi-step Runge-Kutta explicit time integration method

Choosing a Solver

- The pressure-based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high-speed compressible flow.
  - Requires less memory (storage).
  - Allows flexibility in the solution procedure.
- The pressure-based coupled solver (PBCS) is applicable for most single phase flows, and yields superior performance to the standard pressure-based solver.
  - Not available for multiphase (Eulerian), periodic mass-flow and NITA cases.
  - Requires 1.5–2 times more memory than the segregated solver.
- The density-based coupled solver (DBCS) is applicable when there is a strong coupling, or interdependence, between density, energy, momentum, and/or species.
  - Examples: High speed compressible flow with combustion, hypersonic flows, shock interactions.
  - The implicit option is generally preferred over explicit since it has a very strict limit on time step size
  - The explicit approach is used for cases where the characteristic time scale of the flow is on the same order as the acoustic time scale. (e.g.: propagation of high-Ma shock waves).

Discretization (Interpolation Methods)

- Field variables (stored at cell centers) must be interpolated to the faces of the control volumes.

$$V_{\text{cell}}\frac{\partial(\rho\phi)}{\partial t}+\boxed{\sum_{f}^{N_{\text{faces}}}\rho_f\mathbf{V}_f\phi_f\cdot\mathbf{A}_f}=\sum_{f}^{N_{\text{faces}}}\Gamma_\phi\nabla\phi_f\cdot\mathbf{A}_f+S_\phi V_{\text{cell}}$$

- Interpolation schemes for the convection term:
  - First-Order Upwind – Easiest to converge, only first-order accurate.
  - Power Law – More accurate than first-order for flows when $Re_{cell} < 5$ (typ. low Re flows)
  - Second-Order Upwind – Uses larger stencils for 2nd order accuracy, essential with tri/tet mesh or when flow is not aligned with grid; convergence may be slower.
  - Monotone Upstream-Centered Schemes for Conservation Laws (MUSCL) – Locally 3rd order convection discretization scheme for unstructured meshes; more accurate in predicting secondary flows, vortices, forces, etc.
  - Quadratic Upwind Interpolation (QUICK) – Applies to quad/hex and hybrid meshes, useful for rotating/swirling flows, 3rd-order accurate on uniform mesh.

Interpolation Methods (Gradients)

- Gradients of solution variables are required in order to evaluate diffusive fluxes, velocity derivatives, and for higher-order discretization schemes.

$$V_{\text{cell}}\frac{\partial(\rho\phi)}{\partial t}+\sum_{f}^{N_{\text{faces}}}\rho_f\mathbf{V}_f\phi_f\cdot\mathbf{A}_f=\boxed{\sum_{f}^{N_{\text{faces}}}\Gamma_\phi\nabla\phi_f\cdot\mathbf{A}_f}+S_\phi V_{\text{cell}}$$

- The gradients of solution variables at cell centers can be determined using three approaches:
  - Green-Gauss Cell-Based – The default method; solution may have false diffusion (smearing of the solution fields).
  - Green-Gauss Node-Based – More accurate; minimizes false diffusion; recommended for tri/tet meshes.
  - Least-Squares Cell-Based – Recommended for polyhedral meshes; has the same accuracy and properties as Node-based Gradients.
- Gradients of solution variables at faces computed using multi-dimensional Taylor series expansion.

Interpolation Methods for Pressure

- Interpolation schemes for calculating cell-face pressures when using the segregated solver in FLUENT are available as follows:
  - Standard – The default scheme; reduced accuracy for flows exhibiting large surface-normal pressure gradients near boundaries (but should not be used when steep pressure changes are present in the flow – PRESTO! scheme should be used instead.)
  - PRESTO! – Use for highly swirling flows, flows involving steep pressure gradients (porous media, fan model, etc.), or in strongly curved domains
  - Linear – Use when other options result in convergence difficulties or unphysical behavior
  - Second-Order – Use for compressible flows; not to be used with porous media, jump, fans, etc. or VOF/Mixture multiphase models
  - Body Force Weighted – Use when body forces are large, e.g., high Ra natural convection or highly swirling flows

Pressure-Velocity Coupling

- Pressure-velocity coupling refers to the numerical algorithm which uses a combination of continuity and momentum equations to derive an equation for pressure (or pressure correction) when using the pressure-based solver.

- Four traditional pressure-velocity coupling algorithms
  - Semi-Implicit Method for Pressure-Linked Equations (SIMPLE)
    - The default scheme, robust
  - SIMPLE-Consistent (SIMPLEC)
    - Allows faster convergence for simple problems (e.g., laminar flows with no physical models employed).
  - Pressure-Implicit with Splitting of Operators (PISO)
    - Useful for unsteady flow problems or for meshes containing cells with higher than average skewness
  - Fractional Step Method (FSM) for unsteady flows.
    - Used with the NITA scheme; similar characteristics as PISO.

Initialization

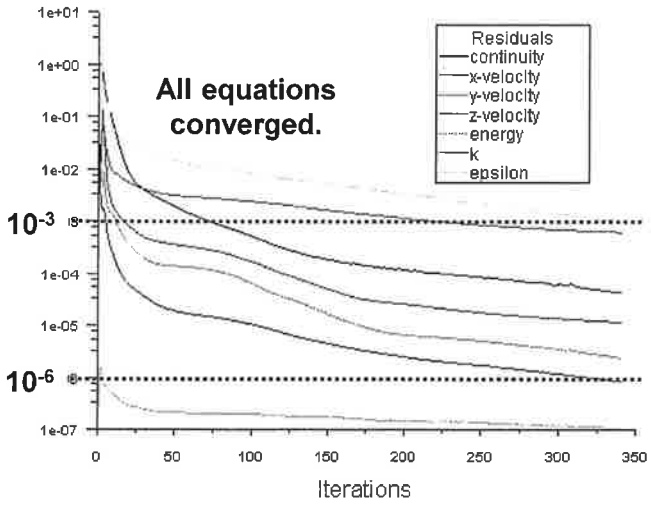
- All solution variables need to be initialized.
  - Realistic guesses improves solution stability and accelerates convergence.
  - In some cases, a good initial guess is required.
- Patch values for individual variables in certain regions.
  - Free jet flows (high velocity for jet)
  - Combustion problems (high temperature region to initialize reaction)
  - Cell registers (created by marking the cells in the Adaption panel) can be used for patching values into various regions of the domain.

Convergence

- At convergence, the following should be satisfied:
  - All discrete conservation equations (momentum, energy, etc.) are obeyed in all cells to a specified tolerance OR the solution no longer changes with subsequent iterations.
  - Overall mass, momentum, energy, and scalar balances are achieved.
- Monitoring convergence using residual history:
  - Generally, a decrease in residuals by three orders of magnitude indicates at least qualitative convergence. At this point, the major flow features should be established.
  - Scaled energy residual should decrease to 10^{-6} (for the pressure-based solver).
  - Scaled species residual may need to decrease to 10^{-5} to achieve species balance.
- Monitoring quantitative convergence of other relevant key variables/physical quantities for a confirmation.
- Ensure that the overall mass/heat/species conservation is satisfied.

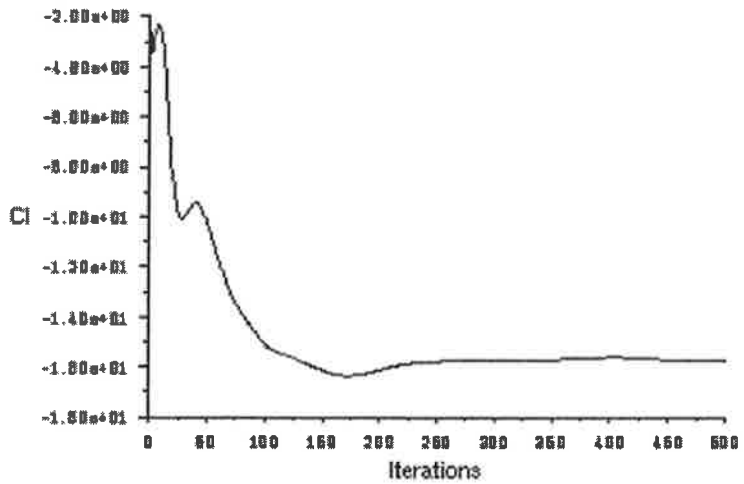
Convergence Monitors – Residuals

- **Residual plots show when the residual values have reached the specified tolerance.**



Convergence Monitors – Forces and Surfaces

- **In addition to residuals, you can also monitor lift, drag and moment coefficients.**
- **Relevant variables or functions (e.g. surface integrals) at a boundary or any defined surface.**



Checking Overall Flux Conservation

- In addition to monitoring residual and variable histories, you should also check for overall heat and mass balances.
- The net flux imbalance should be less than 1% of the smallest flux through the domain boundary

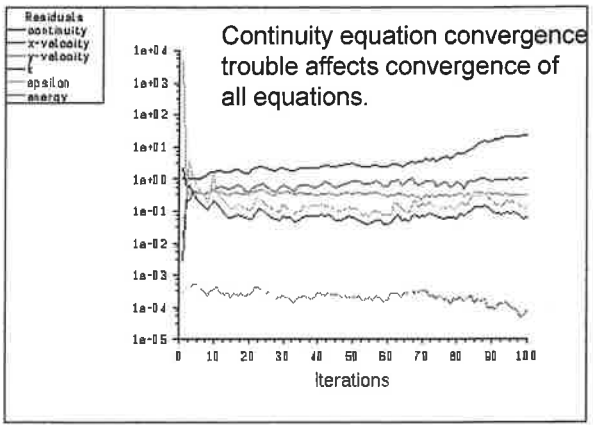
Tightening the Convergence Tolerance

- If solution monitors indicate that the solution is converged, but the solution is still changing or has a large mass/heat imbalance, this clearly indicates the solution is not yet converged.
- In this case, you need to:
  - Reduce values of Convergence Criterion or disable Check Convergence in the Residual Monitors panel.
  - Continue iterations until the solution converges.
- Selecting None under Convergence Criterion disables convergence checking for all equations.

Convergence Difficulties

- Numerical instabilities can arise with an ill-posed problem, poor-quality mesh and/or inappropriate solver settings.
  - Exhibited as increasing (diverging) or "stuck" residuals.
  - Diverging residuals imply increasing imbalance in conservation equations.
  - Unconverged results are very misleading!

- Troubleshooting
  - Ensure that the problem is well-posed.
  - Compute an initial solution using a first-order discretization scheme.
  - For the pressure-based solver, decrease underrelaxation factors for equations having convergence problems.
  - For the density-based solver, reduce the Courant number.
  - Remesh or refine cells which have large aspect ratio or large skewness.
    - Remember that you cannot improve cell skewness by using mesh adaption!



Modifying Under-Relaxation Factors

- Under-relaxation factor, α, is included to stabilize the iterative process for the pressure-based solver
- Use default under-relaxation factors to start a calculation.

$$\phi_P = \phi_{P,\text{old}} + \alpha \Delta \phi_P$$

- Decreasing under-relaxation for momentum often aids convergence.
  - Default settings are suitable for a wide range of problems, you can reduce the values when necessary.
  - Appropriate settings are best learned from experience!
- For the density-based solver, under-relaxation factors for equations outside the coupled set are modified as in the pressure-based solver.

Modifying the Courant Number

- A transient term is included in the density-based solver even for steady state problems.
  - The Courant number defines the time step size.
- For density-based explicit solver:
  - Stability constraints impose a maximum limit on the Courant number.
    - Cannot be greater than 2 (default value is 1).
    - Reduce the Courant number when having difficulty converging.
- For density-based implicit solver:
  - The Courant number is not limited by stability constraints.
    - Default value is 5.

Accelerating Convergence

- Convergence can be accelerated by:
  - Supplying better initial conditions
    - Starting from a previous solution (using file/interpolation when necessary)
  - Gradually increasing under-relaxation factors or Courant number
    - Excessively high values can lead to solution instability convergence problems
    - You should always save case and data files before continuing iterations
  - Controlling MultiGrid solver settings (not generally recommended)
    - Default settings provide a robust Multigrid setup and typically do not need to be changed.

Starting from a Previous Solution

- A previously calculated solution can be used as an initial condition when changes are made to the case setup.
  - Use solution interpolation to initialize a run (especially useful for starting fine-mesh cases when coarse-mesh solutions are available).
  - Once the solution is initialized, additional iterations always use the current data set as the starting point.
  - Some suggestions on how to provide initial conditions for some actual problems:

Actual Problem	Initial Condition
Heat Transfer	Isothermal
Natural convection	Low Rayleigh number
Combustion / reacting flow	Cold flow (no combustion)
Turbulence	Inviscid (Euler) solution

Solution Accuracy

- A converged solution is not necessarily a correct one!
  - Always inspect and evaluate the solution by using available data, physical principles and so on.
  - Use the second-order upwind discretization scheme for final results.
  - Ensure that solution is grid-independent:
    - Use adaption to modify the grid or create additional meshes for the grid-independence study
- If flow features do not seem reasonable:
  - Reconsider physical models and boundary conditions
  - Examine mesh quality and possibly remesh the problem
  - Reconsider the choice of the boundaries' location (or the domain): inadequate choice of domain (especially the outlet boundary) can significantly impact solution accuracy

Mesh Quality and Solution Accuracy

- Numerical errors are associated with calculation of cell gradients and cell face interpolations.

- Ways to contain the numerical errors:
  - Use higher-order discretization schemes (second-order upwind, MUSCL)
  - Attempt to align grid with the flow to minimize the "false diffusion"
  - Refine the mesh
    - Sufficient mesh density is necessary to resolve salient features of flow
      - Interpolation errors decrease with decreasing cell size
    - Minimize variations in cell size in non-uniform meshes
      - Truncation error is minimized in a uniform mesh
      - Adapt mesh based on cell size variation
    - Minimize cell skewness and aspect ratio
      - In general, avoid aspect ratios higher than 5:1 (but higher ratios are allowed in boundary layers)
      - Optimal quad/hex cells have bounded angles of 90 degrees
      - Optimal tri/tet cells are equilateral

Grid-Independent Solutions

- A grid-independent solution exists when the solution does not change when the mesh is refined.
- Below is a systematic procedure for obtaining a grid-independent solution:
  1. Generate a new, finer mesh.
     - Return to the meshing application and manually adjust the mesh.
     - OR Use the solution-based adaption capability.
       - VERY IMPORTANT: Save the case and data files first.
       - Create adaption register(s) and adapt the mesh. Data from the original mesh is interpolated onto the finer mesh. Dynamic mesh adaption which automatically changes the mesh according to user-defined criteria.
  2. Continue calculations until convergence.
  3. Compare the results obtained on the different meshes.
  4. Repeat the procedure if necessary.

Summary

- Solution procedure for both the pressure-based and density-based solvers is identical.
  - Calculate until you get a converged solution
  - Obtain a second-order solution (recommended)
  - Refine the mesh and recalculate until a grid-independent solution is obtained.
- All solvers provide tools for judging and improving convergence and ensuring stability.
- All solvers provide tools for checking and improving accuracy.
- Solution accuracy will depend on the appropriateness of the physical models that you choose and the boundary conditions that you specify.

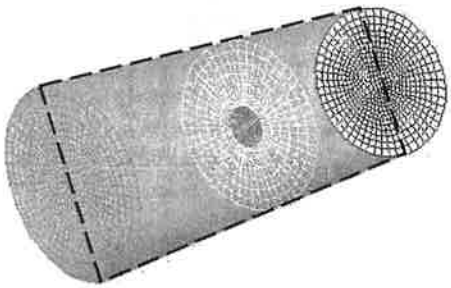
Post processing of simulation results

Overview

- Conventional tools for analyzing CFD results
  - Isosurfaces
  - Vector Plots
  - Contour plots (shaded and graded)
  - Streamlines and pathlines
  - XY plotting
  - Animation creation
- User-defined quantity either defined through custom field functions or user-defined memory (UDF).

Surface Creation

- There are a variety of ways to create surfaces:
  - Zone surfaces (surfaces automatically created by solver from zones)
  - Plane surfaces (specifying a specific plane in the domain)
  - Iso-surfaces (surfaces that have constant value for a specified variable)
  - Clipping Surfaces (iso-surfaces trimmed within specified range of values)
  - Point surfaces (specifying a particular location in the domain)
  - Line and Rake Surfaces (used for display of particle path lines)
- Surfaces can be renamed, deleted or moved and used to write profile files.

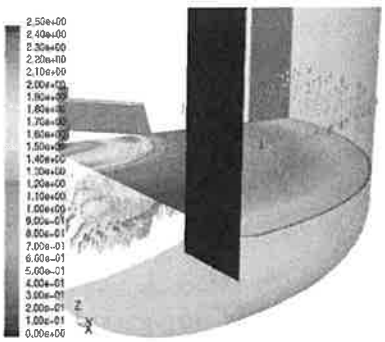


Iso-Surface of a constant grid coordinate can be created for viewing data within a plane.

Post-Processing Rendering Options

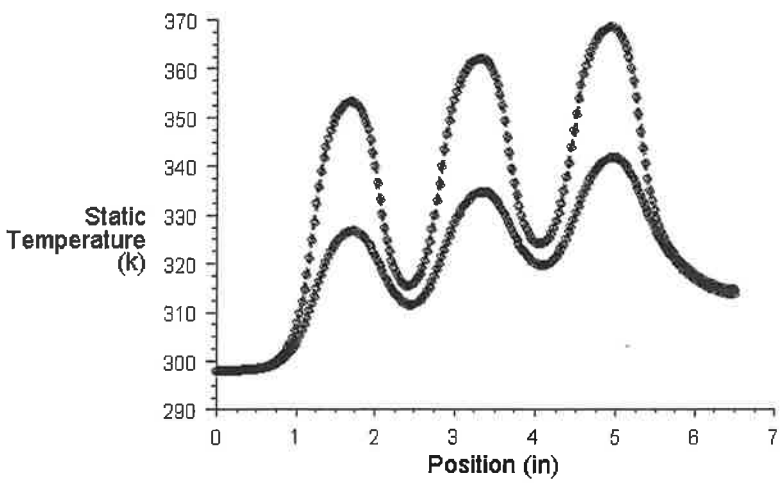
- The rendering options to control of the look-and-feel of the post-processing plots, including:
  - Views and display options
  - Colormaps for contour/vector plots
  - Shading on surfaces using Lights
  - Annotation of plots
  - Surface manipulation
  - Scene Composition using plot overlays, different colors, shading, transparency
  - Scene animation (fly throughs)

Example of scene composition: Overlay of contour and vector plot with transparent walls to show internal details.



Plots

- Tools to generate data plots of the solution:
  - XY plots of solution variables
  - Histograms to illustrate frequency of distribution
  - Fast Fourier Transforms (FFT)
  - Residuals
- You can modify the colors, titles, legend, axis and curve attributes to customize your plots.
- Other data files (experimental, computational) can also be read in to compare results.



Reports

- **Flux reports**
  - **Net flux is calculated.**
  - **Total Heat Transfer Rate includes radiation.**
- **Surface integrals**
  - **slightly less accurate on user-generated surfaces due to interpolation error.**
- **Volume integrals**

$$\int \phi \, dA = \sum_{i=1}^{N} \phi_i |A_i|$$

$$\frac{1}{A} \int \rho \phi \mathbf{v} \cdot d\mathbf{A} = \frac{1}{A} \sum_{i=1}^{N} \rho_i \phi_i \mathbf{v}_i \cdot d\mathbf{A}_i$$

$$\frac{1}{A} \int \phi \, dA = \frac{1}{A} \sum_{i=1}^{N} \phi_i |A_i|$$

Quality Control

Quality Control

Quality control consists of these major steps:
- recognize possible sources of errors
- check for these errors in your own simulations
- estimate the accuracy of the simulations
- improve the simulations, if possible

Main items in this chapter are:
- Steps in a CFD simulation
- Sources of errors and uncertainties
- How to ensure high quality predictions (recommendations)
- Questions to ask the CFD engineer as regards the work reported
- Additional advice and remarks
- A short check list

Quality Controls

- Steps of CFD simulation

CFD Simulation Steps	Group
Define the Problem	1. Problem Definition
Define the Geometry	
Generate Computational Grid	2. Grid
Choose Physical Modes	3. Models
Choose Turbulence Model	
Define Boundary Conditions	4. B.C.
Initial Conditions	5. Numerical
Select Solution Strategy	
Select Numerical Procedure	
Solve the Equations	6. Code
Check the Solutions	7. User
Post-Processing	
Analysis & Interpretation	
Documentation	

Errors

Uncertainties

Quality Controls

- Uncertainties/Errors in CFD simulation
  - Uncertainty
    - Potential deficiency due to lack of common knowledge (e.g. turbulence model) or unknown boundary conditions
    - Improvements are difficult to achieve
  - Error
    - Can be quantified and/or avoided with sufficient care (e,g, numerical or user error)
  - Verification
    - Procedure to ensure equations are solved correctly
  - Validation
    - Procedure to test how model represent reality
  - Calibration
    - Procedure to assess CFD code to predict global quantities of interest for specific geometry of engineering design interest

Quality Controls

- Ensure quality

1. Problem definition
   - What to obtain from predictions?
   - Which are the steps on the way to final answer?
   - How problems are defined for simulation program?
     1. Simulation Method/Approach
        - 2D vs. 3D
        - Steady vs. transient
        - Single fluid vs. multi-component fluid
     2. Simplification, modeling level
        - Geometrical simplification
        - Volumetric heat source w/o flow obstruction vs. heat flux BC
        - Square inlet geometry vs. real round one
     3. Unsuitability of problem
        - Room with many opening and free flow conditions

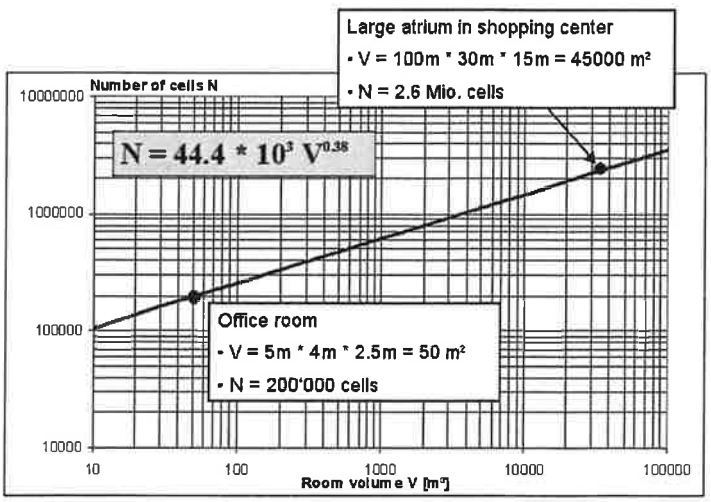
Quality Controls

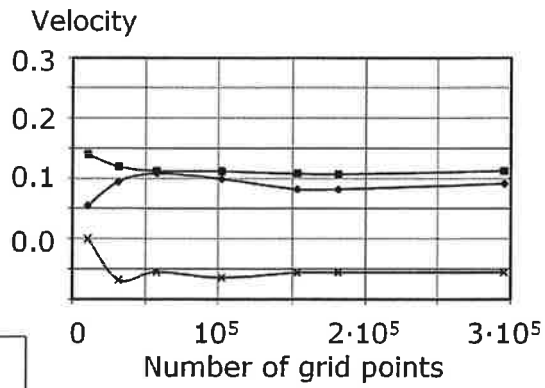
- Ensure quality
  2. Computational Grid
     - Number of cells
       - Grid resolution usually cannot be too high
       - Typical cell numbers for two equation turbulence modeling with standard wall functions (German guideline VDI6019)

$$N = 44.4x10^3 V^{0.38}$$

     - Cell distribution
       - High resolution in areas of relevant flow phenomena, e.g. boundary layer, plume above heat sources etc
       - Boundary layer
         - First cell must lie in logarithmic region; otherwise error will be introduced due to a too high resolution
         - Force convection – standard wall function
         - Free convection – scalable wall function (Menter, 1994) – value of first grid cell ($y^+ > 8$) is removed and sub-layer is treated analytically

Cell Quality





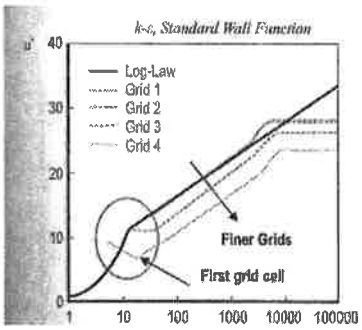
Monitoring velocities versus number of cells in the prediction

Quality Controls

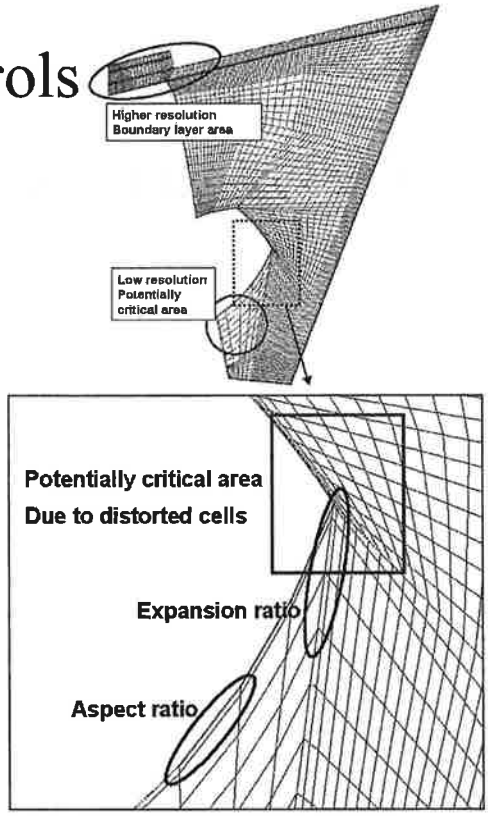
- Ensure quality
  2. Computational Grid
     - Cell quality
       - Distorted, squeezed or degenerated cells introduce numerical error
       - Criteria: angles, determinants, aspect ratio, expansion ratio etc
         - Angles between 20° and 160°
         - Expansion ratio < 2
         - Aspect ratio < 10 (hexahedral cells)
         - 5 to 10 grid points within important flow phenomena (boundary layer, boundary sources, inlet)
         - 10cm grid distance in normal room (size 5m) and 30cm in larger room (size 20m); finer in important flow phenomena
       - Hexahedral cell introduce less numerical error as compared to tetrahedral cell; but grid sensitivity also depend on discretization scheme
       - Grid independence study is required

Quality Controls

- Ensure quality
  2. Computational Grid



Errors on too fine grids with log-law wall function



Grid areas with different resolution and cell shapes in structured multi-block grid

Quality Controls

- Ensure quality
  3. Modeling
     - Turbulence model
       - SST model (blend between κ-ε and κ-ω) to overcome deficiencies of κ-ε model
       - κ-ε model: ceiling jet becomes too thick vs. RSM model: ceiling jet remains thin due to strong damping of turbulence fluctuation perpendicular to ceiling (experimentally observed)
     - Buoyancy model
       - Boussinesq approximation for incompressible flow with temperature difference < 50°C
       - Ideal gas law for ΔT> 50°C or gas mixture with significant density difference (e.g. CO_2 with high concentration, 0.5% as exhaust gas
     - Radiation model
       - For certain results, it is a must to consider radiation between surfaces in room air flow in conjunction with heat flux BC
       - Neglecting radiation in displacement ventilation leads to wrong temperature distribution
       - Gas absorption of infrared radiation is important in displacement ventilation

Quality Controls

- Ensure quality
  3. Modeling
     - Aerosols (e.g. water droplets)
     - Two phase flow (phase change between water vapor/liquid)
     - Chemical reaction
     - Checklist
       - Turbulence model: two equation model with adequate wall treatment
       - Buoyancy model: Boussinesq approximation
       - Gravity direction
       - Radiation model: used if necessary

Quality Controls

- Ensure quality
  4. Boundary conditions
     - Major source of errors/uncertainties
       - Thermal BCs are not known
       - Too small computational domain for inlet/outlet
       - Exhaust opening with prescribed velocity rather than pressure
       - Open window with free flow boundary
       - Several exhaust openings at different height levels with pressure boundaries
       - Incorrect assumption of symmetry plane
       - Wind influence without boundary layer profile at inlet
     - Checklist
       - Check position and values of inlets and outlets
       - Temperature BCs: difficult to justify, best values can be obtained from thermal simulations
       - Justified symmetry plane (if any)
       - Free flow BCs for window: difficult, predictions can be totally wrong
       - Volumetric heat sources without blocking or with simplified porosity may result in unrealistic flows

Quality Controls

- Ensure quality
  5. Numerical errors
     - Discretization scheme
       - Determine how convection, diffusion, unsteady and source terms calculated using values available from previous iteration step
       - High accuracy exhibit low stability
       - Central difference (2nd order), Upwind ($1^{st}/2^{nd}$ order), QUICK ($2^{nd}/3^{rd}$ order)
       - For fine grid, errors decrease with higher order
       - Refer to Smith and Hutton flow problem for performance of various discretization scheme
     - Iteration procedure and Time steps
       - Incomplete convergence is higher for temperature and scalar quantities
       - Convergence is slow, especially for SIMPLE algorithm
       - Modern code use accelerating multi-grid techniques
     - Initial conditions
       - Begin with zero velocities, except if flow direction is predominant
       - Mean values of temperature and scalar

Quality Controls

- Ensure quality

5. Numerical errors
   - Computer resolution (32-bit or 64-bit)
     - Round-off errors can be critical in low-Re flow or conjugated heat transfer problem
     - Double precision is preferable
   - Checklist
     - Grid independency
     - Use 2nd order discretization scheme
     - Use meaningful convergence criteria
     - Global balance, e.g. mass, energy and scalar balance)
     - Monitor values at important locations (e.g. large gradient)

Quality Controls

- Ensure quality

6. Code errors
   - In particular for new feature in software released
   - Check for plausibility/known balances/expected flow features
   - Search within CFD user forum for appropriate keywords, e.g. www.cfd-online.com
7. User errors
   - In particular for novice user
   - Results should be checked by experience users

Quality Controls

- Questions to CFD engineer
  - Check
    1. Plausibility
       - Mass balance between inlet and outlet
       - Energy balance of room
       - Average room temperature vs. mean temperature for energy balance (exhaust temperature)
    2. Calculation grid
       - Hexahedral grid is better than tetrahedral grid
       - Number of cells, largest and smallest cell in different areas
       - Grid resolution
         - 3m high rooms: 10cm vertical spacing, on average 30cm horizontal spacing free of sources or obstacles
         - 20m high atrium: 50cm vertical spacing, on average 30cm horizontal spacing free of sources or obstacles, finer grid near wall (5 cells within 10cm)
         - Radiation, first cell thickness ~ 5mm to yield correct relation between convection and radiation heat transfer

Quality Controls

- Questions to CFD engineer
  - Check
    3. Numerical method
       - 2nd order scheme required
       - Monitor important points (temperature and scalars)
    4. Models
       - Energy equation to include buoyancy force
       - Boussinesq approximation for $\Delta T < 50°C$
       - Radiation model if necessary
       - Turbulence model, justify κ-ε
    5. Boundary conditions
       - Locations of inlets/outlets justified?
       - BCs for supply diffusers realistic/validated?
       - Thermal BCs for heat source realistic?
       - Symmetry plane justified?
       - Volume sources for persons without obstructions justified?

Quality Controls

- Questions to CFD engineer
  - Check
    6. Simulation results
       - Upward/Downward flows plausible?
       - Boundary layer flow visible?
       - Area of high/low temperature plausible?
       - Stagnant flow plausible?
    7. Post-processing & conclusions drawn
       - Method/Formula used for comfort assessment
       - Explanation of approach to derive conclusions

Quality Controls

- Additional advices
  - Always start with simplified case and few grid cells (100K) to establish proper setting of models and boundary conditions
  - Good documentation of CFD work
  - Complete overview of all cases simulated
  - For result presentation, use same view, color distributions and value scales. Also avoid scientific notation, use 19°C instead of 291.115K
  - User has to be sensitive to critical points

Check-List Quality Controls

Topic	Subtopic	Checks, advices
1. Problem Definition	Simulation method	Choose proper approach (2D/3D, steady/transient)
	Simplification	Choose proper solution domain
	Step-by-step approach	From coarse model to fine model
2. Grid	Number of cells	Hexahedral cells are better than tetrahedral cells
	Distribution	For simple room (5m x 5m x 3m), Coarse grid, 60^3~200K
	Cell quality	Medium grid, 100^3~1M, Fine grid, 200^3~8M
3. Models	Buoyancy	Boussinesq or variable density, check gravity direction
	Turbulence model	SST model best choice of 2 equation model for indoor flow
	Wall treatment	Wall functions, low Re-number wall treatment
	Radiation	Discrete ordinate, surface-to-surface
4. B.C.	Wall temperature	Check assumptions (obtained from BEPS)
	Air supply parameters	Realistic? Validated?
	Symmetry plane	Hides asymmetric phenomena?
	Wind situation	Boundary layer profile, check domain extent (10 times building size)
5. Numerical	Discretization scheme	2nd order
6. Code		Latest version, updates, user forums
7. User		Training, experience, ask expert
8. Documentation		Full documentation of parameters, readable scale

Quality Controls

- Typical Quote for CFD Practitioner
  - Garbage in, Garbage out
  - Everybody believe in the experimental results; except the experimentalists
  - Nobody believe in the simulation results; except the simulation scientists
  - CFD is NOT Colorful Fraud Deceive
  - Treat CFD tool as Grey box; NOT Black box

Day 3 (1:30pm – 5:30pm)

- Tutorial for Practice 2 - Simulation of air flow around and across a Silsoe Cube (step by step guideline)

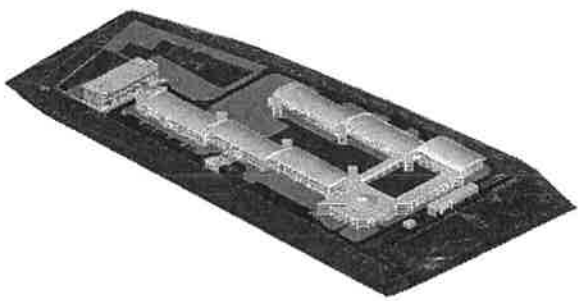
Day 4 (9:00am – 9:45pm)

- CFD model demonstration
  - Natural ventilation scenario in BCA Academy 2-Storey Training Block – Engineering Perspective

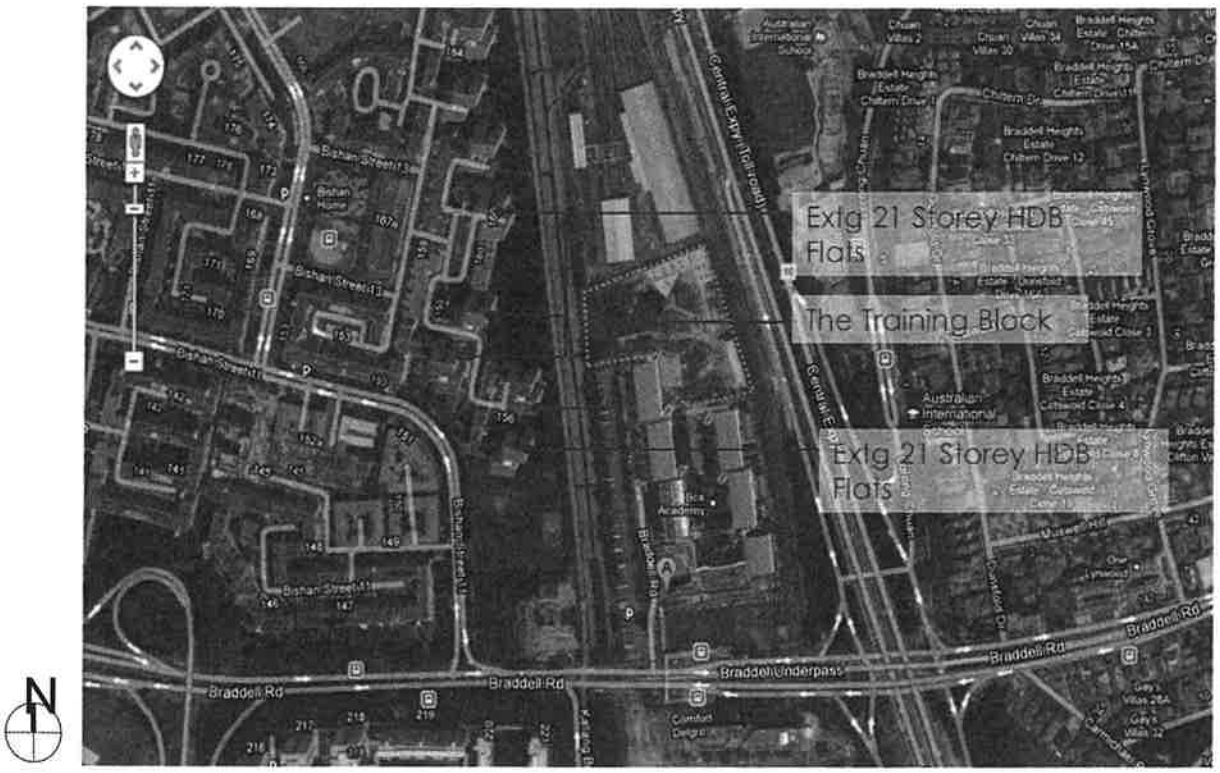
Natural ventilation scenario in BCA Academy 2-Storey Training Block – Engineering Perspective

Case Study 1: Natural Ventilation for BCAA Training Block

- This case study talk is about the BCAA Training Block as an example of a real project now undergoing GM assessment, and our experiences wrt the CFD - how the building was designed from the sustainable angle and what sort of design and construction adjustments we had to do to achieve the best wind flow
- Informal and lively discussion
- Participants are high encouraged to share their experiences/opinions

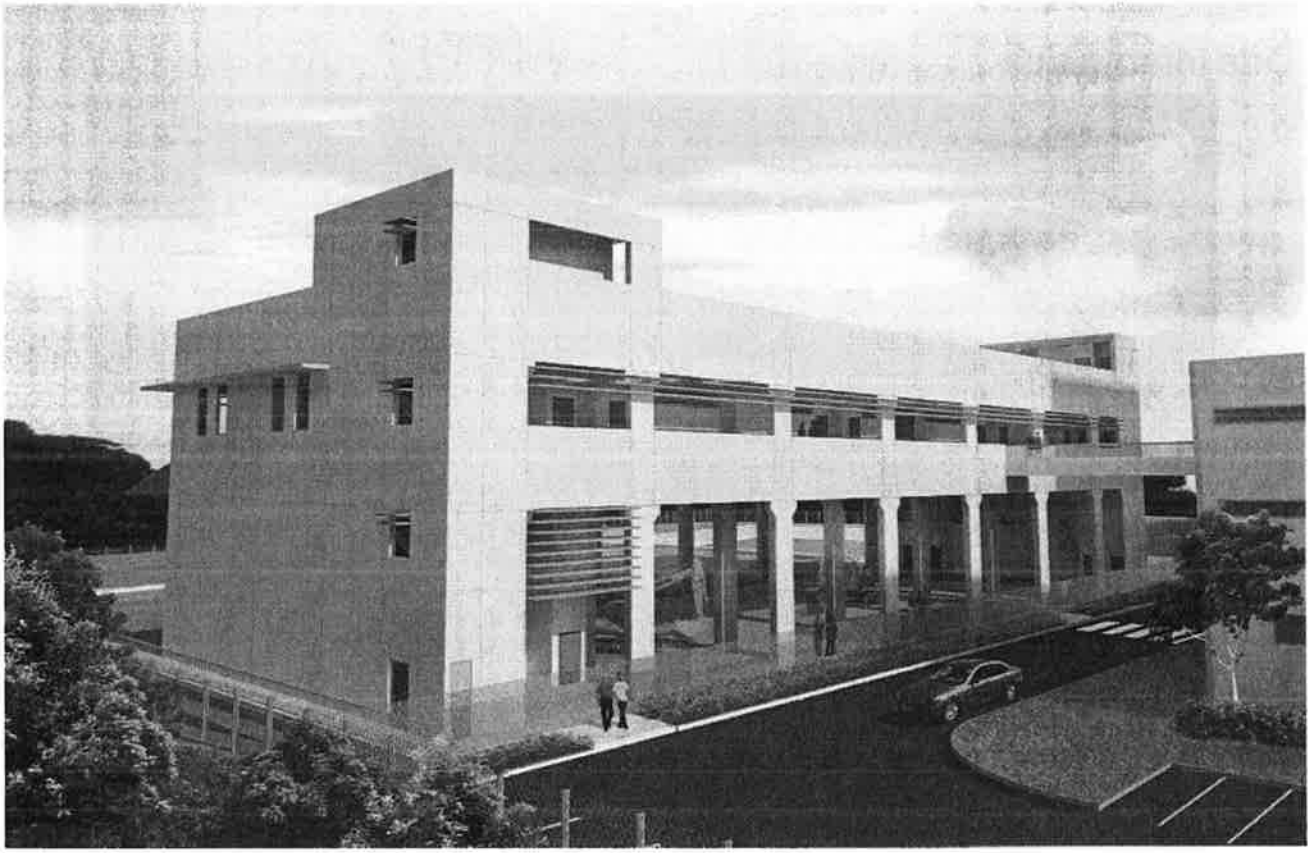


Site Information



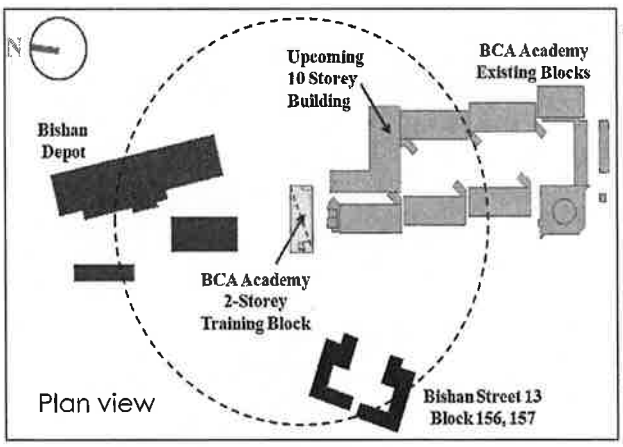
29 Feb 2012 – 2pm (View of sun onto Training Block Site)

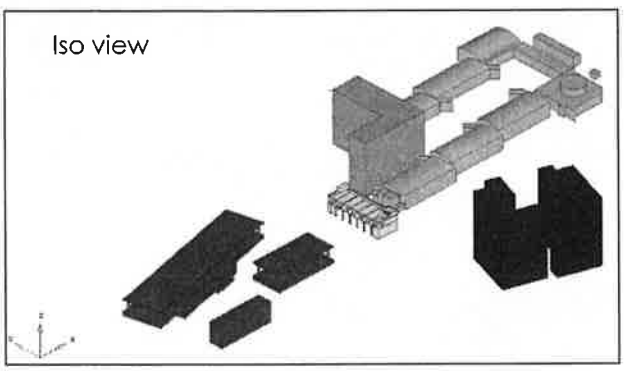




01 Energy Efficiency – Natural Ventilation

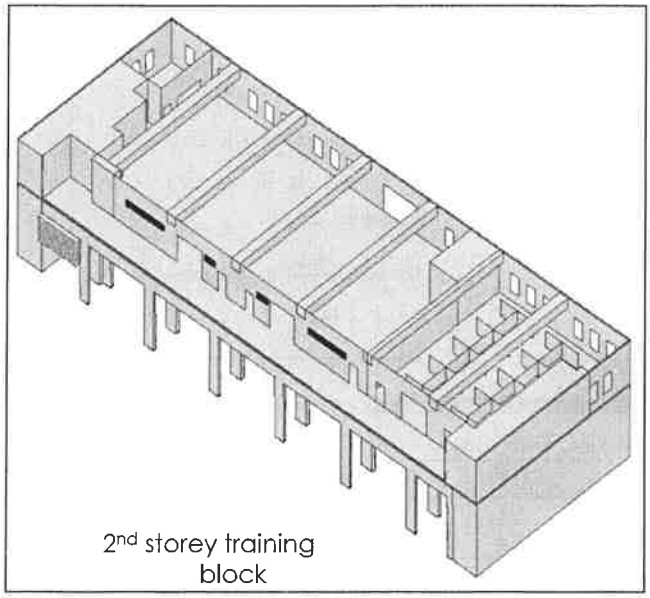
Computational Model

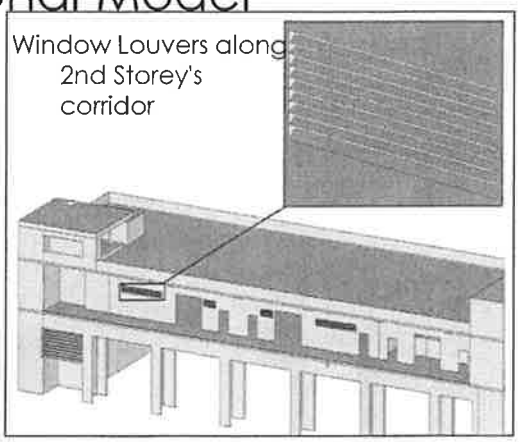


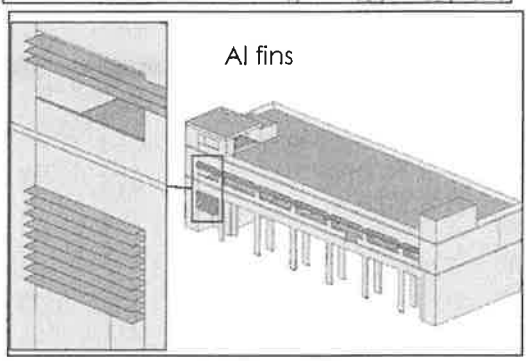


01 Energy Efficiency – Natural Ventilation

Computational Model





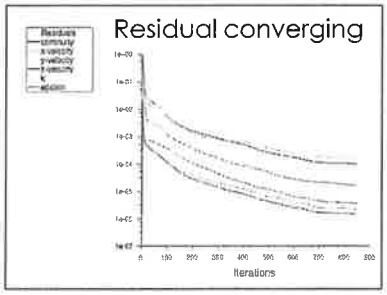


01 Energy Efficiency – Natural Ventilation

Simulation Methodology

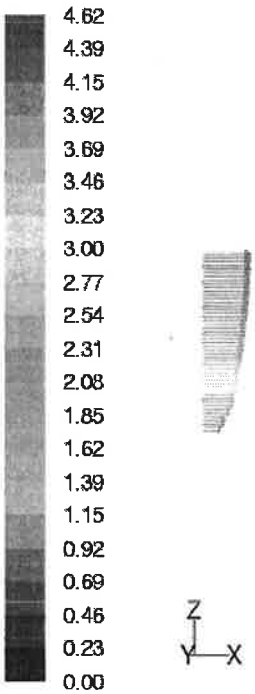
Software	FLUENT 6.3
Mesh Size	Approximately 23 million cells
Mesh Type	Unstructured tetrahedral / prisms
Turbulence Model	Standard *k-ε* turbulence model
Inlet Boundary	Logarithmic boundary layer profile
Reference Speed	North wind: 2.0m/s
Outlet Boundary	Pressure outlet
Domain Height	North wind direction: 150m
Domain Length	North wind direction: 700m
Solution Control	About 1000 iterations and monitor the average velocity at the 1.2m height in the workshop to reach steady value

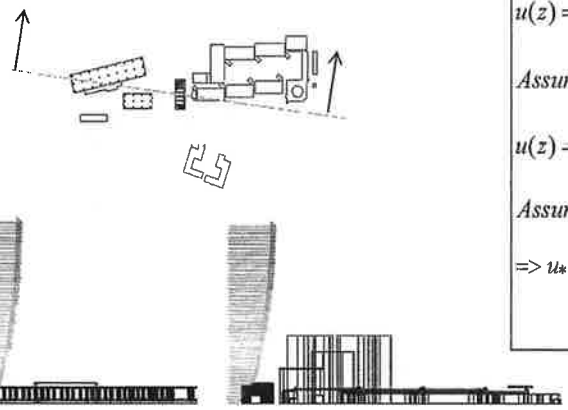
Type of Model	**Mesh Size**
Louver	0.05m
BCAA training facility unit	0.2m
Near the buildings and ground level	1.0m
Far field boundary	10.0m



01 Energy Efficiency – Natural Ventilation

Simulation Result – Base Case – Log Law wind





$$u(z) = \frac{u_*}{k}\left(\ln\left(\frac{z-d}{z_0}\right) + \psi\left(z, z_0, L\right)\right)$$

$$Assume:\ \psi\left(z, z_0, L\right) = 0$$

$$u(z) = \frac{u_*}{k}\ln\left(\frac{z-d}{z_0}\right)$$

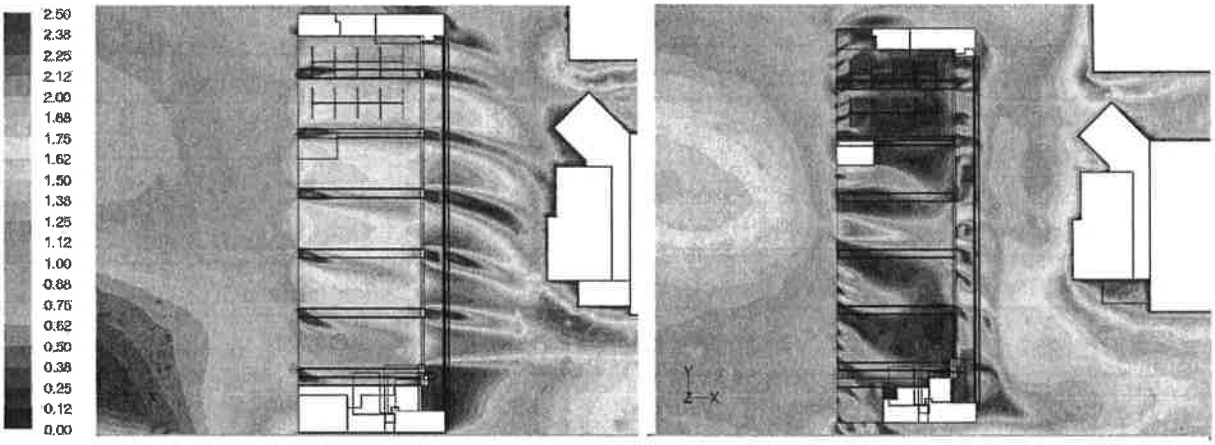
$$Assume:\ d = 0$$

$$\Rightarrow u_* = \frac{ku(z)}{\ln\left(\frac{z}{z_0}\right)} \Rightarrow u_* = \frac{kU}{\ln\left(\frac{z_{ref}}{z_0}\right)}$$

z_o [m] =	2.0	roughness length
z_{ref} [m] =	15.0	reference height of 15m
K =	0.41	von Karman's constant
d [m] =	0.0	zero plane displacement

01 Energy Efficiency – Natural Ventilation

Simulation Result – Base Case

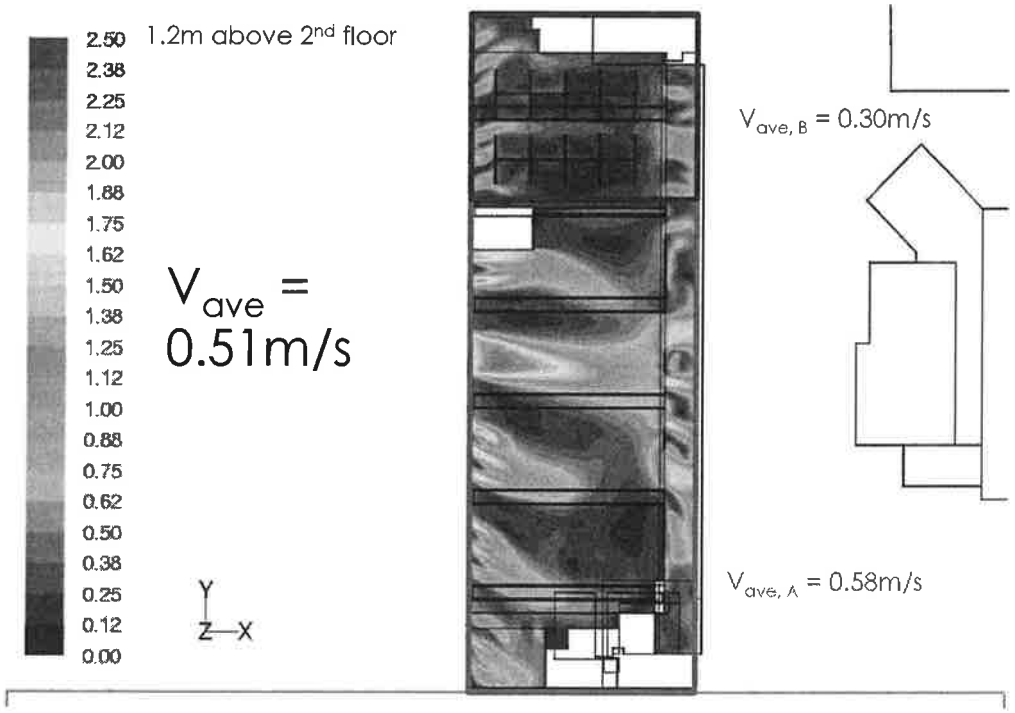


1.2m above ground floor

1.2m above 2nd floor

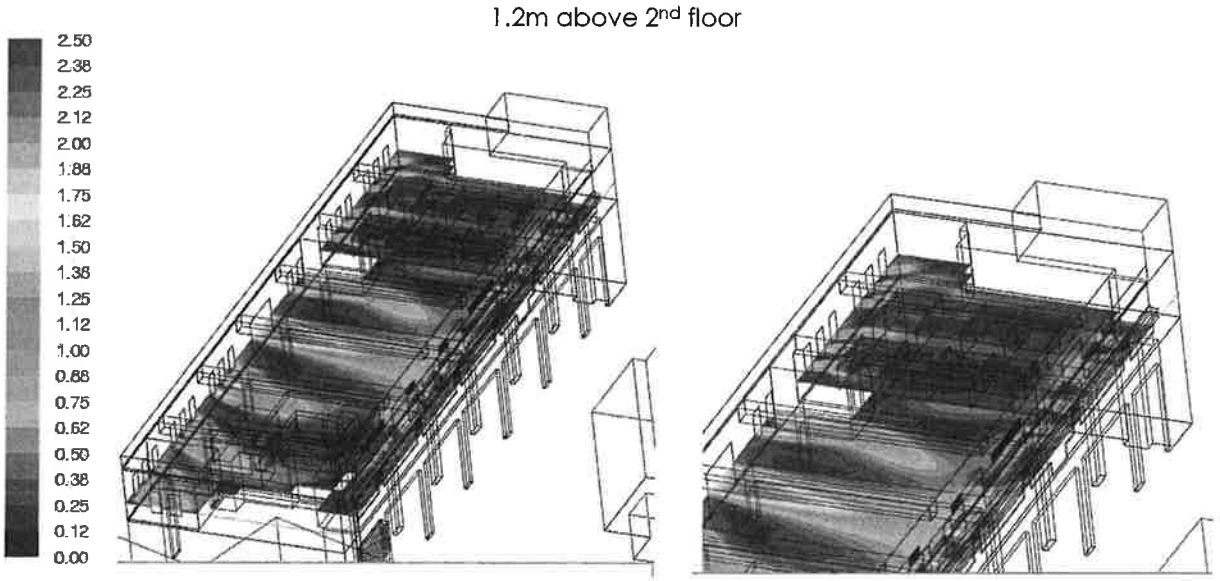
01 Energy Efficiency – Natural Ventilation

Simulation Result – Base Case



01 Energy Efficiency – Natural Ventilation

Simulation Result – Base Case



01 Energy Efficiency – Natural Ventilation

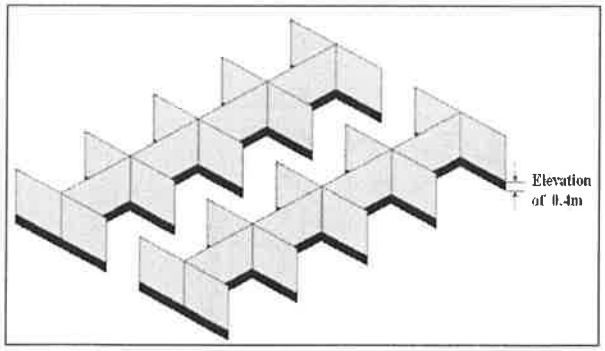
2-Storey Training Block	Minimum Velocity (m/s)	Area-Weighted Average Velocity (m/s)	Maximum Velocity (m/s)
First Storey (1.2m)	0.002	1.364	2.385
Second Storey (9.3m)	0.002	**0.505**	2.179
Roof (14.65m)	0.003	1.098	2.743
Lift Motor Room (14.65m)	0.003	**0.126**	0.719

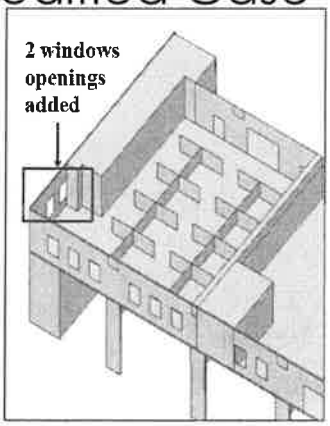
01 Energy Efficiency – Natural Ventilation

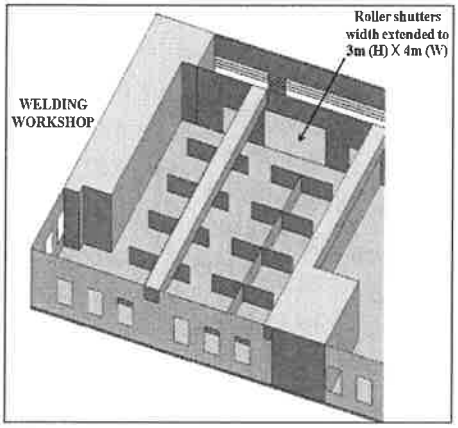
- Further recommendations for airflow improvements (targeting to achieve average velocity of 0.6m/s)
  - Elevating the partition in welding workshop, e.g. 0.4m
  - Opening provision for the linking bridge
  - Increasing the width of the 1 number of roller shutter at the welding area by 1m , ie to size 3m high and width 4 m wide
- Modified Case CFD study to include the followings
  - Louver height to be increased by 10cm (upwards)
  - Two opening windows (1m x 2m) at welding workshop
  - Include pantry in office as directed by BCA
  - To put in the door at Grid G/2
  - Geometry for the Upcoming BCAA 10 storey building is updated

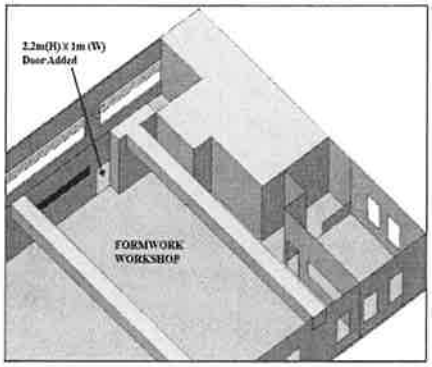
01 Energy Efficiency – Natural Ventilation

Geometry – Modified Case



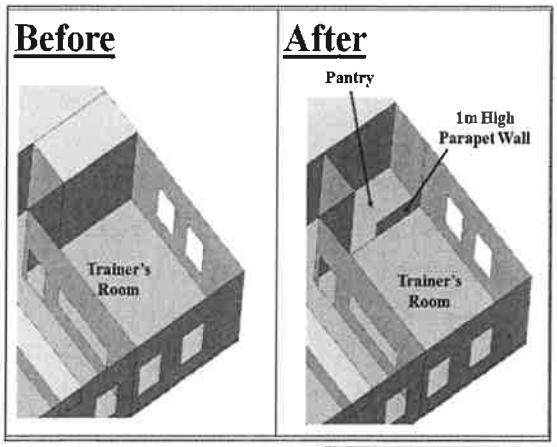


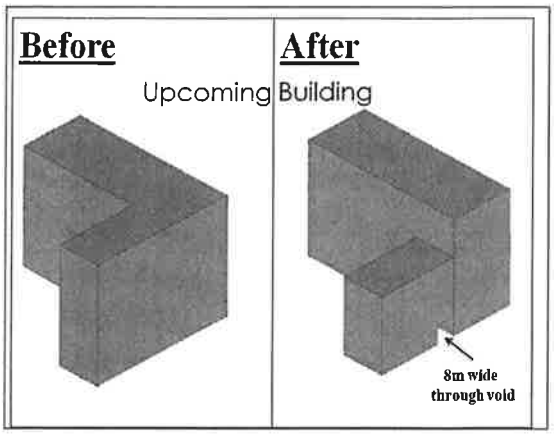


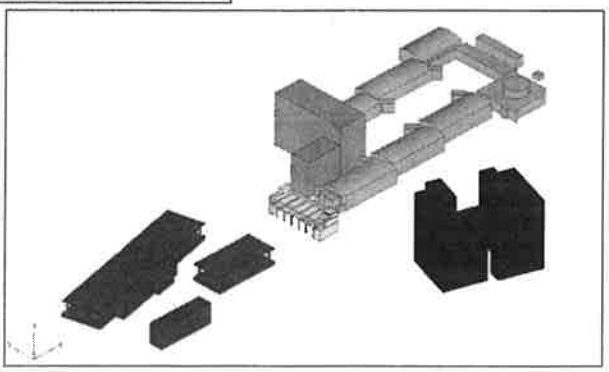


01 Energy Efficiency – Natural Ventilation

Geometry – Modified Case

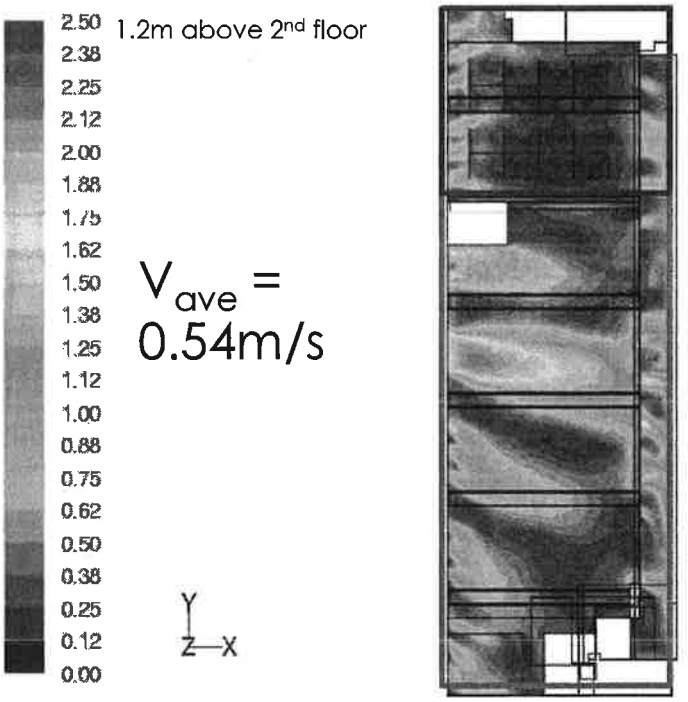


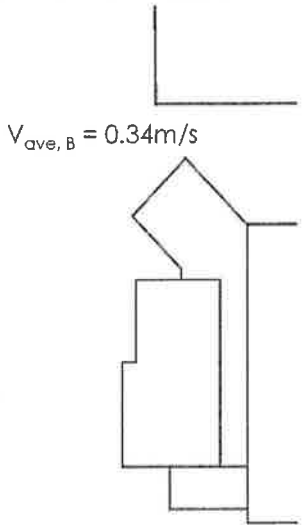




01 Energy Efficiency – Natural Ventilation

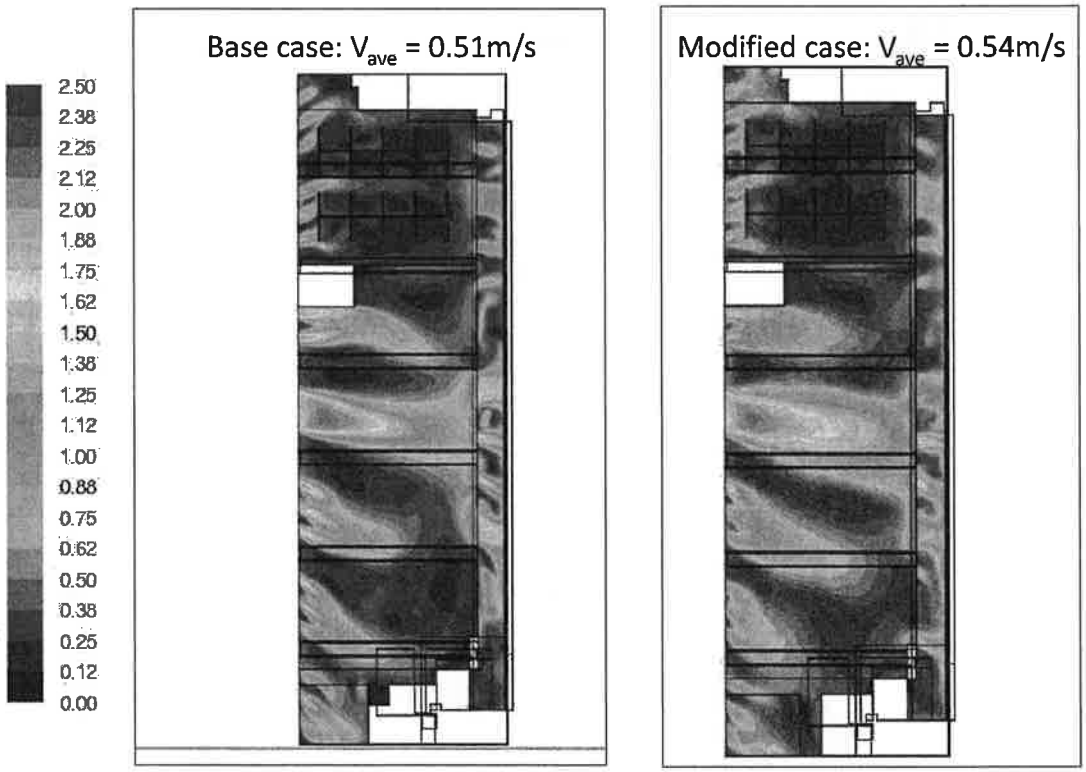
Simulation Result – Modified Case





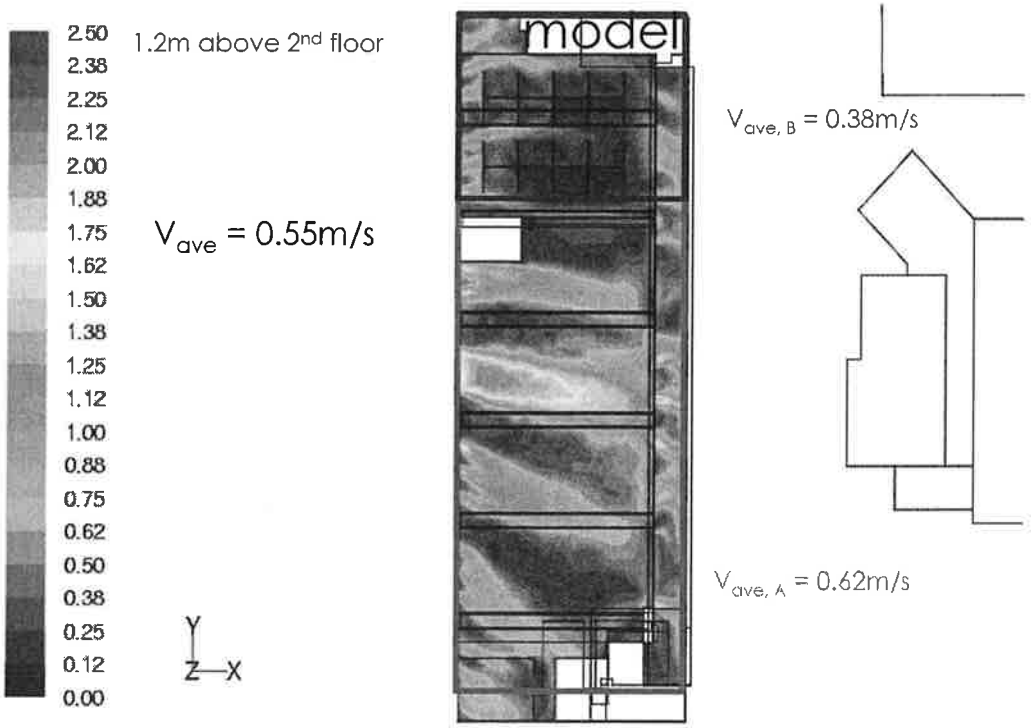
01 Energy Efficiency – Natural Ventilation

Simulation Result – Comparison



01 Energy Efficiency – Natural Ventilation

Simulation Result – Modified Case – with more advanced RNG K-Epsilon turbulence

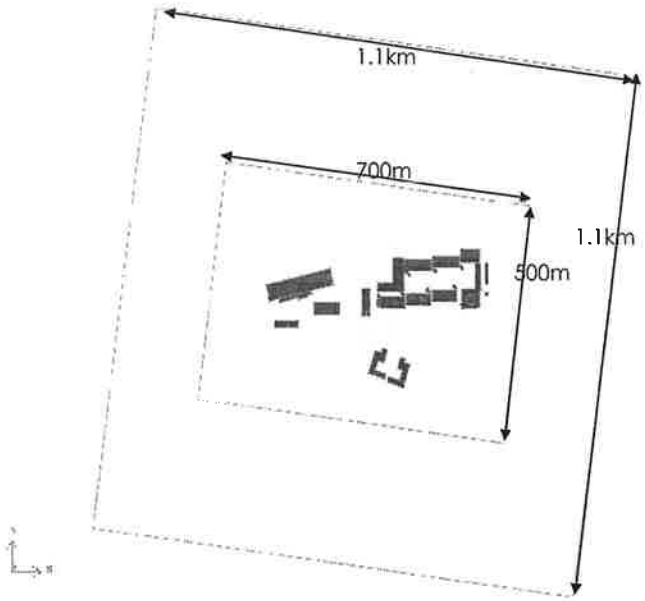


01 Energy Efficiency – Natural Ventilation – Platinum Requirement

1. The welding workshop area is recommended to be excluded from the computation due to the nature of the operating environment (in elevated temperature). It is assumed that the mechanical ventilation system for this workshop is well designed for, in accordance to safety standards.
2. The computational domain is recommended to be extended, such that the flow behavior about the training facility is not significantly influenced by the extension of the far field boundaries.
3. To use the RNG k-ε turbulence model
4. To present both the area-weighted & mass-weighted average velocity
5. To proceed with additional simulation, in any 2 (out of 4) favorable wind directions

01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Domain Extension

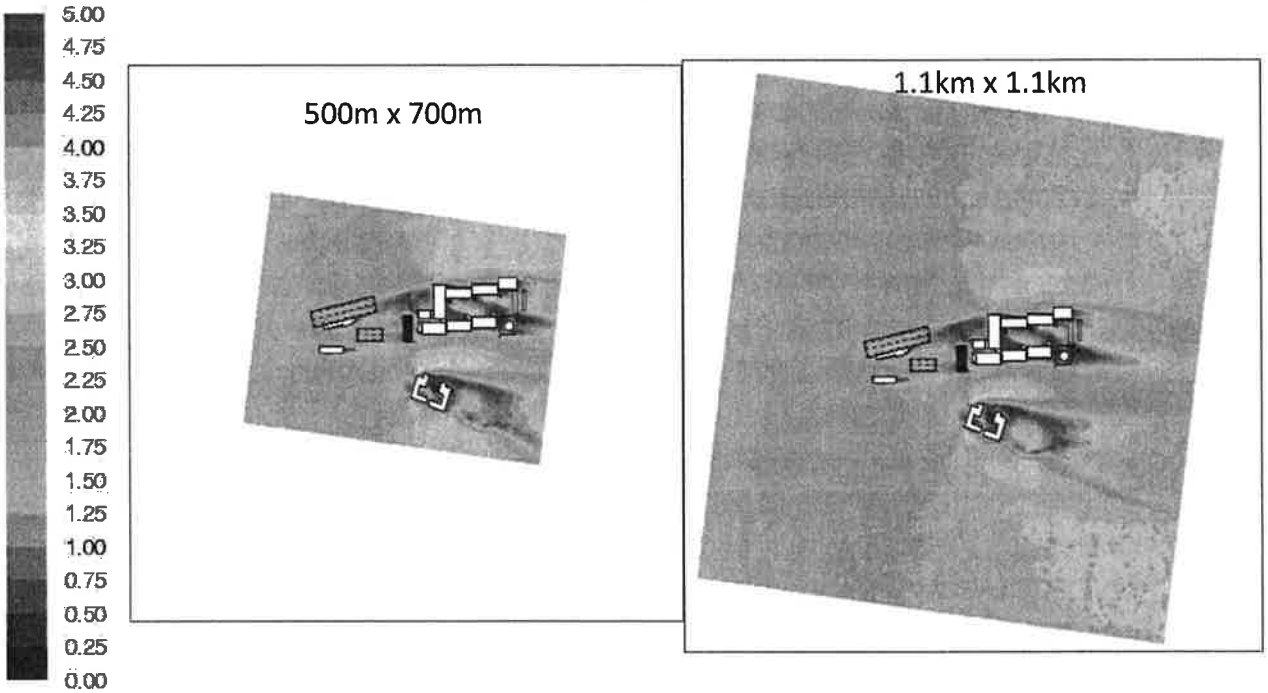


Computational height is also increased from 150m to 300m

23 million to 27 million cells

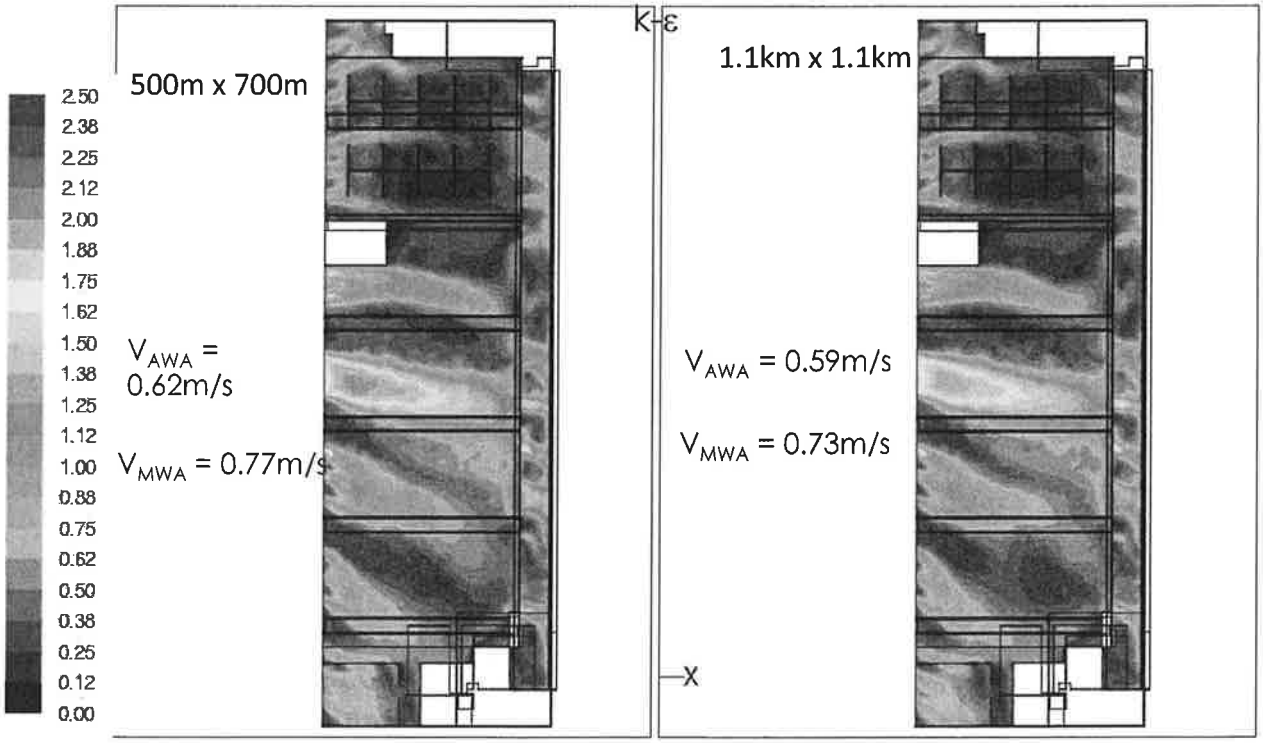
01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Comparison for different domain size on North Wind Results with RNG k-ε



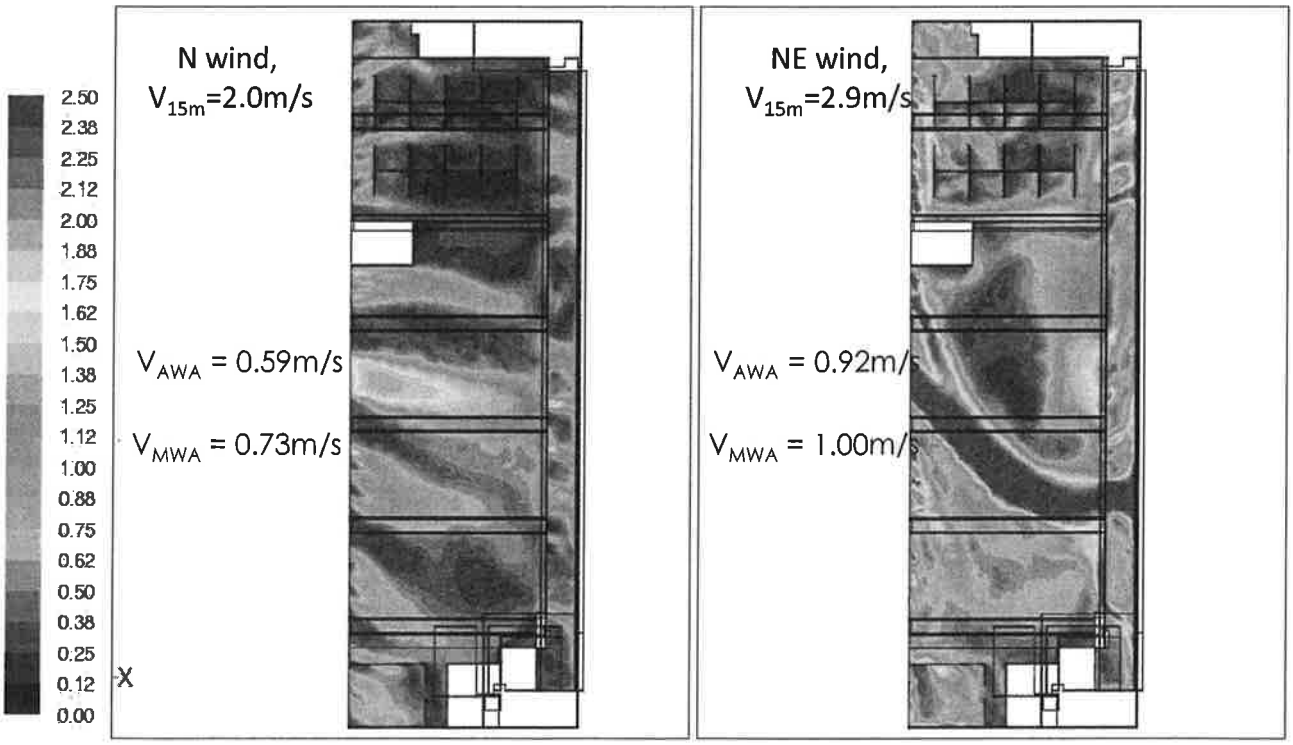
01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Comparison for different domain size on North Wind Results with RNG



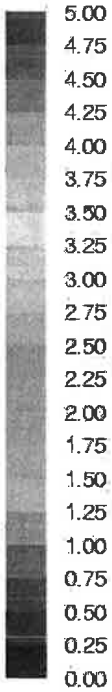
01 Energy Efficiency – Natural Ventilation – Platinum Requirement

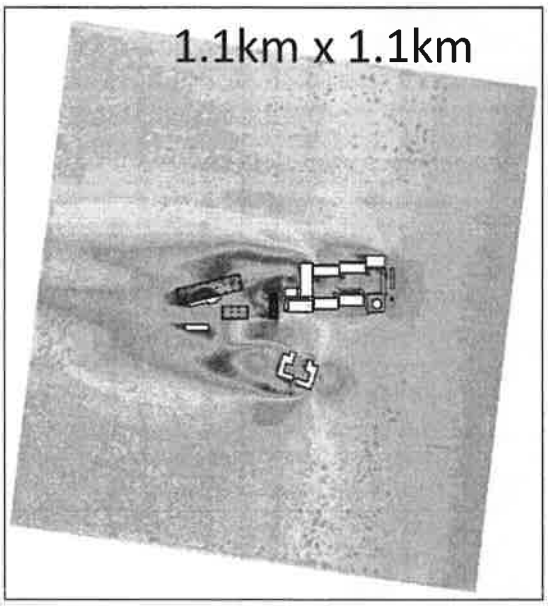
Simulation Result Comparison on North and North East Wind



01 Energy Efficiency – Natural Ventilation – Platinum Requirement

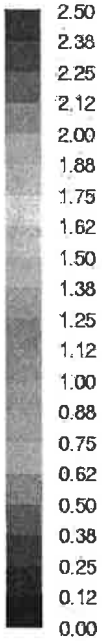
South Wind Results with RNG k-ε

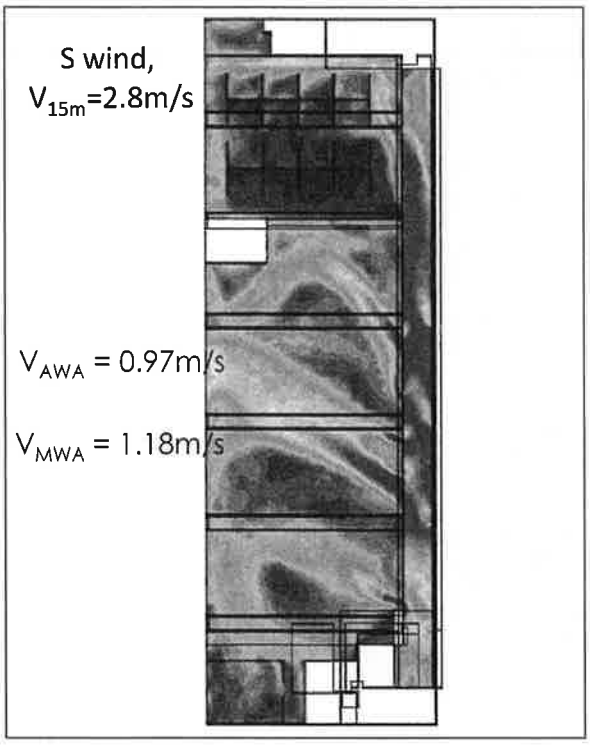




01 Energy Efficiency – Natural Ventilation – Platinum Requirement

Simulation Result on South Wind





01 Energy Efficiency – Natural Ventilation – Platinum

Requirement Quantification on Natural Ventilation

- The percentage of units achieving good natural ventilation

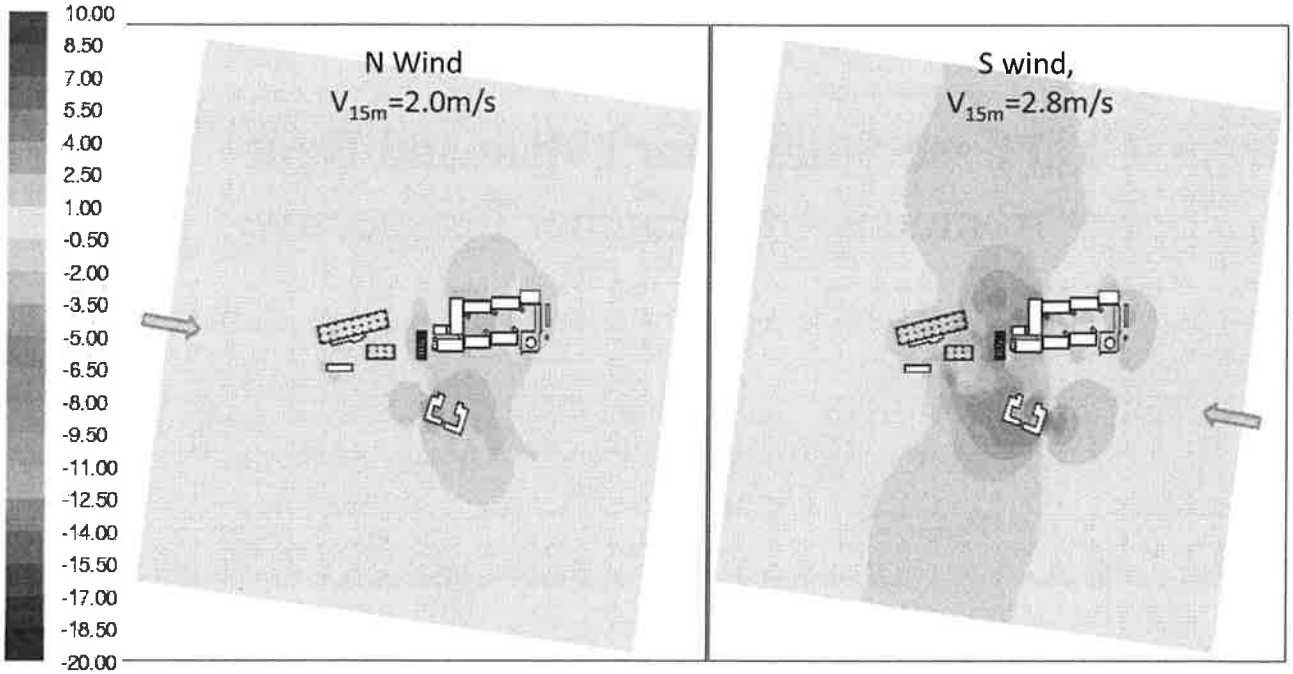
$$\frac{\sum(\text{No. selected units for Each Layout x Area Weighted Average Wind Velocity})}{\text{Total number of selected units x 0.6m/s}} \text{ x } 100\%$$

- The percentage of units achieving good natural ventilation (North East & South wind) =

$$\frac{0.92+0.97}{2 \text{ x } 0.6} \text{ x } 100\% = 158\% > 100\%$$

01 Energy Efficiency – Natural Ventilation – Platinum

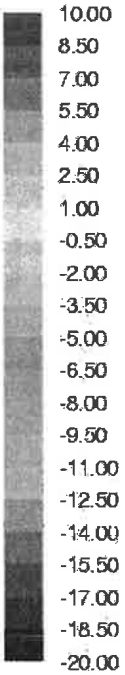
Requirement Pressure Distribution



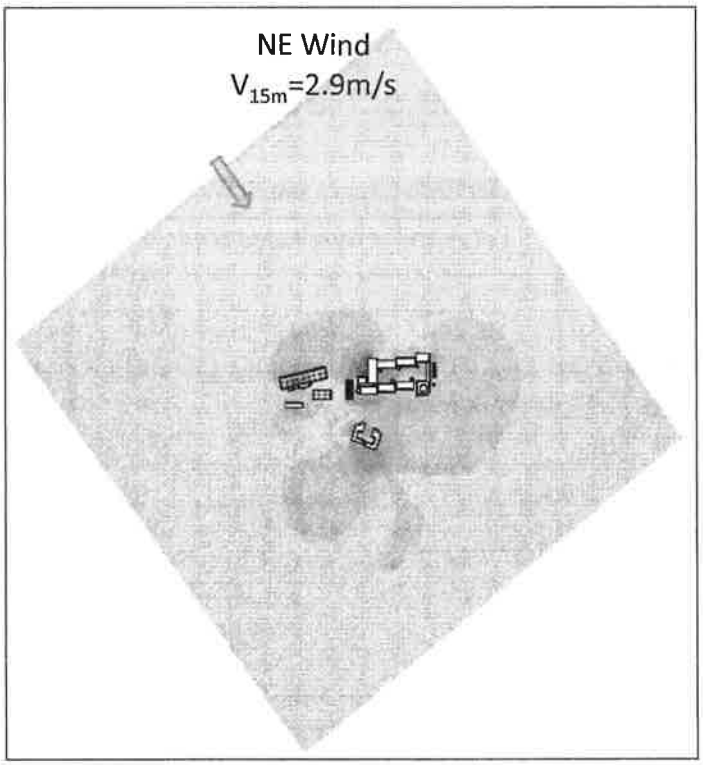
01 Energy Efficiency – Natural Ventilation – Platinum

Requirement

Pa



Pressure Distribution



390

Day 4 (9:45am – 10:30am)

- CFD Case Studies for Urban and Built Environment – Researcher Perspective

403

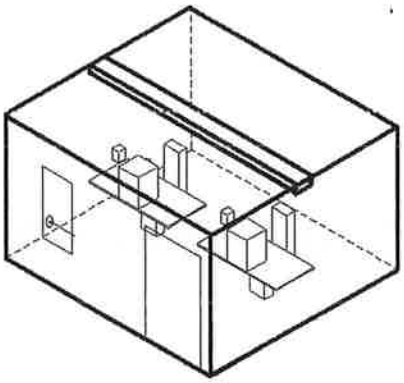
CFD Case Studies for Urban and Built Environment – Researcher Perspective

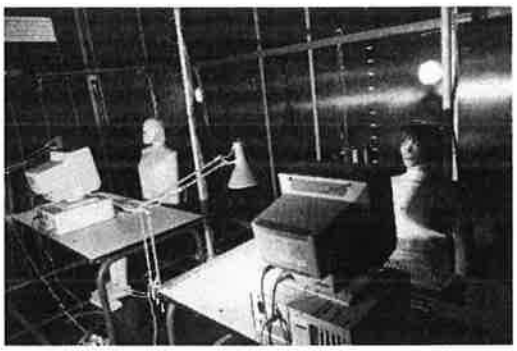
Case Studies on Different Air Distribution Systems

Five air distribution systems are compared with each other. They are all installed in the same room, and they all handle the same situation and the same load.

Mixing ventilation with end wall mounted diffuser.	
Vertical ventilation with a textile terminal..	
Displacement ventilation. End wall mounted low velocity diffuser.	
Mixing ventilation generated by a ceiling mounted radial diffuser.	

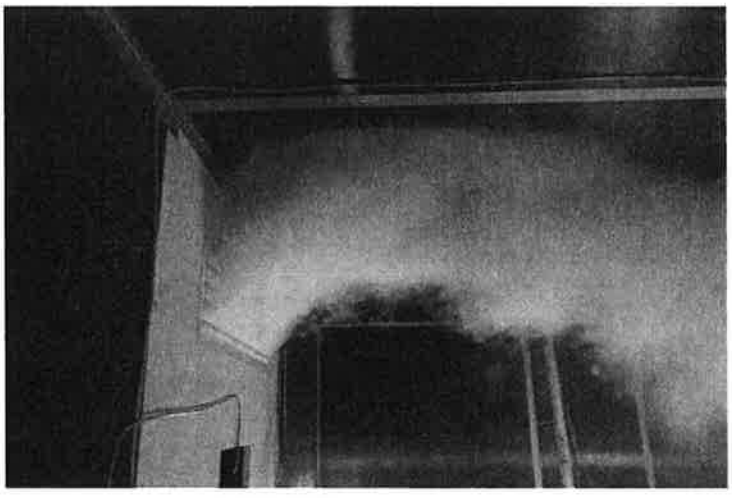
The Test Room



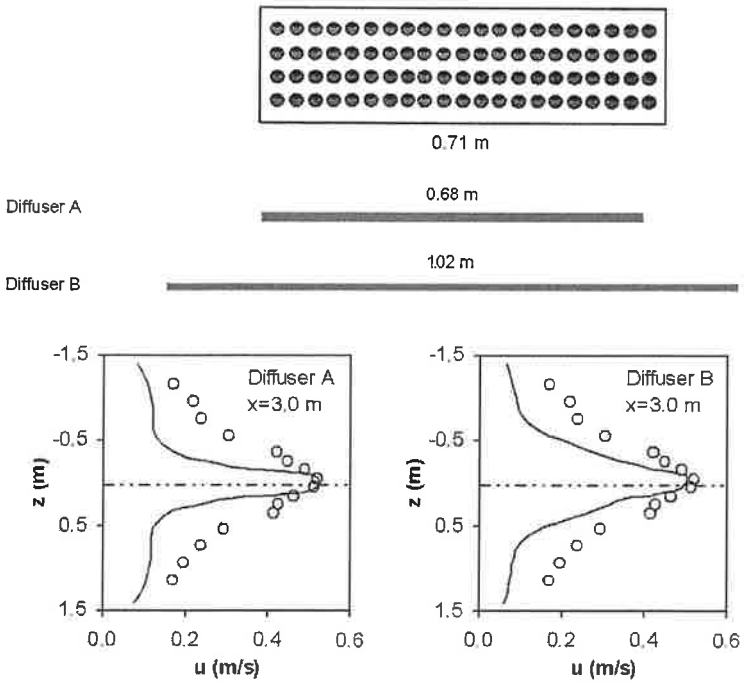


The test room is the IEA Annex 20 room with length, width and height equal to 4.2 m, 3.6 m and 2.5 m.
The heat load consists of two PCs, two desk lamps and two manikins producing a total heat load of 480 W. One work place is used in some of the experiments (240 W).

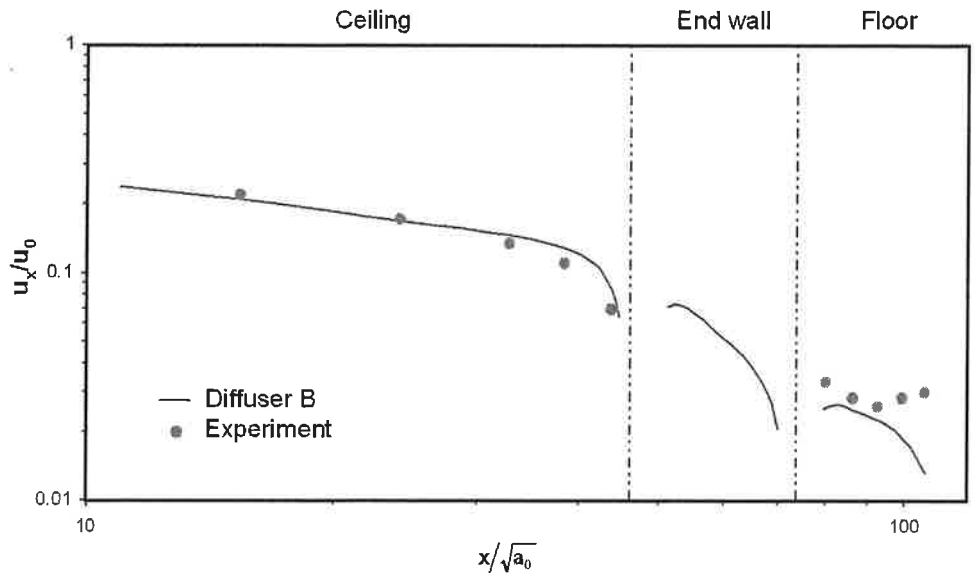
Case Study, Mixing Ventilation with Wall Mounted ATD



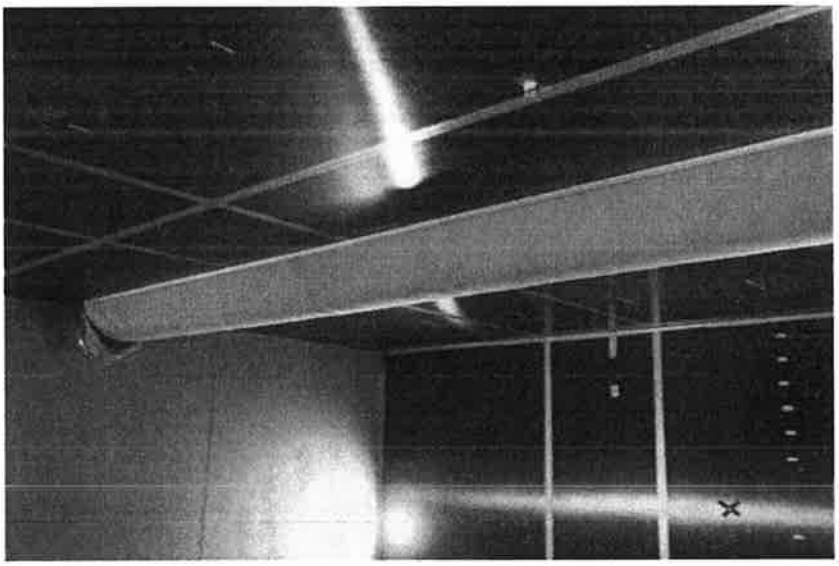
Simulation of the Diffuser



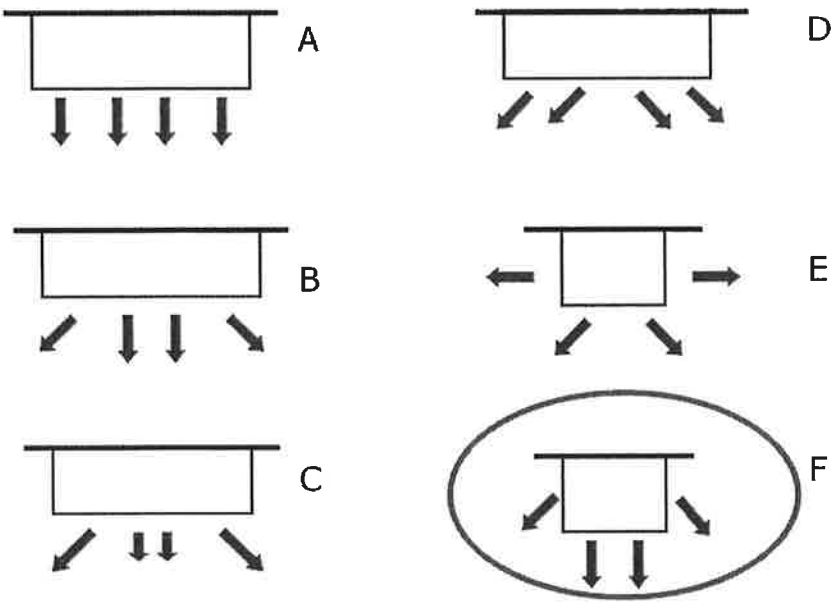
Mixing Ventilation with End Wall Mounted Diffuser



Case Study, Vertical Ventilation

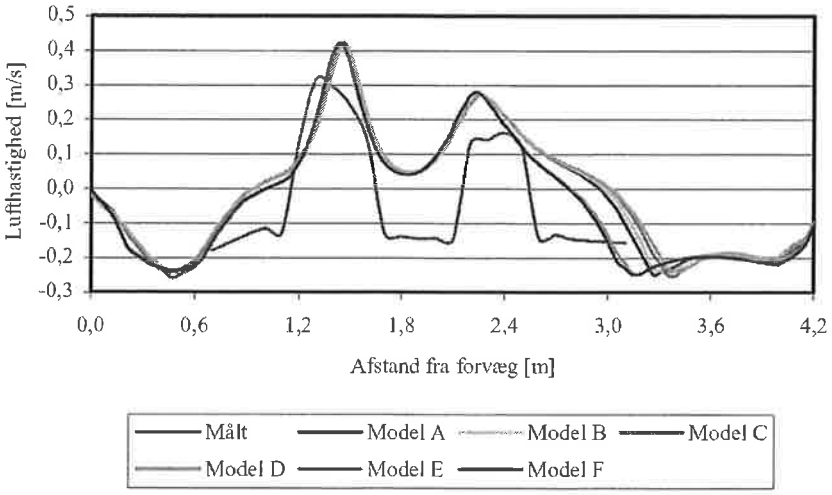


Vertical Ventilation, Diffuser

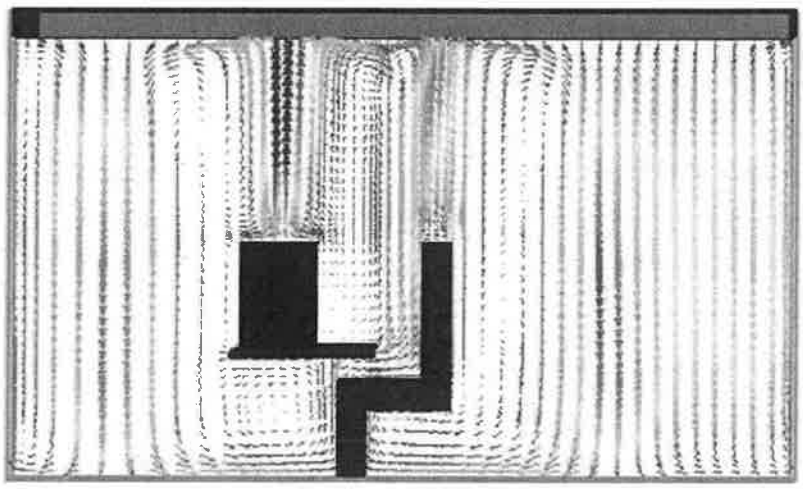


Vertical Ventilation, Diffuser

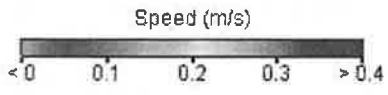
1st order steady state equations, k-ε turbulence model, 300,000 cells



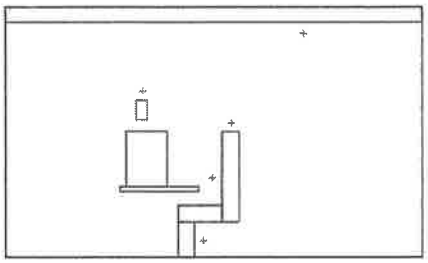
Vertical Ventilation



BC: Diffuser F

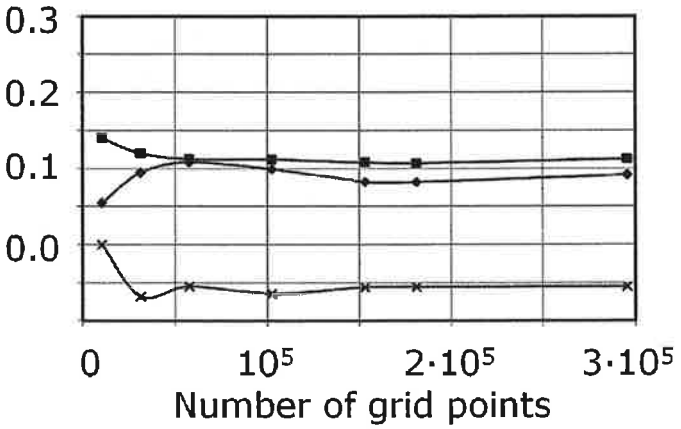


Vertical Ventilation, Quality Control



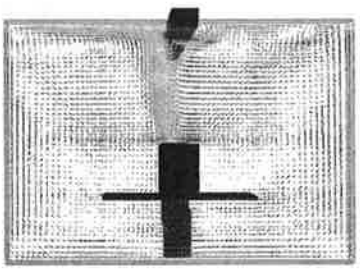
Monitoring points

Velocity



Velocity in y-direction

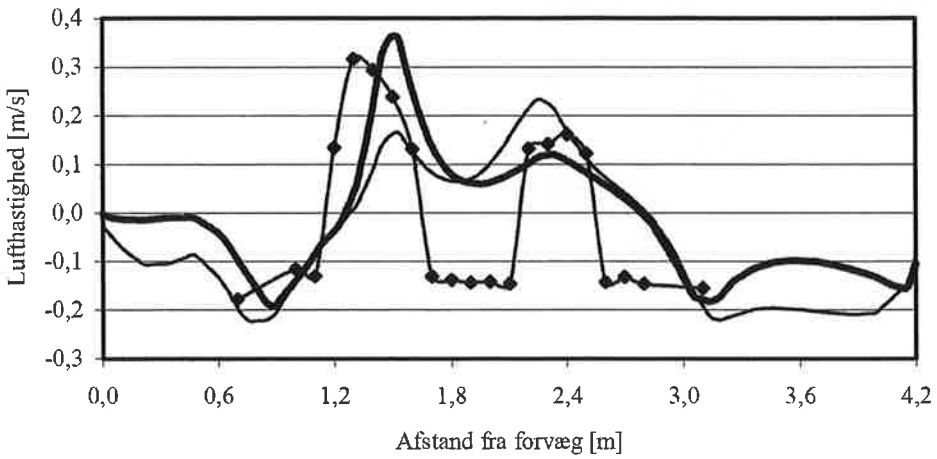
Vertical Ventilation



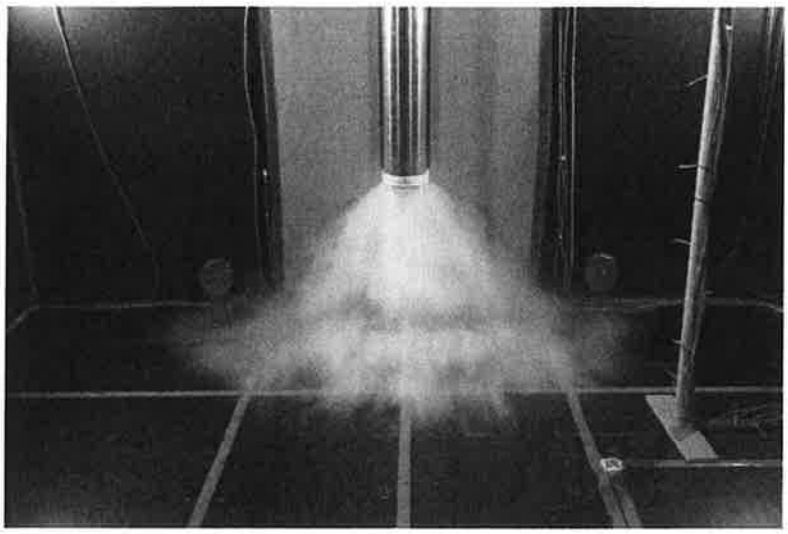
Predictions in the **whole** room

n = 5 h^{-1}

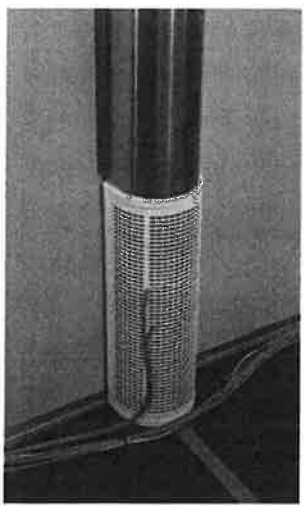
218.400 grid points

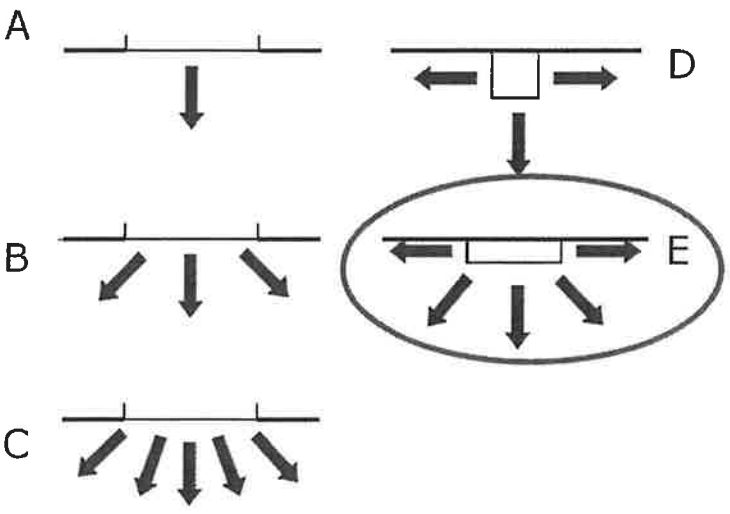


Case Study, Displacement Ventilation

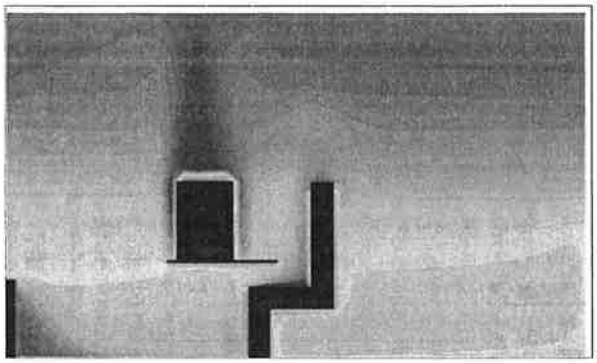


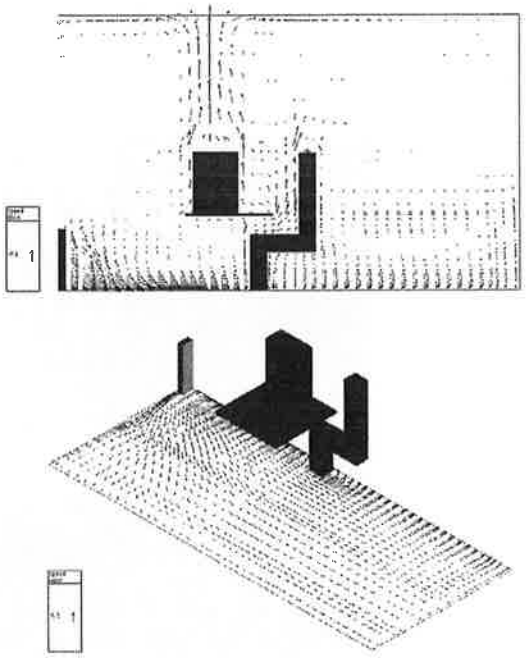
Diffuser for Displacement Ventilation





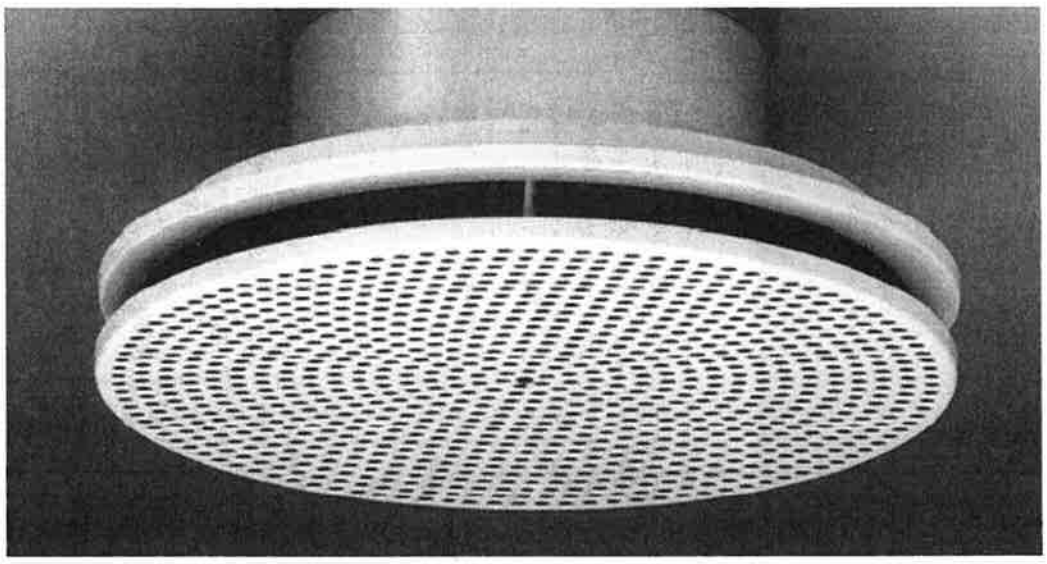
Displacement Ventilation





418

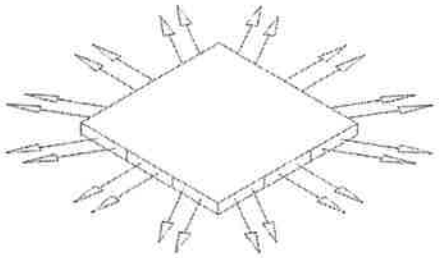
Case Study, Mixing Ventilation with Ceiling Mounted ATD



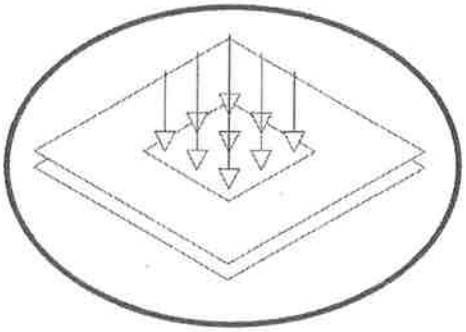
419

Mixing Ventilation with Ceiling Diffuser, Diffuser Models



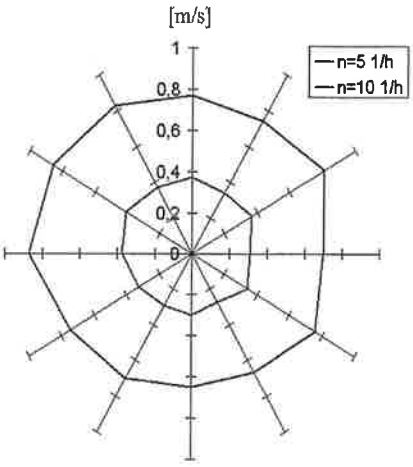


- Fixed-flow diffuser
- Diffuser with horizontal surface

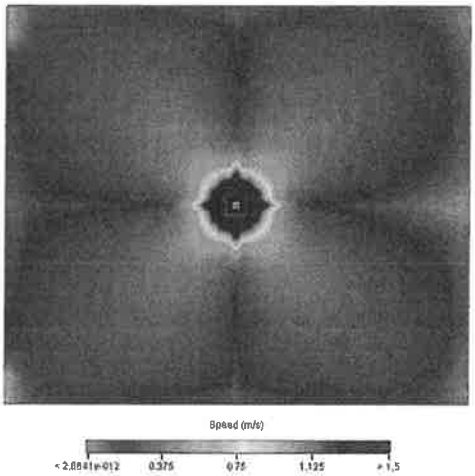


Diffuser Model

Measurements

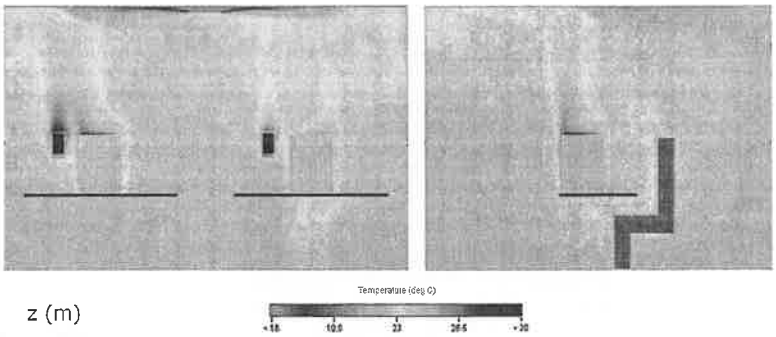


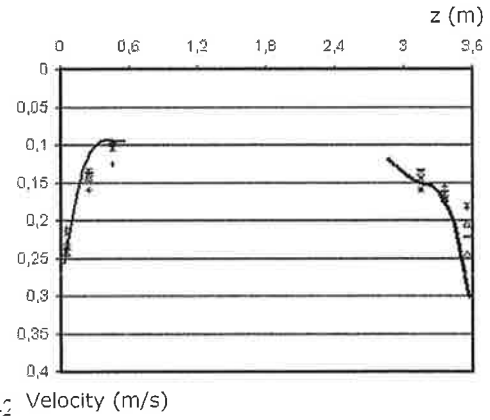
Diffuser with horizontal surface



CFD Simulations

Temperatures

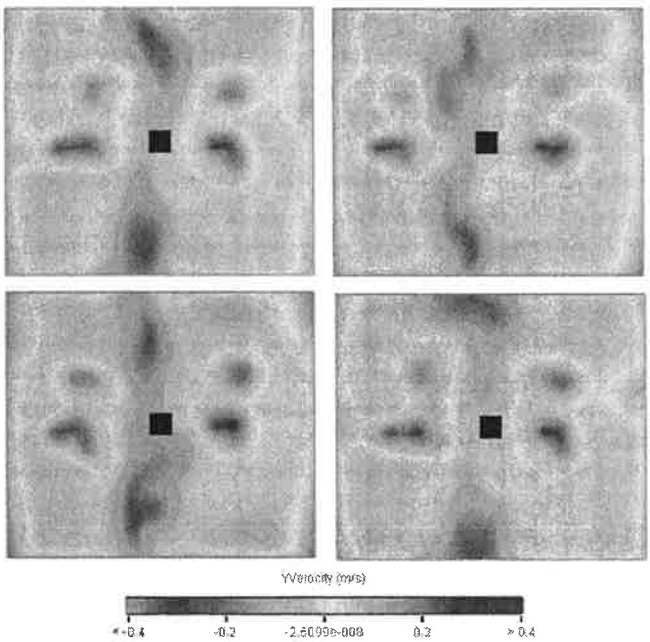




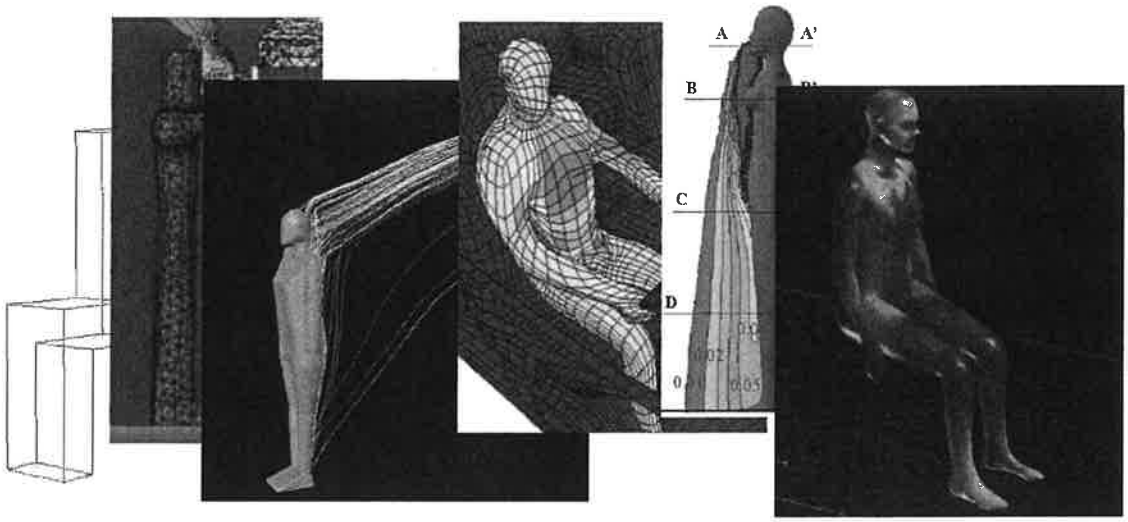
Velocities at the the height of 1.80 m close to the side walls

Simulation of Unsteady Flow

(Time Dependent Equations)



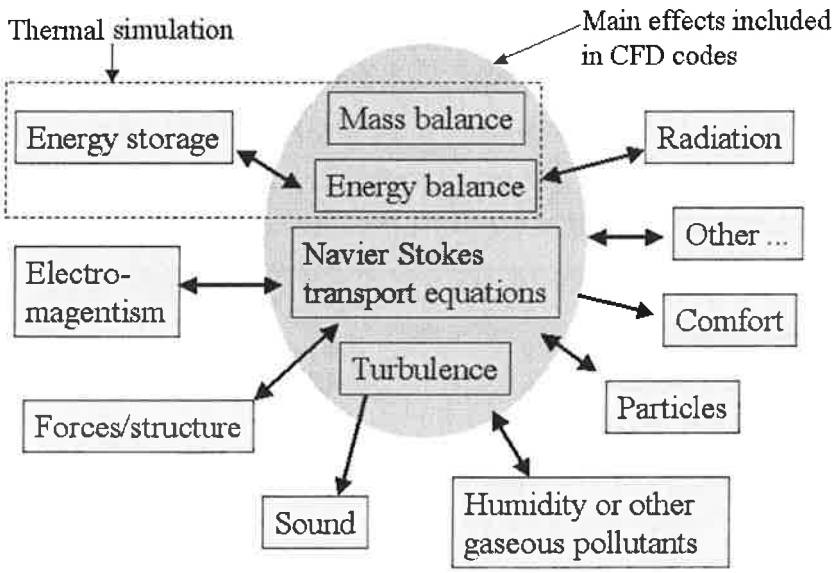
CFD Manikins



Benchmark test for different CFD manikins:

www.cfd-benchmarks.com

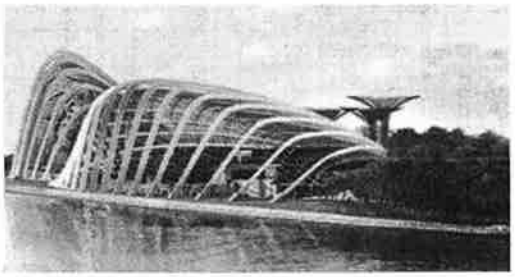
CFD Combined with Other Prediction Models



CFD in Built & Urban Environment

Research Fields	Research Opportunity/Impact
Urban Airflow Dynamics	Climate Change Impact & Urban Planning
Building Airflow Dynamics	Building Energy Efficiency & Integrated Design Tool
Fire Driven Flow Dynamics	Risk Analysis & Mitigation
Pollutant Dispersion Flow Dynamics	Air Quality & Health Assessment





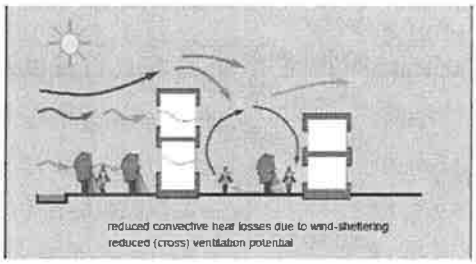


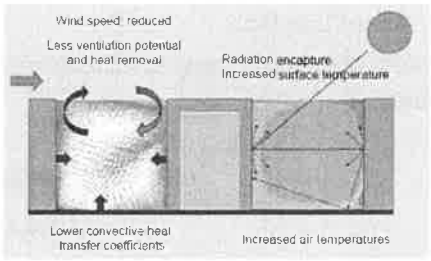
Computer Modelling of Indoor Environment

1. Building thermal analysis codes for predicting the heat transfer to and from the building fabric
2. Air infiltration calculation codes for identifying air leakage through the building fabric and between different zones of the same building (known as zonal and network models)
3. Computational fluid dynamics (CFD) for analysing the wind flow around buildings and the air movement, contaminant distribution, temperature and thermal comfort profiles in rooms
4. Lighting design modelling for day and electric lighting
5. Acoustic analysis codes
6. System simulation codes for controlling environmental systems and energy management
7. Building Information Modelling (BIM)

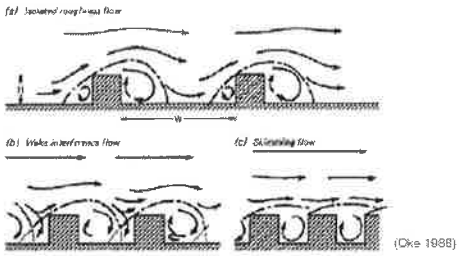
CFD in Urban Physics

1. Urban Street Canyon CFD simulation
   - Buoyant flow in street canyon
   - Urban Microclimatic Modeling : Airflow, Radiation, Urban Heat Island effect





2. Ventilation CFD simulation
   - Urban ventilation, Building Natural & Mechanical Ventilation



(Oke 1988)

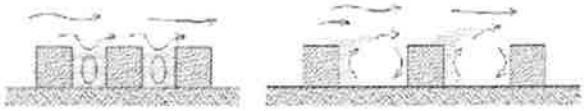
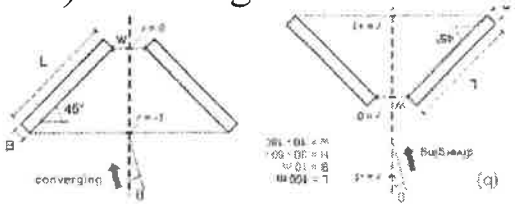


Figure 2.7 Effect of building spacing (street width) to building height on traffic pollution dispersion.

(Awbi 2003)

CFD in Urban Physics

3. Urban Pollutant Dispersion CFD simulation
   - RANS vs. LES; Eulerian vs. Lagrangian model
   - Uncertainty Quantification for ABL Flows – PDF to estimate error bars and evaluate likelihood of certain outcome
4. Rain droplet simulation
   - Wind driven rain and evaluation of catch ratio
   - Impacting rain droplets and film forming on porous materials
5. Building Aerodynamics & Wind Energy
   - Surface roughness near wall (ground) modeling for ABL
   - Venturi effect between buildings
   - AIJ & COST 732 Guidelines

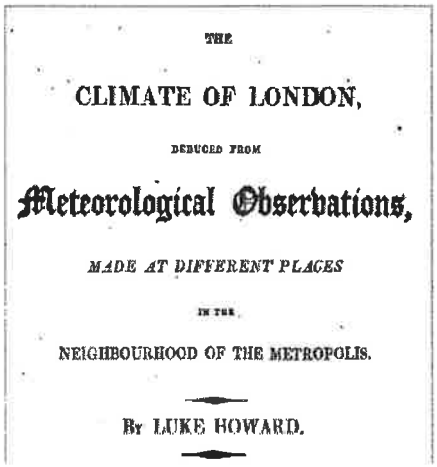


Richardson'(1922)' – Turbulence Energy Cascading

'Big whirls have little whirls that feed on their velocity, and Little whirls have lesser whirls and so on to viscosity'

UHI – Introduction

- Urban areas exhibit higher temperatures than their rural surroundings
- First records of temperature differences:
  - By Luke Howard in 1820;
  - "Tottenham Green, where my latter observations have been made, is four miles from the north side of London…"
  - "… but in the denser parts of the metropolis, the heat is raised, by the effect of population and fires…"



THE

CLIMATE OF LONDON,

DEDUCED FROM

Meteorological Observations,

MADE AT DIFFERENT PLACES

IN THE

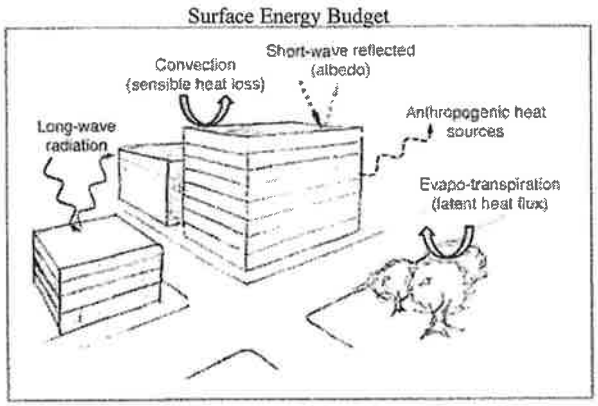
NEIGHBOURHOOD OF THE METROPOLIS.

By LUKE HOWARD.

Lisa Gartland, Heat islands: understanding and mitigating heat in urban areas, London : Earthscan, 2008

UHI – Introduction

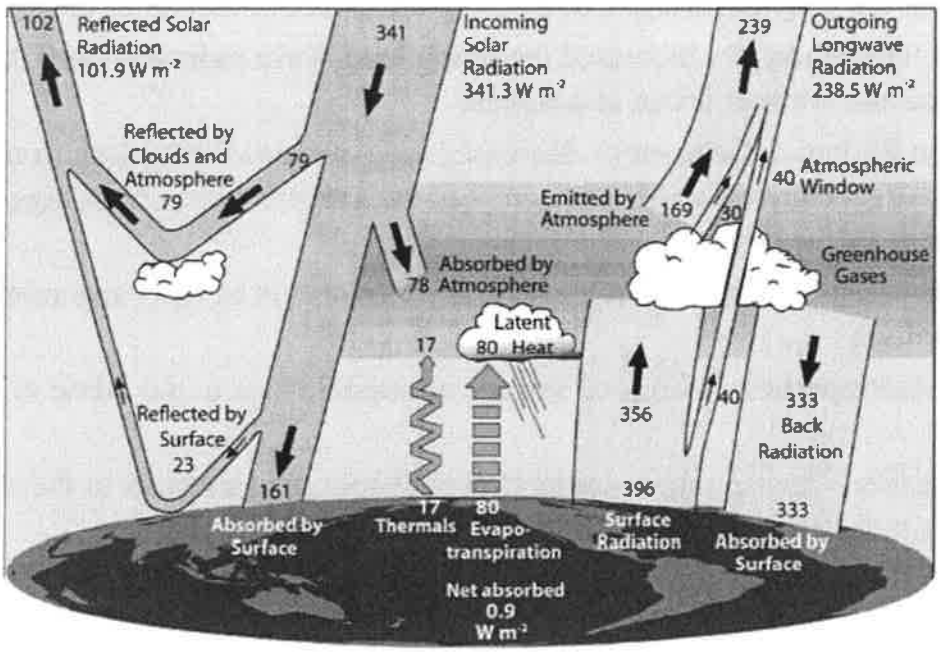
- Heat Island : characterized by consistent rise in urban atmosphere temperature, an increase in air pollutants (e.g. NO_X & SO_X) and decreased in RH
- Numerical approaches to analyze UHI
  - Mechanical model – emphasize flow characteristics
  - Energy balance model – concerned with the difference between energy balance of urban and suburban area



M.E. Hulley, The Urban Heat Island effect: causes and potential solutions, Metropolitan Sustainability: Understanding and Improving the Urban Environment, edited by Frank Zeman, 2012

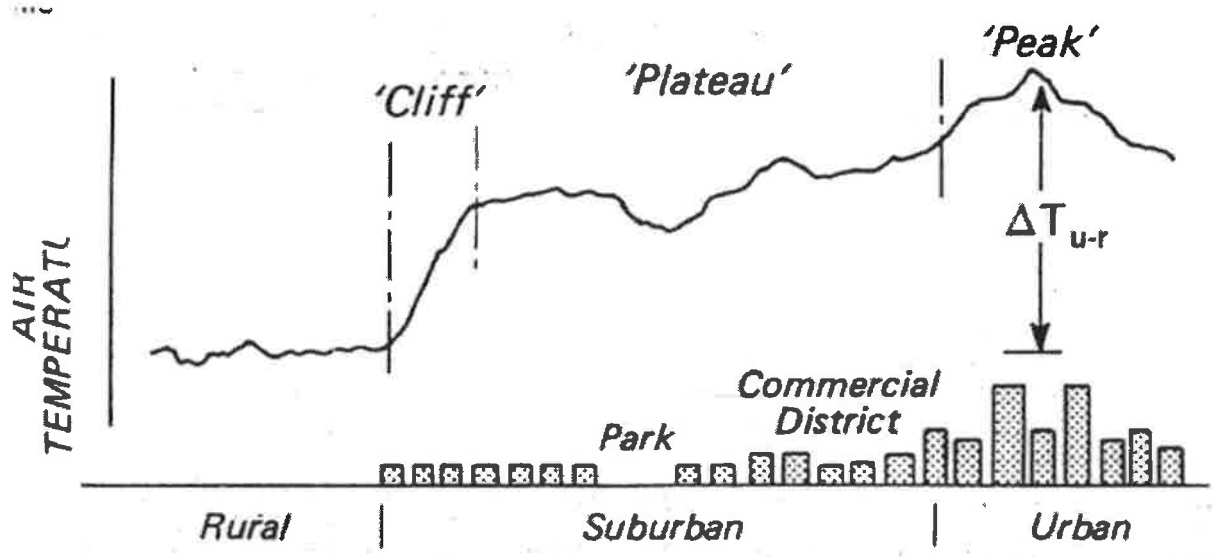
UHI – Surface Energy Budget

Global energy balance in W/m2



The background values (shown in black) of the energy fluxes are based on observations for 2000–05 (Trenberth et al. 2009).

UHI – Temperature Profile



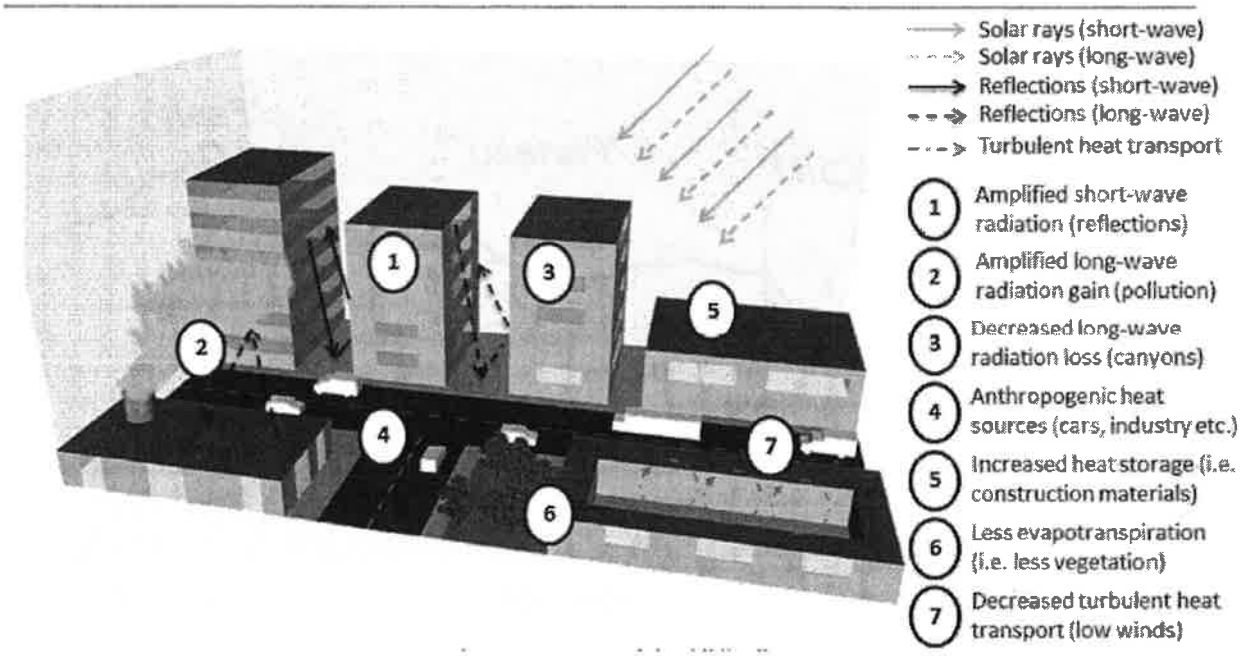
T. R. Oke, "The energetic basis of the urban heat island," *Quarterly Journal of the Royal Meteorological Society, vol. 108, pp. 1-24, 1982.*

UHI – Causes

1. Canyon Radiative Geometry - multiple reflection of short-wave radiation between the canyon surfaces, decreasing the effective albedo of the system
2. Urban 'Greenhouse' - increased incoming long-wave radiation from the polluted and warmer urban atmosphere
3. Canyon Radiative Geometry - decreased long-wave radiation loss from within street canyons due to the complex exchange between buildings and the screening of the skyline
4. Anthropogenic Heat - heat released from combustion of fuels and animal metabolism
5. Thermal Properties - increased storage of sensible heat in the fabric of the city.
6. Evaporation - transpiration - reduction of evaporating surfaces in the city putting more energy into sensible, and less into latent, heat.
7. Shelter - reduced turbulent transfer of heat from within streets

T. R. Oke, et al, Simulation of surface urban heat islands under 'ideal' conditions at night part 2: Diagnosis of causation, Boundary-Layer Meteorology, Volume 56, Issue 4, pp 339-358, September 1991

UHI – Causes



(Picture is courtesy of Torpalar from EUT (personal communication on Mar 2014)

Environmental Fluid Mechanics Modeling

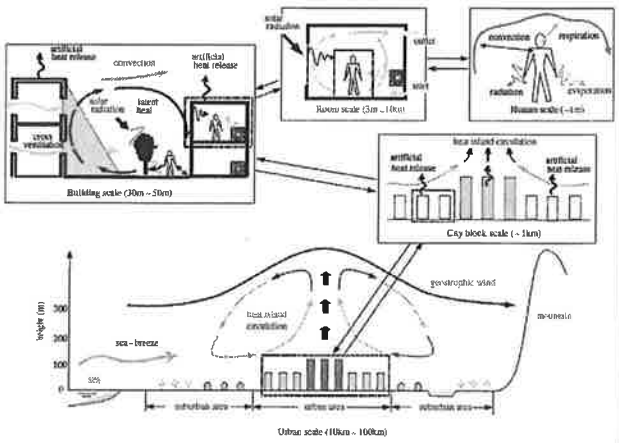
- EFM (Environmental Fluid Mechanics) – Scientific study of naturally occurring fluid flows of air and water on our planet earth, especially those affecting environmental quality of air and water
- Only concern with two bulk of fluids, air and water, under relatively narrow range of ambient pressure and temperature
- EFM is aimed at prediction and decision – generally not aims at design
- Multi-Physics, Multi-Scales & Multi-Disciplinary CFD and Urban Heat Island simulation

Cushman-Roisin, B, Fluid Mechanics of Environmental Interface, Chp. 1; Environmental Fluid Mech: Current Issue and Future Outlook, Taylor & Francis , 2008

Research Challenges

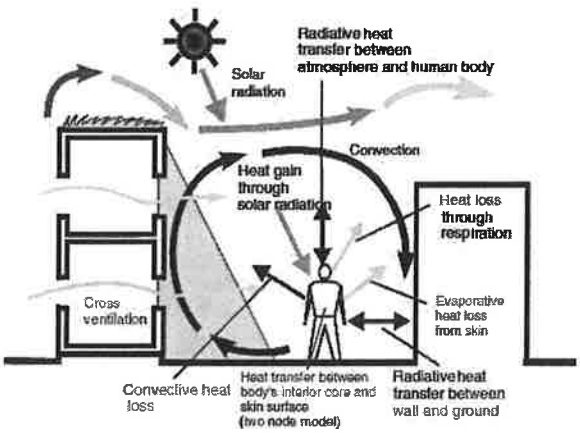
- Challenging issues – multi-scale & multi-physics

Multi-scale



Interaction of phenomena at various scales from indoor to urban

Multi-physics



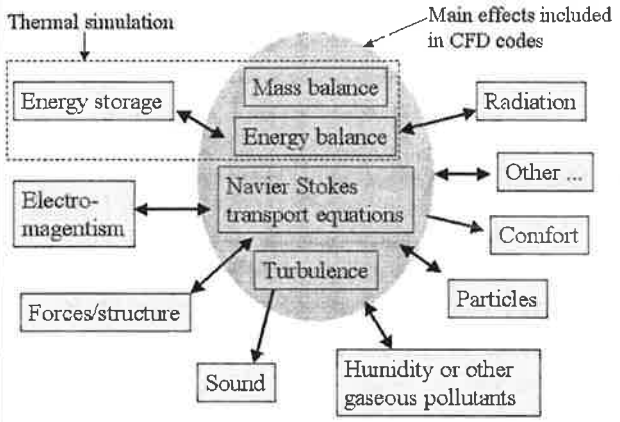
Complicated mechanism of heat exchange between the human body and the surrounding outdoor environment

Picture courtesy of Prof. Shuzo Murakami

Research Challenges

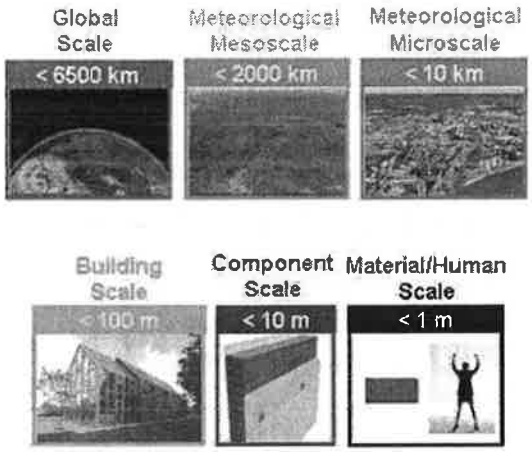
- Challenging issues – multi disciplinary

Multi-disciplinary – modeling methodology



CFD Combined with Other Prediction Models

Multi-disciplinary – urban physics

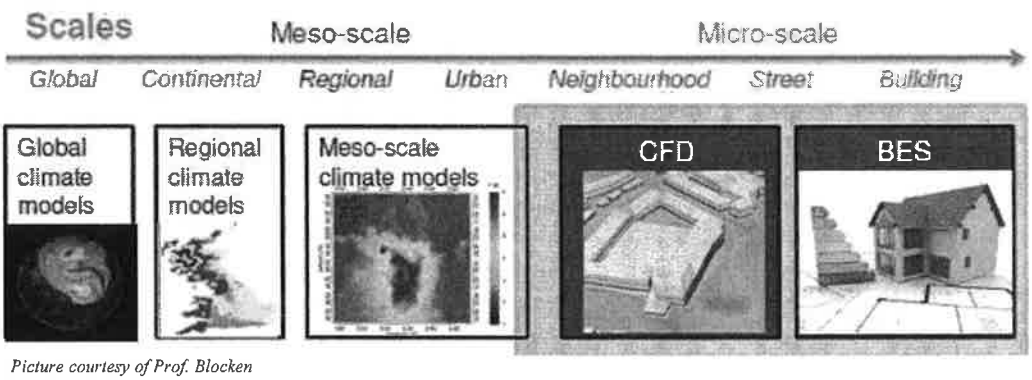


From macro-scale meteorology to human thermo physiology

Picture courtesy of Prof. Hazim Awbi

Research Challenges

- A multi-scale analysis of the Urban Heat Island (UHI) effect: From city averaged temperatures to energy demand of individual buildings
  - Analysis of urban wind flow field
  - Building Energy Simulations: Analysis of energy use for individual buildings.



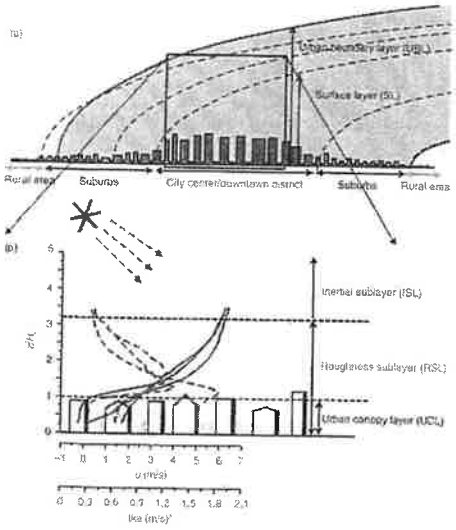
Picture courtesy of Prof. Blocken

Literature Gap

- Literature gaps for complex UHI CFD modelling work
  - treatment of atmospheric stability, urban canopy modelling,
  - quantifying turbulence, energetic, and transport process in Urban Boundary Layer(UBL)
  - prior UHI CFD work was carried out elsewhere (e.g. Tokyo, Japan) and not performed for Singapore island state

Literature Gap – Atmospheric Stability Treatment

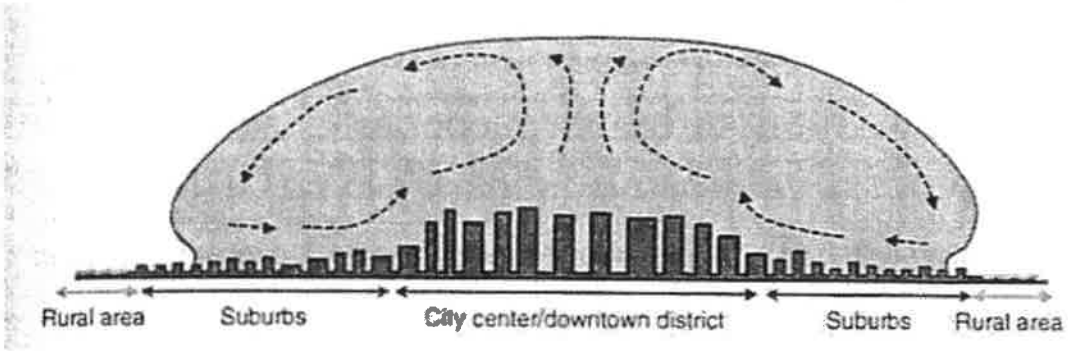
- Layered structure of ABL over urban areas for moderately strong wind speeds (>3m/s)



441 Reference : Mestayer, P.G., et al, The urban Boundary Layer field campaign in Marseille, Boundary Layer Meteorology. 2005. 114:315–365

Literature Gap – Atmospheric Stability Treatment

- Development of thermal driven urban circulation for weak wind (<3m/s)

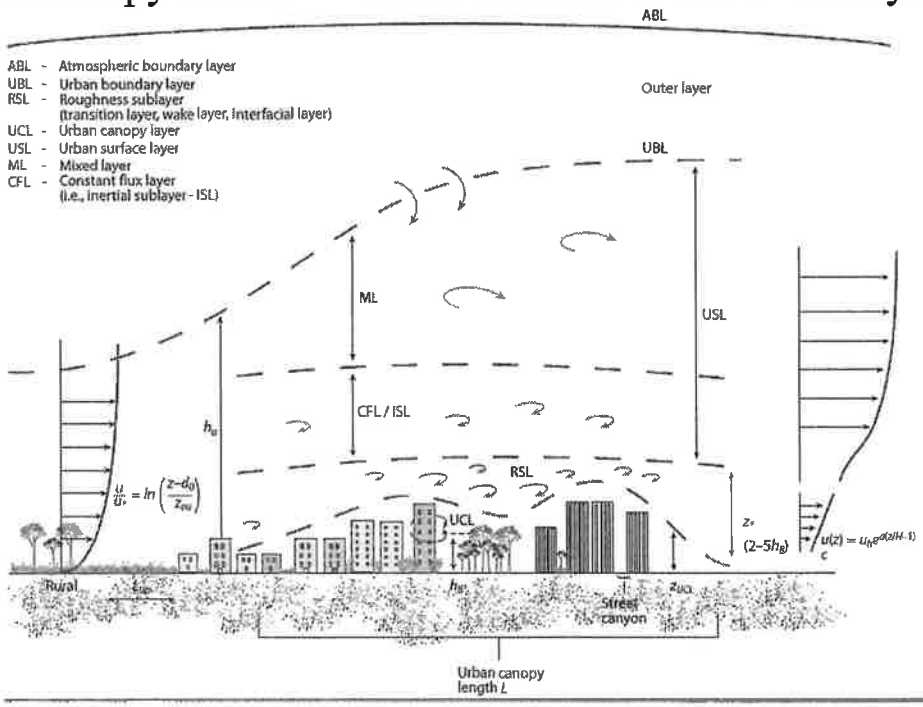


Metropolitan sustainability

442Reference : Mestayer, P.G., et al, The urban Boundary Layer field campaign in Marseille, Boundary Layer Meteorology. 2005. 114:315–365

Literature Gap – Urban Canopy Flow

- Urban canopy effects – various zones of urban boundary layer

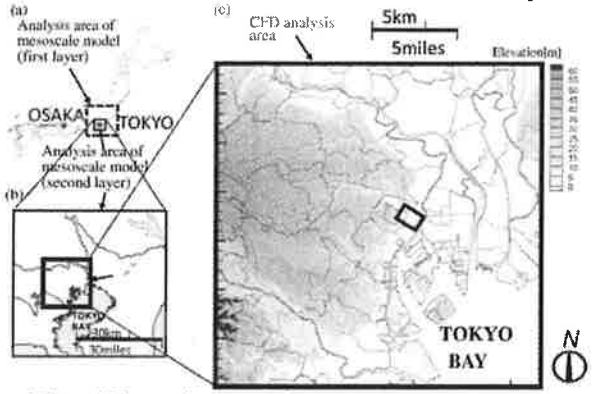


443Reference : H.J.S. Fernando, Fluid Dynamics of Urban Atmospheres in Complex Terrain, Annu. Rev. Fluid Mech. 2010. 42:365–89

Literature Gap – UHI Modeling only for Tokyo

444

- Literature work of Large Scale Computing for Urban Heat Island CFD simulation
  - Domain size – 1,000km^2 (Tokyo Prefecture) x 500m
  - Computational size – 5 billion cells, horizontal spacing 5m
  - Multi-scale geometry with buildings, greenery & water body
  - 300 nodes of Earth Simulator and requires 16hr to complete



CFD analysis domain is 33km x 33km (includes the 33 wards of Tokyo, 621 km^2)

Yasunobu Ashie and Takaaki Kono (2011), 'Urban-scale CFD analysis in support of a climate-sensitive design for the Tokyo Bay area', *International Journal of Climatology*, 31: 174–188

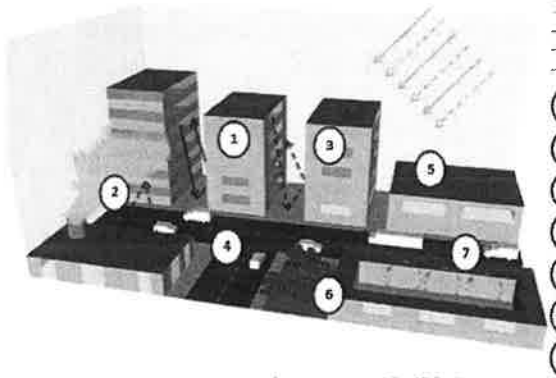
444

445

UHI – Potential Solutions

- To reduce the local air temperature rise in the urban areas and to avoid negative consequences, adaptation measures can be implemented:
  - Evaporative cooling (1, 5, 6)
  - Low anthropogenic heat sources (4)
  - Vegetation (1, 5, 6)
  - High albedo roofs (5)
  - Urban ventilation (3, 7)
  - Street orientation (3, 7)





Solar rays (long-wave)
Reflections (short-wave)
Reflections (long-wave)
Turbulent heat transport

1. Amplified short-wave radiation (reflections)
2. Amplified long-wave radiation gain (pollution)
3. Decreased long-wave radiation loss (canyons)
4. Anthropogenic heat sources (cars, industry etc.)
5. Increased heat storage (i.e. construction materials)
6. Less evapotranspiration (i.e. less vegetation)
7. Decreased turbulent heat transport (low winds)



Moonen, P., Defraeye, T., Dorer, V., Blocken, B., Carmeliet, J., (2012). Urban Physics: Effect of the microclimate on comfort, health and energy demand.

445

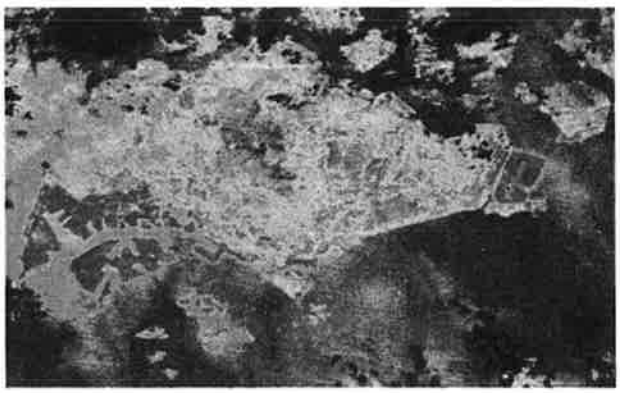
Urban Climatic Mapping Studies for Marina Bay, Singapore



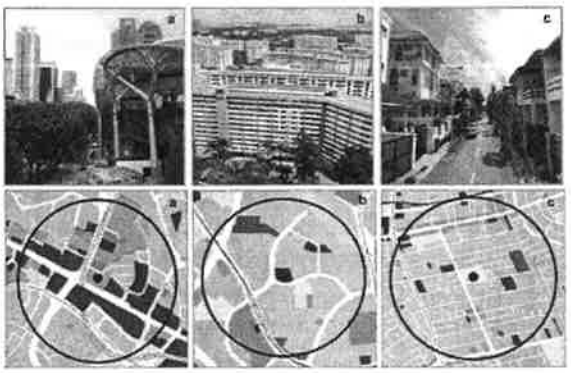
Urban Heat Island profile in Singapore

- UHI effects:
  a. Increasing urban air temperature
  b. Deteriorating air quality and human comfort
  c. Increasing air conditioning cooling load and subsequently building energy consumption

Urban heat island analysis in Singapore



Satellite image of temperature between the 'rural' and the 'urban' areas, (R. Priyadarsini et al., 2008)



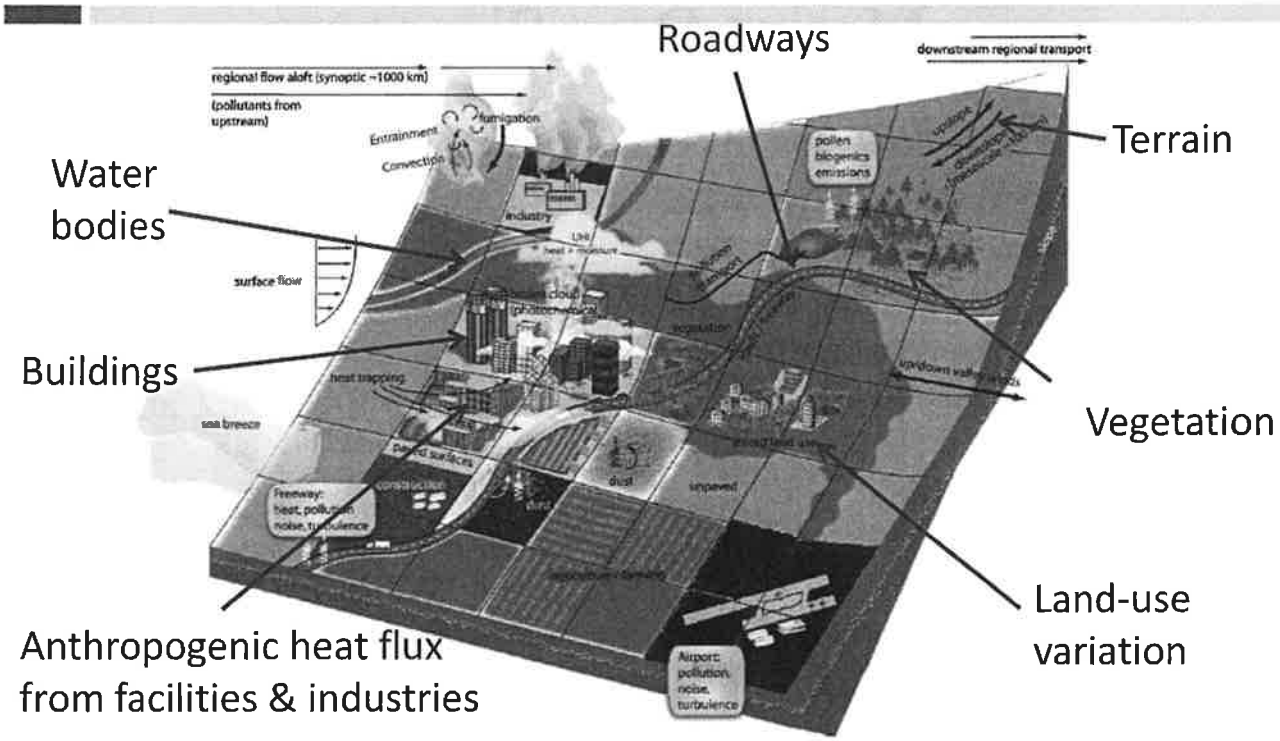
Measurements of anthropogenic heat emissions in Singapore (Quah & Roth, 2012)

How has the UHI phenomenon been tackled so far?

1. Remote Sensing technology with Satellite image
2. Mobile survey/Urban on site Measurement
3. Meteorological data analysis
4. **Multi-physics CFD Analysis**

IHPC
A*STAR

Multi-Physics CFD Analysis

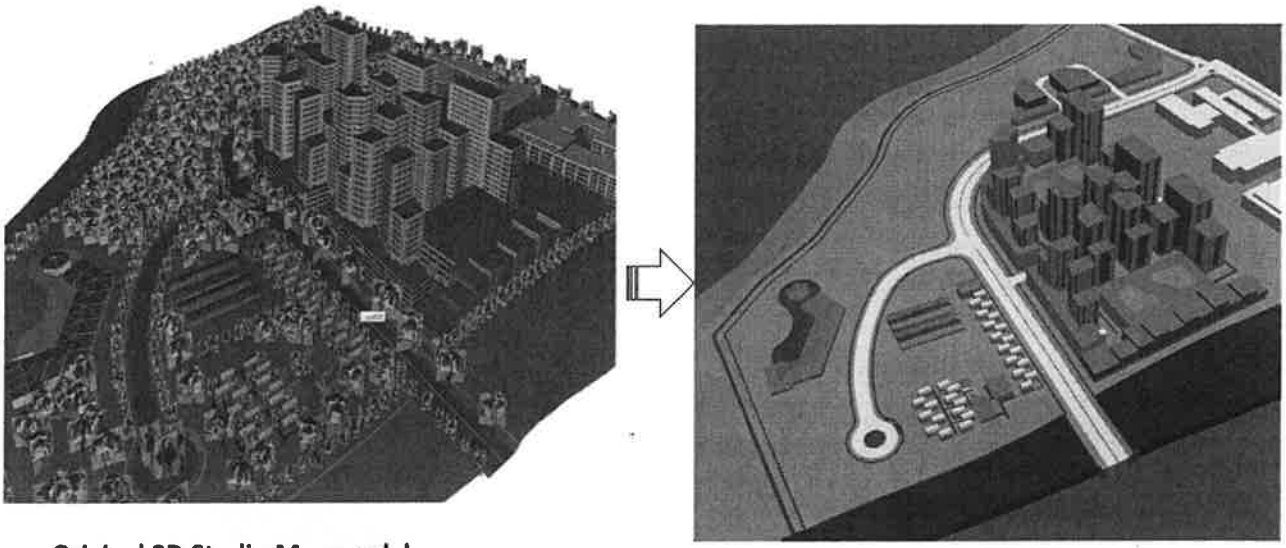


Schematic of flow and turbulence related phenomena typical of urban environments

H. J. S. Fernando, *Phys. Fluids*, 2010

IHPC A*STAR

Geometrical Data Conversion



Original 3D Studio Max model

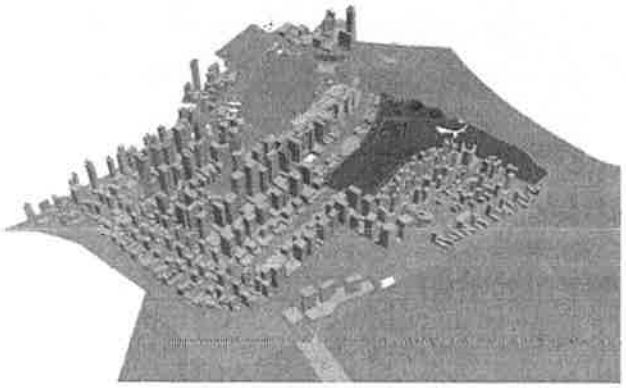
Converted 3D model in STL file format with water tightness and material properties recognition (buildings, windows, vegetation, waterbody, roads, roof, grass, pavement)

H.J. Poh et al, Modelling of Urban Heat Island and Noise Propagation in Singapore, 4th IC2UHI, 2016

IHPC A*STAR

Urban Climatic Mapping Studies for Marina Bay, Singapore

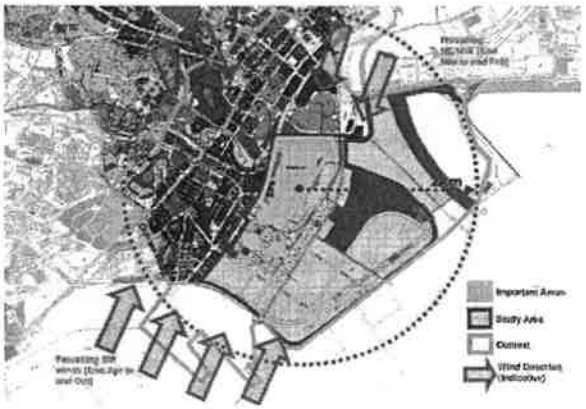
- Objectives
  a. To provide inputs for land use planning that safeguard important environmental conditions from intensive urban development
  b. To reduce heat island effects and energy consumption and improve ventilation in high density environments
  c. To create more liveable and conducive life, work and play spaces in the city



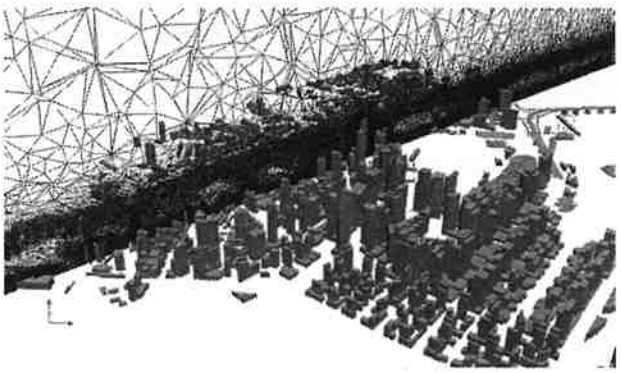
Urban Climatic Mapping Studies for Marina Bay, Singapore

- CFD Wind Simulation
  a. Building geometries are within the proximity of 2 km radius from the heart of the Marina Bay
  b. About 300 buildings of various heights, 90 million tetrahedral cells, meshed with 1.5m resolution

Computational domain

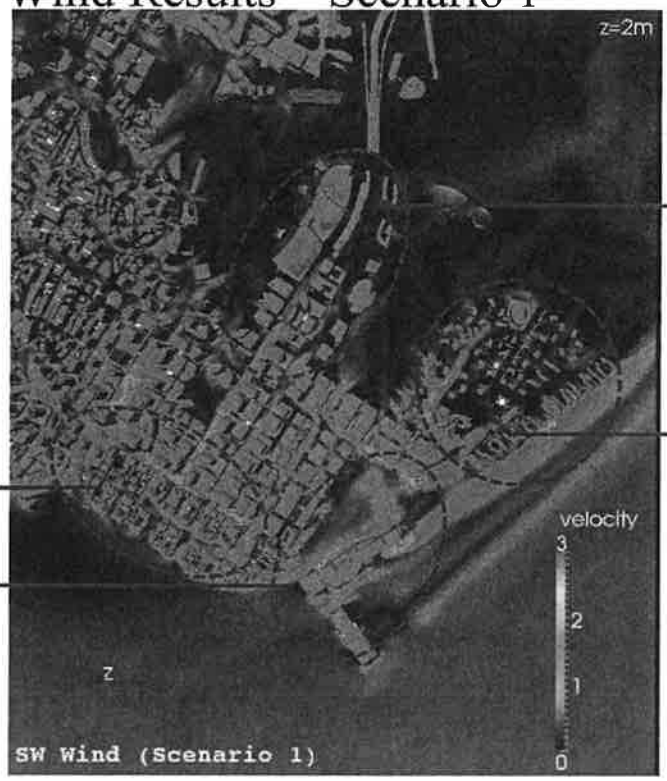


Mesh Resolution



Urban Climatic Mapping Studies for Marina Bay, Singapore

- CFD Wind Results – Scenario 1



Low wind velocity

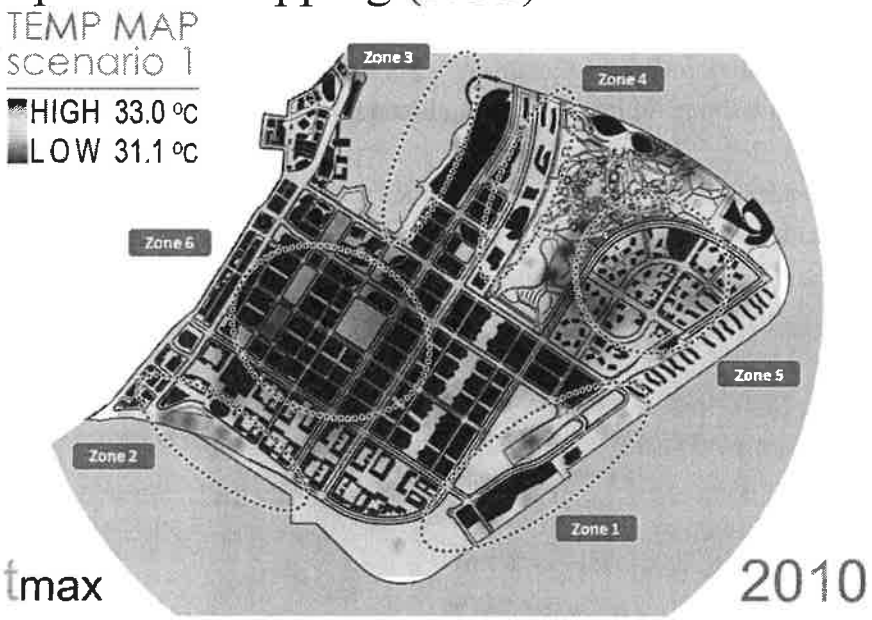
High & low wind velocity

High & low wind velocity

High & low wind velocity

Urban Climatic Mapping Studies for Marina Bay, Singapore

- Temperature mapping (NUS)



Scenario 1's climatic mapping findings highlight that **Zone 1**, **Zone 2** and **Zone 3** are identified to have **high air temperature and low wind movement**.

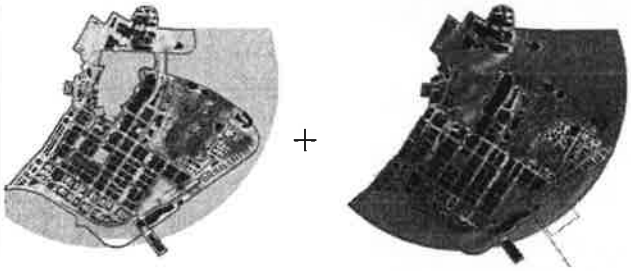
Urban Climatic Mapping Studies for Marina Bay, Singapore

Thermal Sensation Vote

Urban Thermal Comfort

TSV = 0.315Ta - 0.78V - 8.825

TSV = thermal sensation votes;
Ta = air temperature (degC);
V = wind speed (m/s).

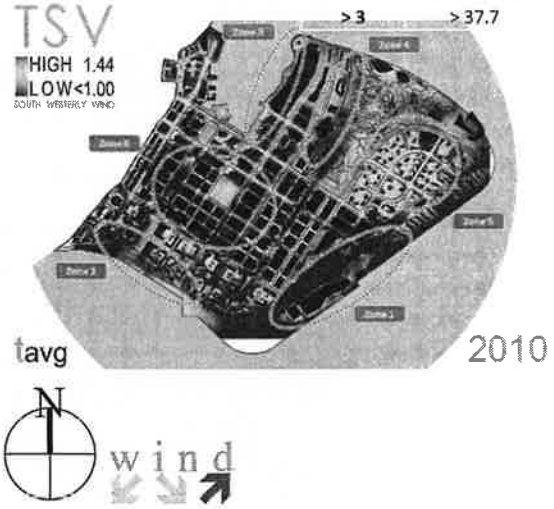


Temperature Map (**GIS**) Wind Map (**CFD**)

454

Acceptable TSV range for particularly for Singapore climate condition

TSV range	Ta range (°C)
-3 ~ -2	Not applicable
-2 ~ -1	22.4 ~ 25.4
-1 ~ 0	25.4 ~ 28.5
0 ~ 1	28.5 ~ 31.6
1 ~ 2	31.6 ~ 34.7
2 ~ 3	34.7 ~ 37.7
> 3	> 37.7



Urban Climatic Mapping Studies for Marina Bay, Singapore

- Scenario 2 improvement - modifications on Marina Bay urban morphology predictors, **mostly done at southern part of Marina Bay**, are listed as follow:
  a. Increasing urban porosity at street level
  b. Removal of low-lying buildings from south-central area
  c. Widening of urban corridor
  d. Application of landscape replacement scheme
  e. Increasing height of the buildings
  f. Increasing greenery provision (building and street level)

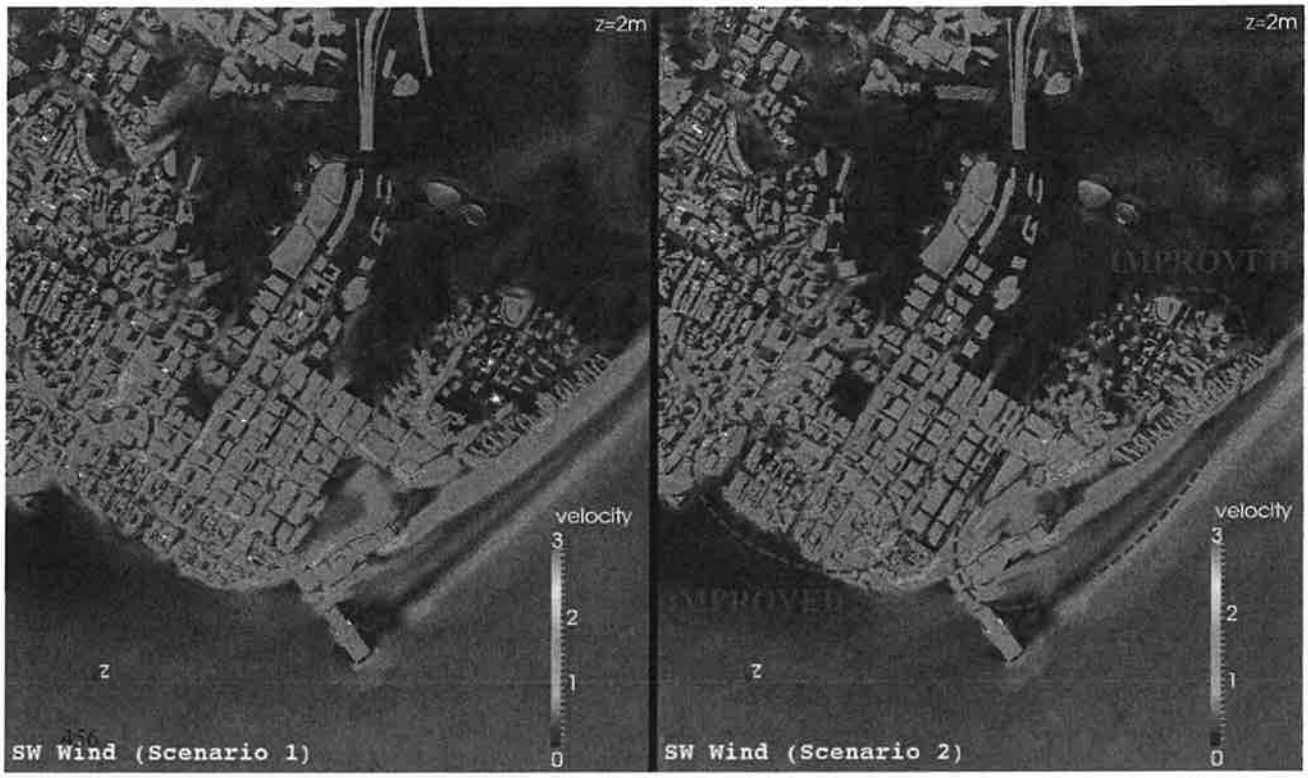


Geometrical comparison between Scenario 1 and Scenario 2

455

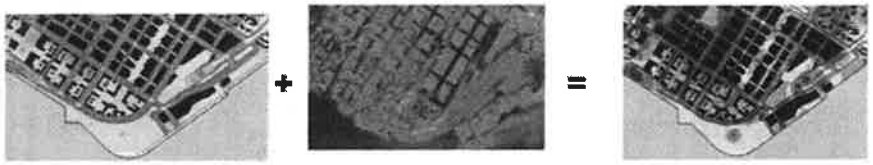
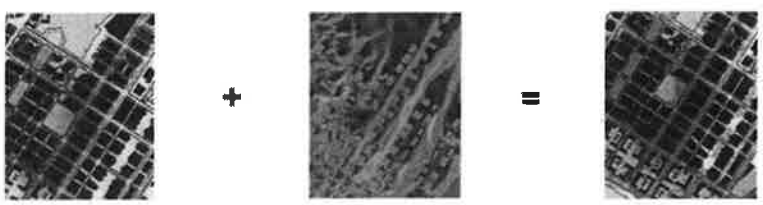
Urban Climatic Mapping Studies for Marina Bay, Singapore

- CFD Wind Results – Scenario 2



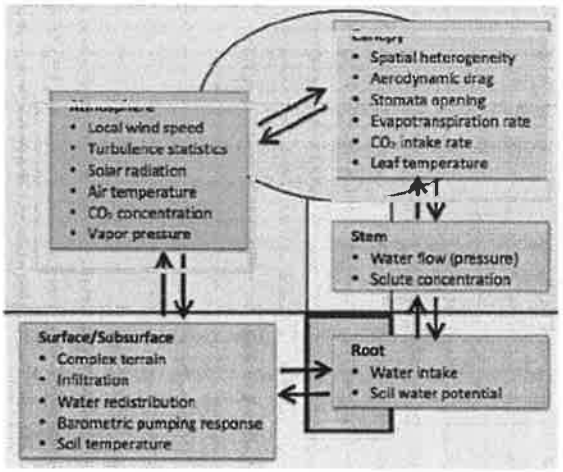
Urban Climatic Mapping Studies for Marina Bay, Singapore

- Urban Climatic Guides In Relation To Urban Geometry For Marina Bay Area

Urban morphology modifications and parameters	Air temperature characteristics (in urban scale)	Wind ventilation characteristics (in urban scale)	Outdoor thermal comfort (in urban scale)
Additional greenery at street level	Reduction of air temperature during daytime benefits from greenery effect	No significant changes. Requires micro-scale study.	Improve outdoor thermal comfort condition.
			
Landscape replacement scheme: -Additional sky garden and roof garden - Staggered building geometry - Increase building height	Reduction of air temperature during daytime benefits from greenery effect and building shading	Higher wind speed at and around the staggered level	Improve outdoor thermal comfort condition.
			

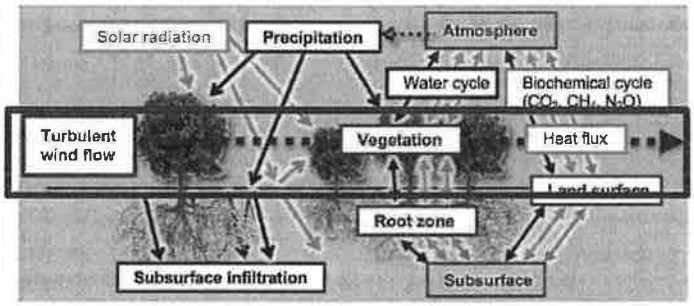
Research Challenge - Vegetation/atmosphere coupling

Multi-Physics Plant-Air-Soil Model Framework (MPAPS)



Developing the comprehensive framework for **canopy model**; then coupling with **atmosphere model**

Coupling between atmosphere, vegetation & substrate
Solar radiation – precipitation – Wind flow



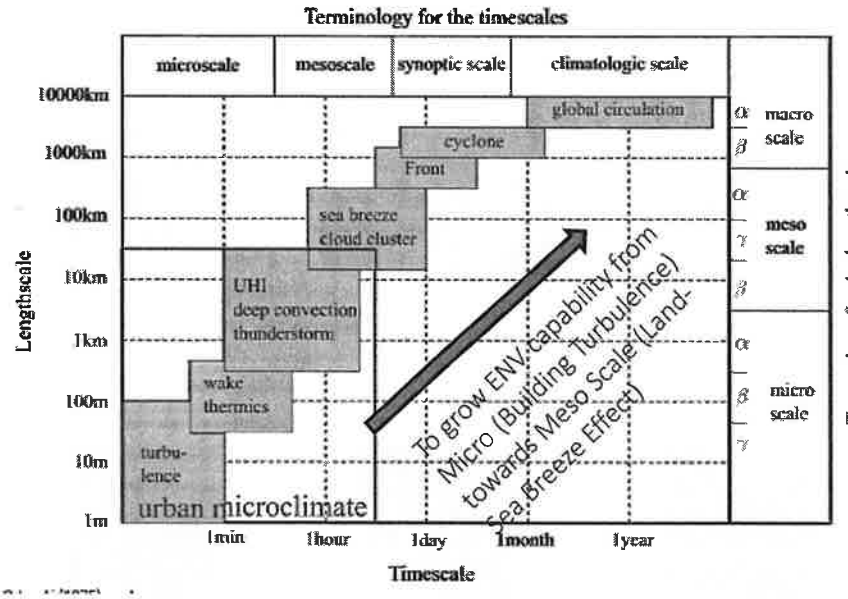
Development on the vegetation module: Interaction of turbulent wind flow and heat flux

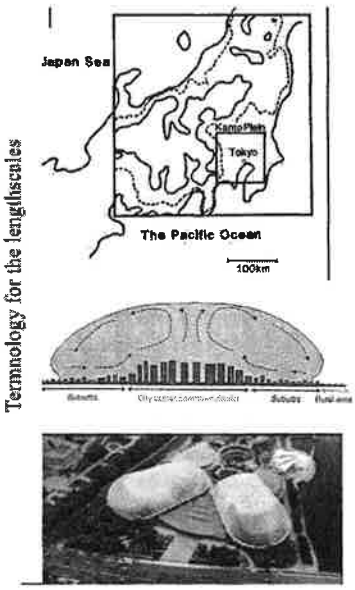
Rod Linn, Urban flow, Wind Energy and Fire Modelling, Urban Physics Winter School, 2015, Courtesy by LANL

Research Challenges: Multi-Scales CFD on UM

- Spatial and temporal scales of atmospheric phenomena – Meso Vs. micro

Background: the scales of weather





Bert Blocken, 50 years of Computational Wind Engineering: Past, present and future, Journal of Wind Engineering and Industrial Aerodynamics, 2014, (129) 69 – 102

Remarks

- Good prediction and decision making for environmental fluid mechanics modeling to achieve urban planning and sustainability needs:
  - Bottom up approach : details urban geometry elements coupled with multi-physics CFD model
  - Top down approach : accurate weather prediction coupled with multi-scale CFD model
- Large scale super computing resources is required for billion cells environmental flow simulation for CSI Singapore

CSI – City State Island

Day 4 (11:00am – 5:00pm)

- CFD Assignment
  - Hands-on Assignment 1 – Use CFD simulation tool (ANSYS-FLUENT) to perform ventilation simulation modelling and analysis in order to ensure good natural ventilation within the functional unit (100%)

Closure

- Understand and apply the working principle of CFD tool to building ventilation
- User/Reviewer should aware the various capabilities and limitations of CFD tool for room air prediction
  - Differential equations
  - Turbulence models
  - Numerical Method
  - Boundary conditions
  - Computational Grid
- If utilized correctly, CFD can be useful tool for room air movement prediction; otherwise it is merely a toy!

Acknowlegement

Acknowledgement

- Prof. Arun Mujumdar, McGill University
- Prof. Hazim Awbi, Reading University
- Prof. Chen Qingyan, Purdue University
- Prof. Zhiqiang (John) Zhai, University of Colorado

References

Literature Review

- Nielsen(1973) made the first prediction of room air movement by CFD
- John and Whittle(1992) discussed the status and capabilities of CFD for building airflow prediction.
- Russel and Surendran (2000) also reviewed CFD related building airflow work
- Many more researchers, e.g. Chen, Q., Awbi, H.B., Murakami, S. etc

Reading Text

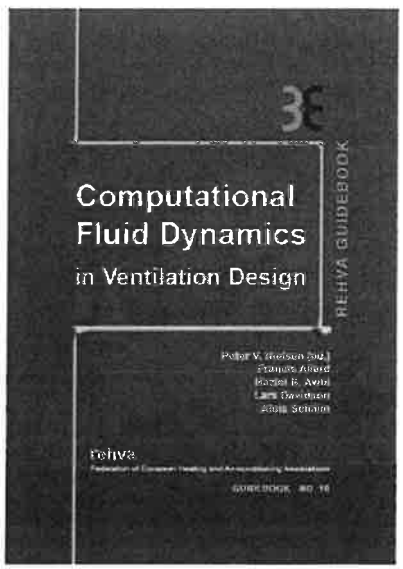
1. Nielsen, P.V. et al, 2007, Computational Fluid Dynamics in Ventilation Design, Brussels, Belgium : REHVA/Federation of European Heating and Air-conditioning Associations, Chp. 1-7
2. Awbi, H.B. 2003, Ventilation of Buildings, 2nd edition, Taylor & Francis, London, Chp. 8
3. Versteeg, H.K. and Malalasekera, W. 1995, An introduction to Computational Fluid Dynamics – The finite volume method, Longman Scientific & technical, Longman Ltd, England

References

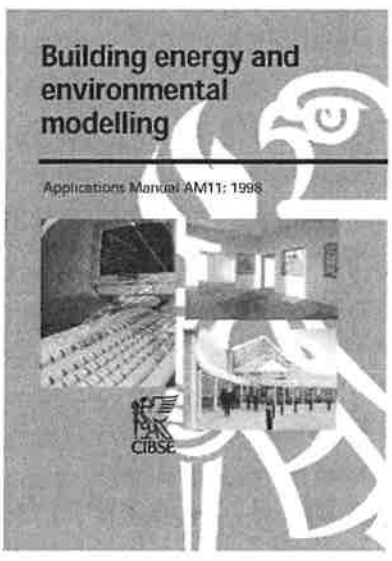
1. Jones, P.J. and Whittle, G.E. 1992. CFD for Building Airflow Prediction – Current Status and Capabilities , Building and Environment, Vol. 27, No.3, pp.321-338
2. Russel, M.B. and Surendran, P.N. 2000. Use of CFD to Aid Studies of Room Air Distribution: A Review of Some Recent Work. Proc. CIBSE A: Building Serv. Res. Technol. 21(4) 241-247
3. Sagaut, P. 2002, Large Eddy Simulation for Incompressible Flow, Springer Verlag
4. Hanjalic, K. 1999, Second moment turbulence closures for CFD: Needs and Prospects, Int. J. Heat and Fluid Flow, Vol. 12, pp. 667-697
5. Chen, Q and Jiang, Z. 1992, Significant Questions in Predicting Room Air Motion, ASHRAE Transaction, Vol. 98, Pt. 1, pp. 929-939
6. Murakami, S., Kato, S. and Zeng, J. 1995, Development of a Computational Thermal Manikin – CFD Analysis of Thermal Environment around Human Body, Proc. Tsinghua HVAC 95, Beijing, Vol. 2, pp. 349-354
7. Menter, F.R., 1994, Two equation eddy-viscosity turbulence models for engineering applications, AIAA, Vol. 32, pp. 1598-1605
8. Patankar, S.V., 1980, Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp.
9. AIAA, 1998, AIAA Guide for the verification and validation of computational fluid dynamics simulations, AIAA G-077-1998
10. Roache, P.J., 1998, Verification and Validation in Computational Science and engineering, Hermosa Publishers, Albuquerque

Modelling Guide Books

REHVA CFD Guidebook



CIBSE Application Manual 11 – Under revision



CFD Modelling Guideline – Selected Publication

Best Paper Award from journal in 2012

Contents lists available at SciVerse ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv



CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters

R. Ramponi [a,b,*], B. Blocken [b]

[a] Building Environment Science & Technology Department, Politecnico di Milano, via Bonardi 3, 20133 Milano, Italy
[b] Building Physics and Services, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ARTICLE INFO

Article history:
Received 11 November 2011
Received in revised form
10 January 2012
Accepted 11 January 2012

Keywords:
Computational Fluid Dynamics (CFD)
Building
Natural ventilation
Air flow
Sensitivity study
Parametric study

ABSTRACT

Accurate CFD simulation of coupled outdoor wind flow and indoor air flow is essential for the design and evaluation of natural cross-ventilation strategies for buildings. It is widely recognized that CFD simulations can be very sensitive to the large number of computational parameters that have to be set by the user. Therefore, detailed and generic sensitivity analyses of the impact of these parameters on the simulation results are important to provide guidance for the execution and evaluation of future CFD studies. A detailed review of the literature indicates that there is a lack of extensive generic sensitivity studies for CFD simulation of natural cross-ventilation. In order to provide such a study, this paper presents a series of coupled 3D steady RANS simulations for a generic isolated building. The CFD simulations are validated based on detailed wind tunnel experiments with Particle Image Velocimetry. The impact of a wide range of computational parameters is investigated, including the size of the computational domain, the resolution of the computational grid, the inlet turbulent kinetic energy profile of the atmospheric boundary layer, the turbulence model, the order of the discretization schemes and the iterative convergence criteria. Specific attention is given to the problem of oscillatory convergence that was observed during some of these coupled CFD simulations. Based on this analysis, the paper identifies the most important parameters. The intention is to contribute to improved accuracy, reliability and evaluation of coupled CFD simulations for cross-ventilation assessment.

© 2012 Elsevier Ltd. All rights reserved.

48 *Ramponi & Blocken, Building and Environment, Vol. 55, pg. 34 - 45, 2012*

Chapter 1: Introduction to Using ANSYS Fluent in ANSYS Workbench: Fluid Flow and Heat Transfer in a Mixing Elbow

This tutorial is divided into the following sections:

1.1. Introduction
1.2. Prerequisites
1.3. Problem Description
1.4. Setup and Solution
1.5. Summary

1.1. Introduction

This tutorial illustrates using ANSYS Fluent fluid flow systems in ANSYS Workbench to set up and solve a three-dimensional turbulent fluid-flow and heat-transfer problem in a mixing elbow. It is designed to introduce you to the ANSYS Workbench tool set using a simple geometry. Guided by the steps that follow, you will create the elbow geometry and the corresponding computational mesh using the geometry and meshing tools within ANSYS Workbench. You will use ANSYS Fluent to set up and solve the CFD problem, then visualize the results in ANSYS Fluent postprocessing tool.
Some capabilities of ANSYS Workbench (for example, duplicating fluid flow systems, connecting systems, and comparing multiple data sets) are also examined in this tutorial.

This tutorial demonstrates how to do the following:

- Launch ANSYS Workbench.
- Create a Fluent fluid flow analysis system in ANSYS Workbench.
- Create the elbow geometry using ANSYS DesignModeler.
- Create the computational mesh for the geometry using ANSYS Meshing.
- Set up the CFD simulation in ANSYS Fluent, which includes:
  - Setting material properties and boundary conditions for a turbulent forced-convection problem.
  - Initiating the calculation with residual plotting.
  - Calculating a solution using the pressure-based solver.
  - Examining the flow and temperature fields using ANSYS Fluent .
- Create a copy of the original Fluent fluid flow analysis system in ANSYS Workbench.
- Change the geometry in ANSYS DesignModeler, using the duplicated system.
- Regenerate the computational mesh.
- Recalculate a solution in ANSYS Fluent.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Prerequisites

This tutorial assumes that you have little to no experience with ANSYS Workbench, ANSYS DesignModeler, ANSYS Meshing, ANSYS Fluent, and so each step will be explicitly described.

1.3. Problem Description

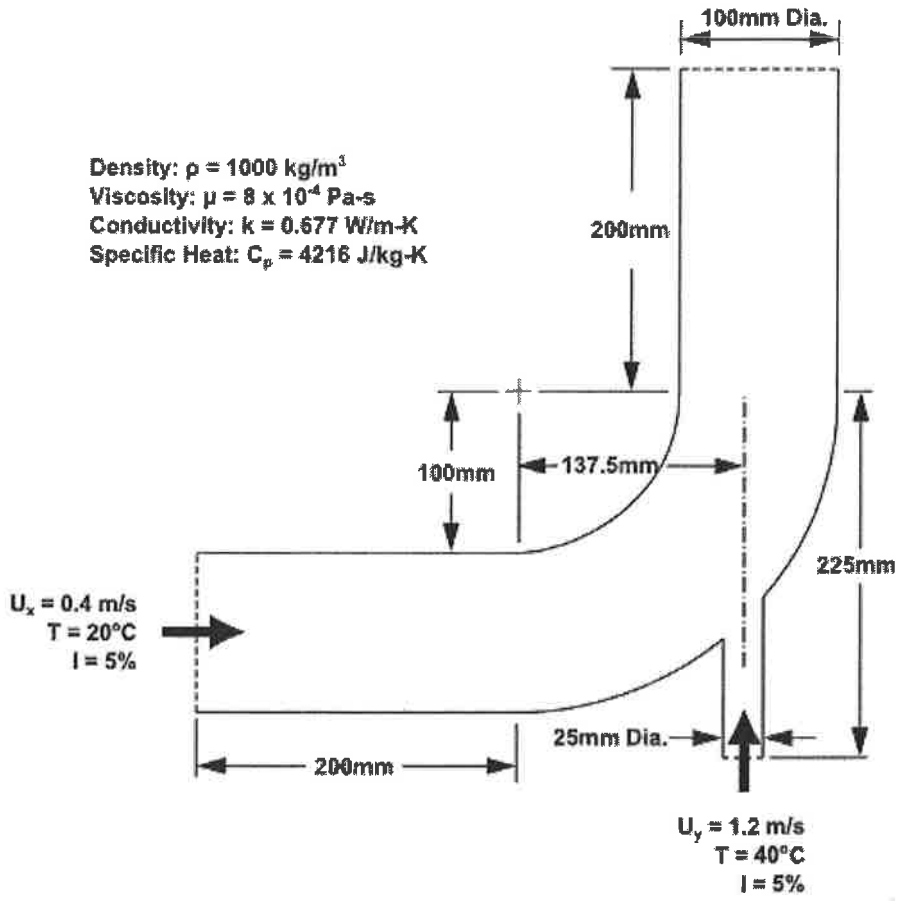
The problem to be considered is shown schematically in Figure 1.1: Problem Specification. A cold fluid at 293.15 K flows into the pipe through a large inlet and mixes with a warmer fluid at 313.15 K that enters through a smaller inlet located at the elbow. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the area of the mixing region in order to properly design the junction.

Note

Because the geometry of the mixing elbow is symmetric, only half of the elbow must be modeled.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: Problem Specification



1.4. Setup and Solution

To help you quickly identify graphical user interface items at a glance and guide you through the steps of setting up and running your simulation, the ANSYS Fluent Tutorial Guide uses several type styles and mini flow charts.

The following sections describe the setup and solution steps for this tutorial:

1.4.1. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench
1.4.2. Creating the Geometry in ANSYS DesignModeler
1.4.3. Meshing the Geometry in the ANSYS Meshing Application
1.4.4. Setting Up the CFD Simulation in ANSYS Fluent
1.4.5. Solving
1.4.6. Displaying Results in ANSYS Fluent
1.4.7. Duplicating the Fluent-Based Fluid Flow Analysis System
1.4.8. Changing the Geometry in ANSYS DesignModeler
1.4.9. Updating the Mesh in the ANSYS Meshing Application
1.4.10. Calculating a New Solution in ANSYS Fluent

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1. Creating a Fluent Fluid Flow Analysis System in ANSYS Workbench

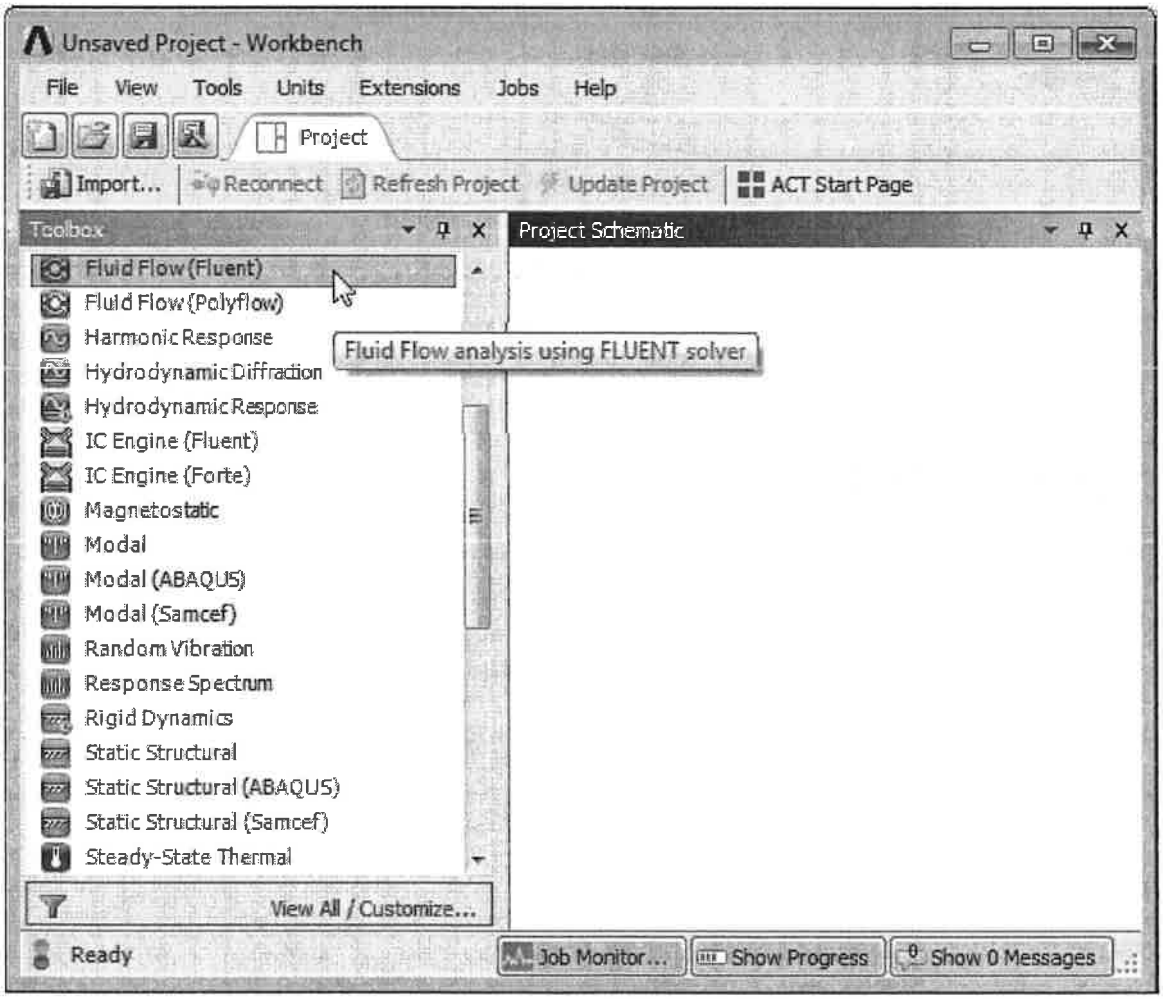
In this step, you will start ANSYS Workbench, create a new Fluent fluid flow analysis system, then review the list of files generated by ANSYS Workbench.

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 17.0 > Workbench 17.0** to start A new ANSYS Workbench session.

   This displays the ANSYS Workbench application window, which has the **Toolbox** on the left and the **Project Schematic** to its right. Various supported applications are listed in the **Toolbox** and the components of the analysis system will be displayed in the **Project Schematic**.

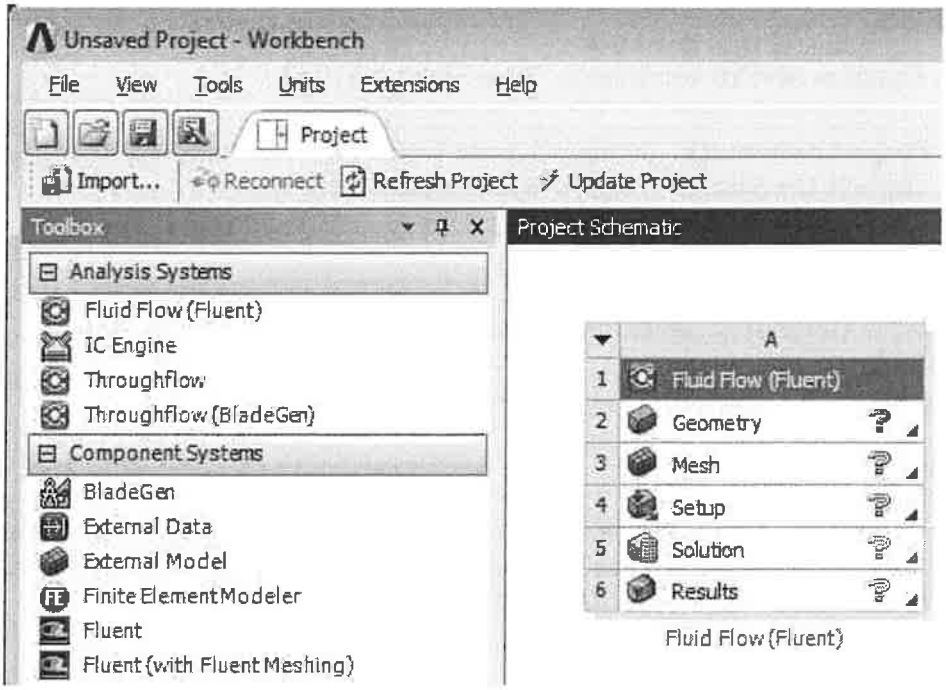
2. Create a new Fluent fluid flow analysis system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the **Toolbox**.

Figure 1.2: Selecting the Fluid Flow (Fluent) Analysis System in ANSYS Workbench



Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.3: ANSYS Workbench with a New Fluent-Based Fluid Flow Analysis System



3. Name the analysis.

   a. Double-click the **Fluid Flow (Fluent)** label underneath the analysis system (if it is not already highlighted).

   b. Enter elbow for the name of the analysis system.

4. Save the project.

   a. In ANSYS Workbench, under the **File** menu, select **Save**.

      File → Save

      This displays the **Save As** dialog box, where you can browse to your working folder and enter a specific name for the ANSYS Workbench project.

   b. In your working directory, enter elbow-workbench as the project **File name** and click the **Save** button to save the project. ANSYS Workbench saves the project with a .wbpj extension and also saves supporting files for the project.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.2. Creating the Geometry in ANSYS DesignModeler

1. Start ANSYS DesignModeler.

In the ANSYS Workbench **Project Schematic**, double-click the **Geometry** cell in the elbow fluid flow analysis system. This displays the ANSYS DesignModeler application.

Tip

You can also right-click the **Geometry** cell to display the context menu, then select **New DesignModeler Geometry...**

2. Set the units in ANSYS DesignModeler.

When ANSYS DesignModeler first appears, you should select desired system of length units to work from. For the purposes of this tutorial (where you will create the geometry in millimeters and perform the CFD analysis using SI units) set the units to **Millimeter**.

Units→ Millimeter

3. Create the geometry.

The geometry for this tutorial (Figure 1.1: Problem Specification) consists of a large curved pipe accompanied by a smaller side pipe. ANSYS DesignModeler provides various geometry primitives that can be combined to rapidly create geometries such as this one. You will perform the following tasks to create the geometry:

- Create the bend in the main pipe by defining a segment of a torus.
- Extrude the faces of the torus segment to form the straight inlet and outlet lengths.
- Create the side pipe by adding a cylinder primitive.
- Use the symmetry tool to reduce the model to half of the pipe assembly, thus reducing computational cost.

a. Create the main pipe:

i. Create a new torus for the pipe bend by choosing the **Create → Primitives → Torus** menu item from the menubar.

ii. In the **Details View** for the new torus (**Torus1**), set **Base Y Component** to -1 by clicking the **1** to the right of **FD10, Base Y Component**, entering -1, and pressing **Enter**. This specifies the direction vector from the origin to the center of the circular cross-section at the start of the torus. In the same manner, specify **Angle**; **Inner Radius**; and **Outer Radius** as shown below.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

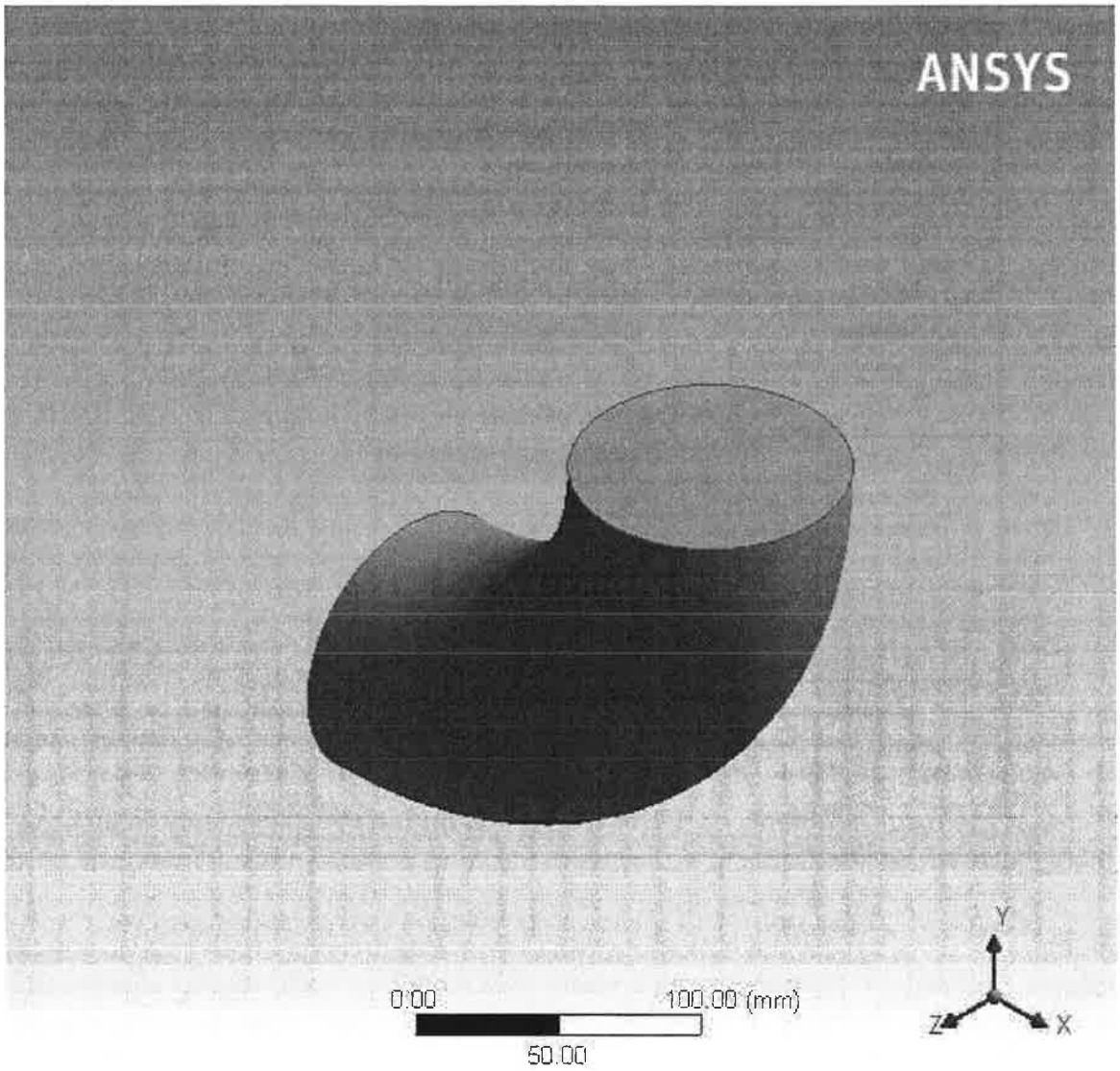
Details View

Details of Torus1	
Torus	Torus1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	0 mm
FD4, Origin Y Coordinate	0 mm
FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
FD6, Axis X Component	0
FD7, Axis Y Component	0
FD8, Axis Z Component	1
Base Definition	Components
FD9, Base X Component	0
FD10, Base Y Component	-1
FD11, Base Z Component	0
FD12, Angle (>0)	90 °
FD13, Inner Radius (>0)	100 mm
FD14, Outer Radius (>0)	200 mm
As Thin/Surface?	No

iii. To create the torus segment, click the **Generate** button 

that is located in the ANSYS DesignModeler toolbar.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The Torus1 item appears in the **Tree Outline** view. If you want to delete this item, you can right-click it and select **Delete** from the context menu that opens.

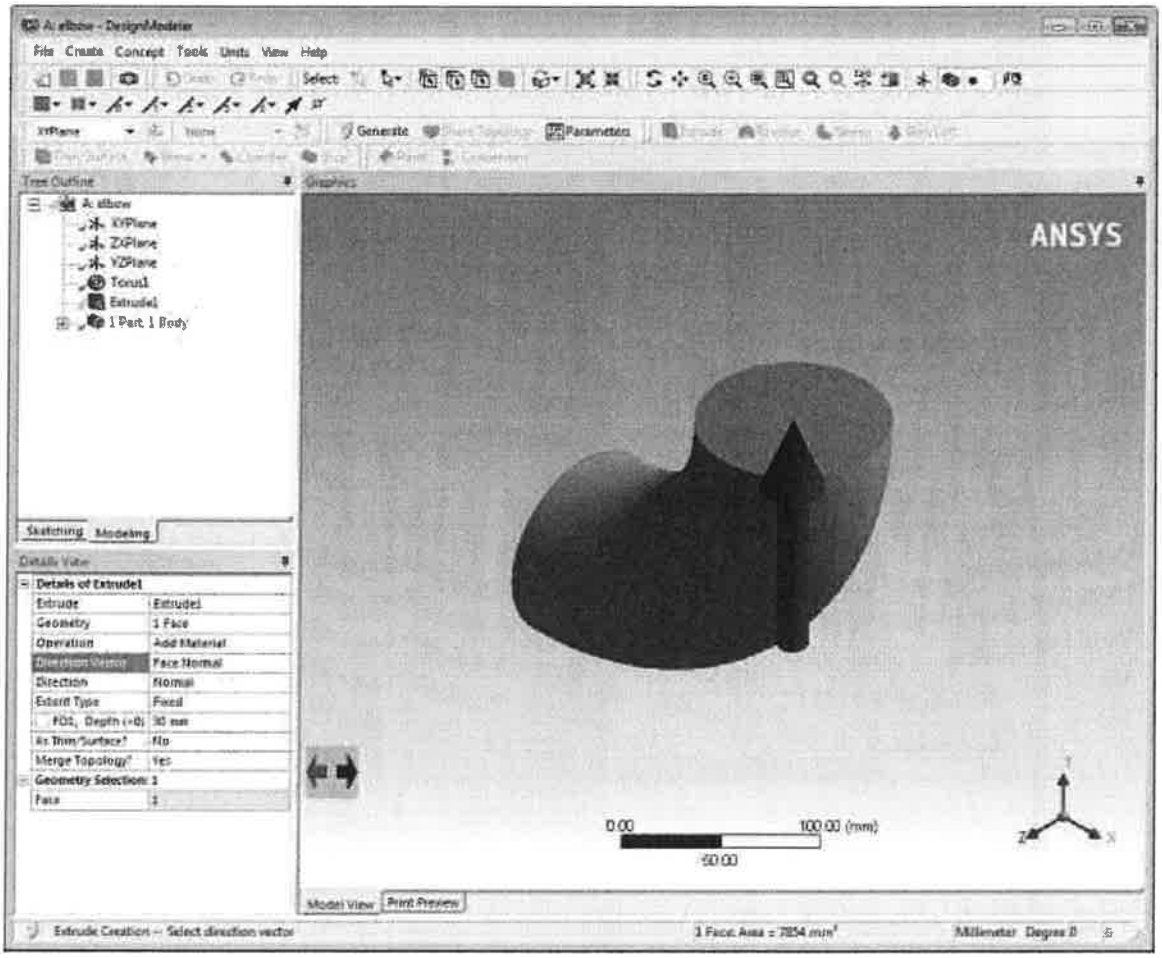
iv. Ensure that the selection filter is set to **Faces**. This is indicated by the **Faces** button appearing depressed in the toolbar and the appearance of the Face selection cursor, when you mouse over the geometry.

v. Select the top face (in the positive Y direction) of the elbow and click the **Extrude** button Extrude from the **3D Features** toolbar.

vi. In the **Details View** for the new extrusion (**Extrude1**), click **Apply** to the right of **Geometry**. This accepts the face you selected as the base geometry of the extrusion.

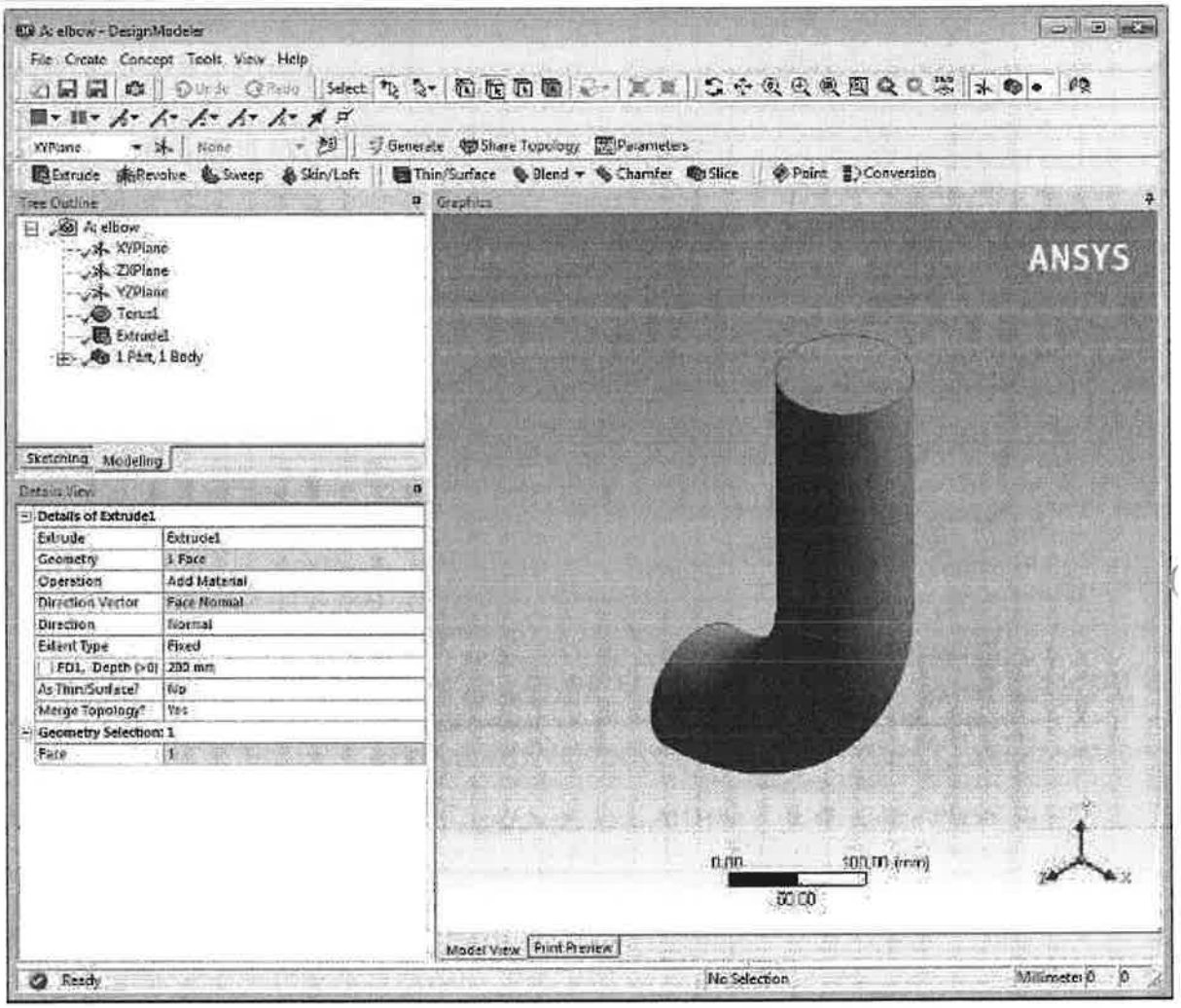
vii. Click **None (Normal)** to the right of **Direction Vector** until **Apply** button appears. Again, ensure that the selection filter is set to **Faces**, select the same face on the elbow to specify that the extrusion will be normal to the face and click **Apply**.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



viii.Enter 200 for **FD1, Depth (>0)** and click **Generate**.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ix. In a similar manner, create an extrusion of the other face of the torus segment to create the 200 mm inlet extension. You will probably find it helpful to rotate the view so that you can easily select the other face of the bend.

You can use the mouse buttons to change your view of the 3D image. The following table describes mouse actions that are available:

Table 1.1: DesignModeler View Manipulation Instructions

Action	Using Graphics Toolbar Buttons and the Mouse
Rotate view (vertical, horizontal)	After clicking the **Rotate** icon, , press and hold the left mouse button and drag the mouse. Dragging side to side rotates the view about the vertical axis, and dragging up and down rotates the view about the horizontal axis.
Translate or pan view	After clicking the **Pan** icon, , press and hold the left mouse button and drag the object with the mouse until the view is satisfactory.
Zoom in and out of view	After clicking the **Zoom** icon, , press and hold the left mouse button and drag the mouse up and down to zoom in and out of the view.

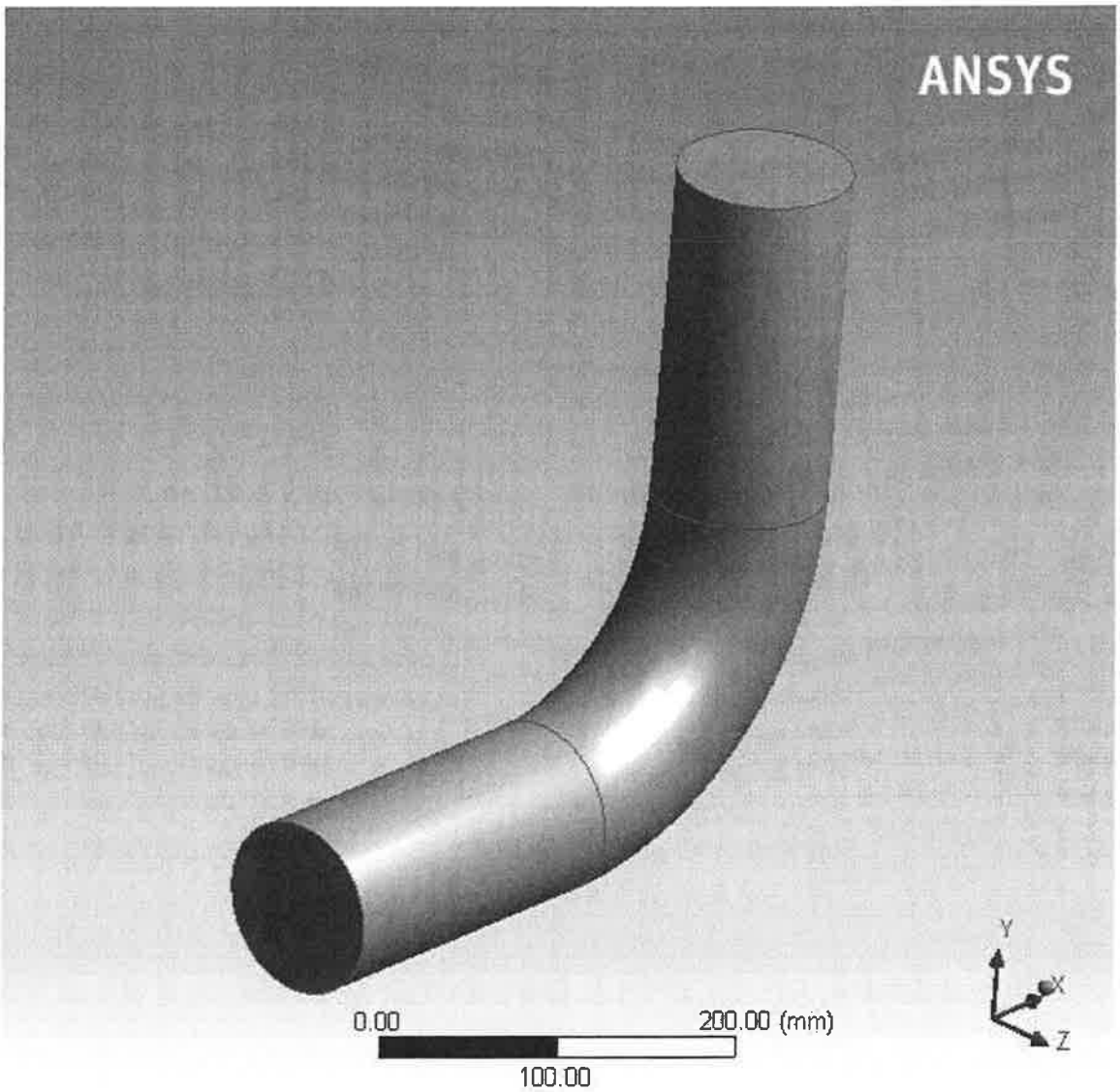
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Action	Using Graphics Toolbar Buttons and the Mouse
Box zoom	After clicking the **Box Zoom** icon, , press and hold the left mouse button and drag the mouse diagonally across the screen. This action will cause a rectangle to appear in the display. When you release the mouse button, a new view will be displayed that consists entirely of the contents of the rectangle.

Clicking the **Zoom to Fit** icon, , will cause the object to fit exactly and be centered in the window.

After entering the extrusion parameters and clicking **Generate**, the geometry should appear as in Figure 1.5: Elbow Main Pipe Geometry.

Figure 1.5: Elbow Main Pipe Geometry



b. Next you will use a cylinder primitive to create the side pipe.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

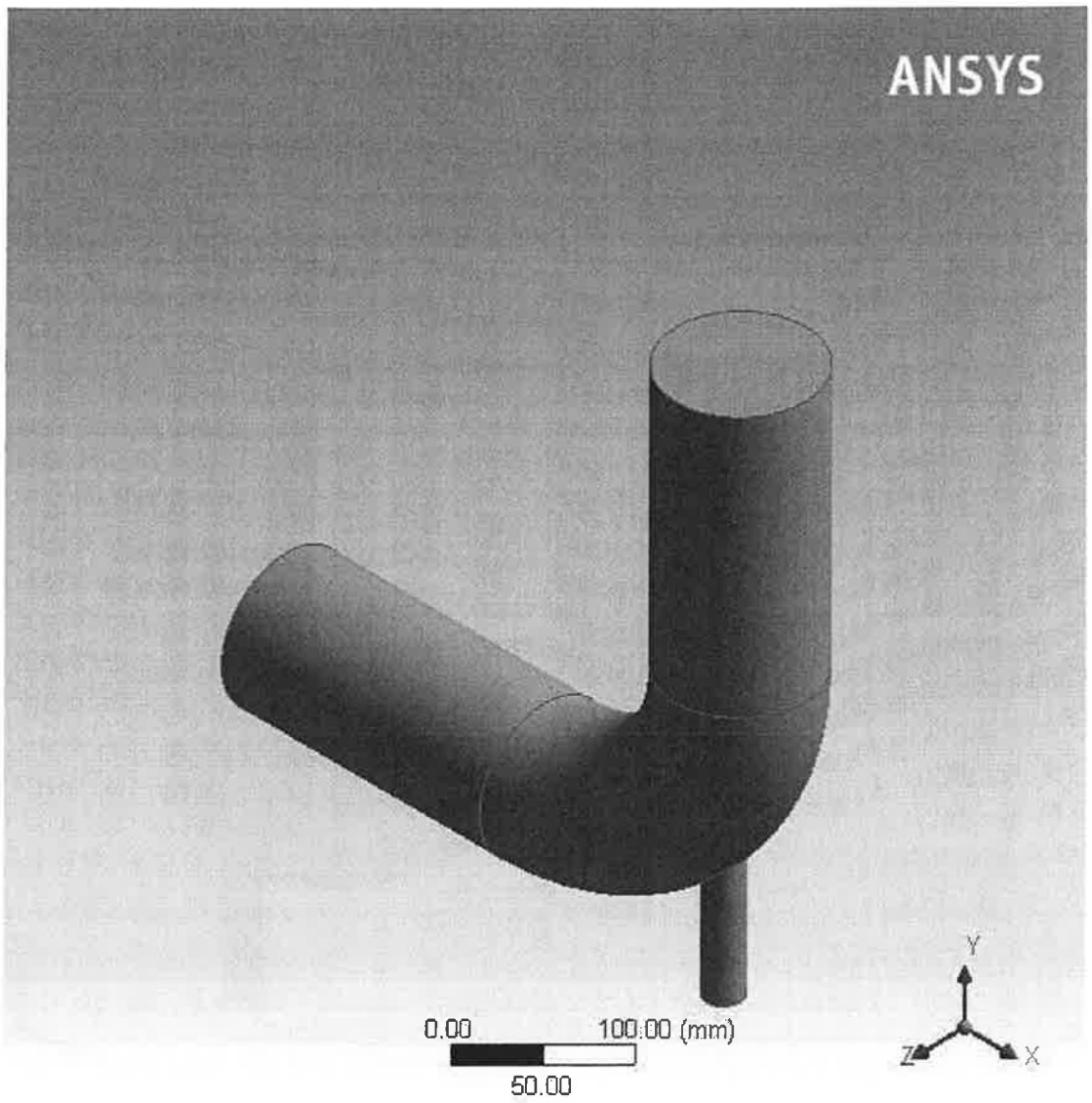
i. Choose **Create → Primitives → Cylinder** from the menubar.

ii. In the **Details View**, set the parameters for the cylinder as follows and click **Generate**:

Tab	Setting	Value
Details of Cylinder1	**BasePlane**	XYPlane
	FD3, Origin X Coordinate	137.5
	FD4, Origin Y Coordinate	-225
	FD5, Origin Z Coordinate	0
	FD6, Axis X Component	0
	FD7, Axis Y Component	125
	FD8, Axis Z Component	0
	FD10, Radius (>0)	12.5

The Origin Coordinates determine the starting point for the cylinder and the Axis Components determine the length and orientation of the cylinder body.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



c. The final step in creating the geometry is to split the body on its symmetry plane which will halve the computational domain.

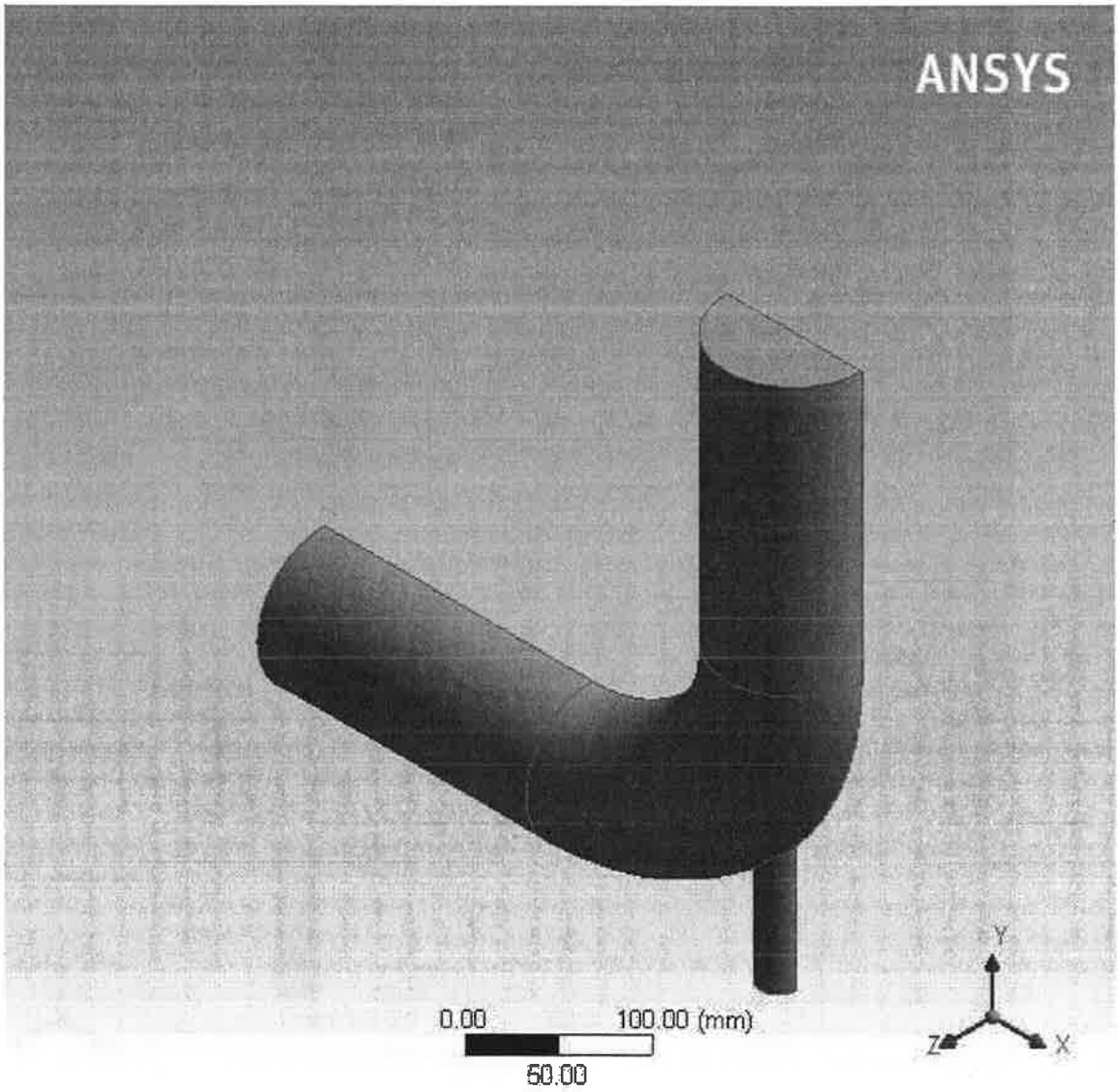
   i. Choose **Tools → Symmetry** from the menu bar.

   ii. Select the **XYPlane** in the **Tree Outline**.

   iii. Click **Apply** next to **Symmetry Plane 1** in the **Details** view.

   iv. Click **Generate**.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The new surface created with this operation will be assigned a symmetry boundary condition in Fluent so that the model will accurately reflect the physics of the complete elbow geometry even though only half of it is meshed.

d. Specify the geometry as a fluid body.

   i. In the **Tree Outline**, open the **1 Part, 1 Body** branch and select **Solid**.

   ii. In the **Details View** of the body, change the name of the **Body** from **Solid** to **Fluid**.

   iii. In the **Fluid/Solid** section, select **Fluid**.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Details View

Details of Body	
Body	Fluid
Volume	2.5159e+006 mm³
Surface Area	1.7636e+005 mm²
Faces	8
Edges	18
Vertices	12
Fluid/Solid	Fluid
Shared Topology Method	Automatic
Geometry Type	DesignModeler

iv. Click **Generate**.

4. Close ANSYS DesignModeler by selecting **File → Close DesignModeler** or by clicking the 'X' icon in the upper right-hand corner. ANSYS Workbench automatically saves the geometry and updates the **Project Schematic** accordingly. The question mark in the **Geometry** cell is replaced by a check mark, indicating that there is a geometry now associated with the fluid flow analysis system.

5. View the list of files generated by ANSYS Workbench by selecting **View → Files**.

Figure 1.6: ANSYS Workbench Files View for the Project After Creating the Geometry

Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		124 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	2 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	designPoint.wbdp		26 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the geometry file (FFF.agdb, where FFF indicates a Fluent-based fluid flow system) to the list of files.

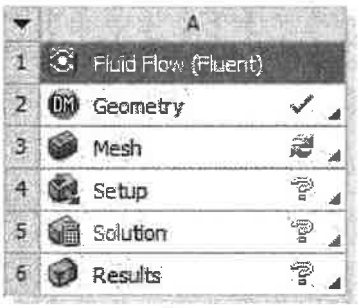
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.3. Meshing the Geometry in the ANSYS Meshing Application

Now that you have created the mixing elbow geometry, you must generate a computational mesh throughout the flow volume. For this section of the tutorial, you will use the ANSYS Meshing application to create a mesh for your CFD analysis, then review the list of files generated by ANSYS Workbench.

Important

Note the **Refresh Required** icon ( ) within the **Mesh** cell for the system. This indicates that the state of the cell requires a refresh and that upstream data has changed since the last refresh or update (such as an update to the geometry). Once the mesh is defined, the state of the **Mesh** cell will change accordingly, as will the state of the next cell in the system, in this case the **Setup** cell.



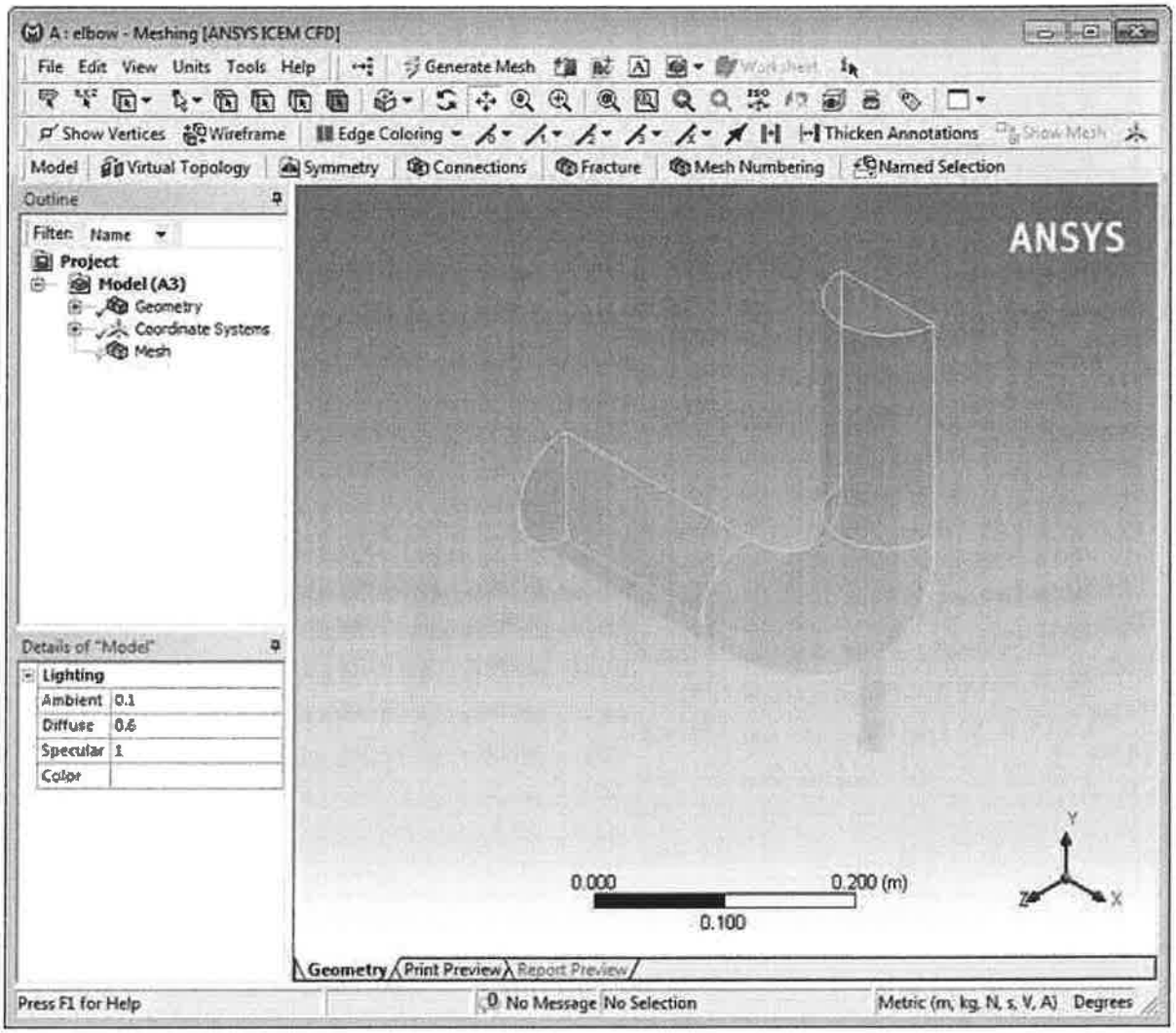
elbow

1. Open the ANSYS Meshing application.

   In the ANSYS Workbench **Project Schematic**, double-click the **Mesh** cell in the elbow fluid flow analysis system (cell A3). This displays the ANSYS Meshing application with the elbow geometry already loaded. You can also right-click the **Mesh** cell to display the context menu where you can select the **Edit...** option.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.7: The ANSYS Meshing Application with the Elbow Geometry Loaded



2. Create named selections for the geometry boundaries.

   In order to simplify your work later on in ANSYS Fluent, you should label each boundary in the geometry by creating named selections for the pipe inlets, the outlet, and the symmetry surface (the outer wall boundaries are automatically detected by ANSYS Fluent).

   a. Select the large inlet in the geometry that is displayed in the ANSYS Meshing application.

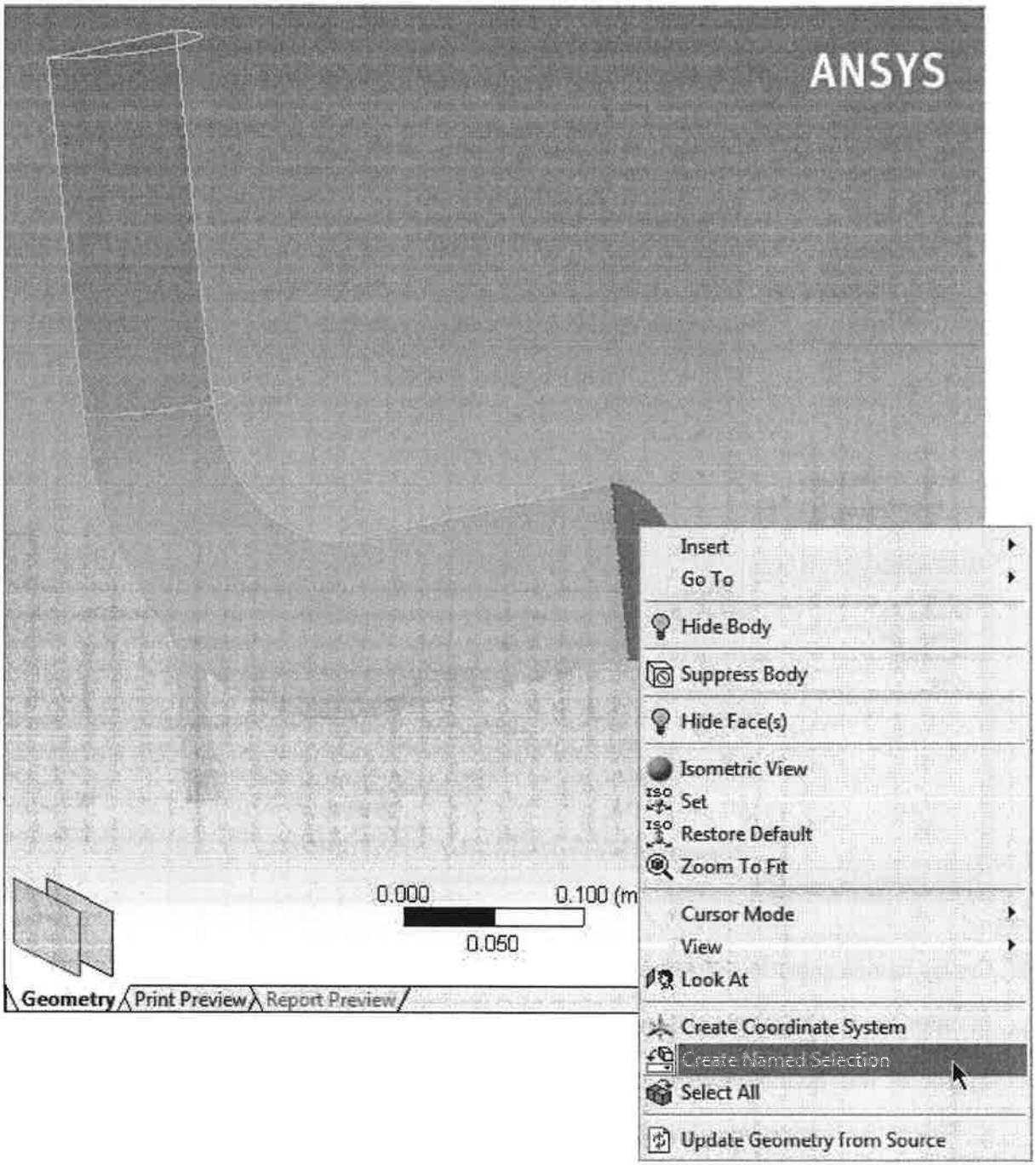
   Tip

   - Use the Graphics Toolbar buttons and the mouse to manipulate the image until you can easily see the pipe openings and surfaces.
   - To select the inlet, the *Single select* ( ) mode must be active.

   b. Right-click and select the **Create Named Selection** option.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

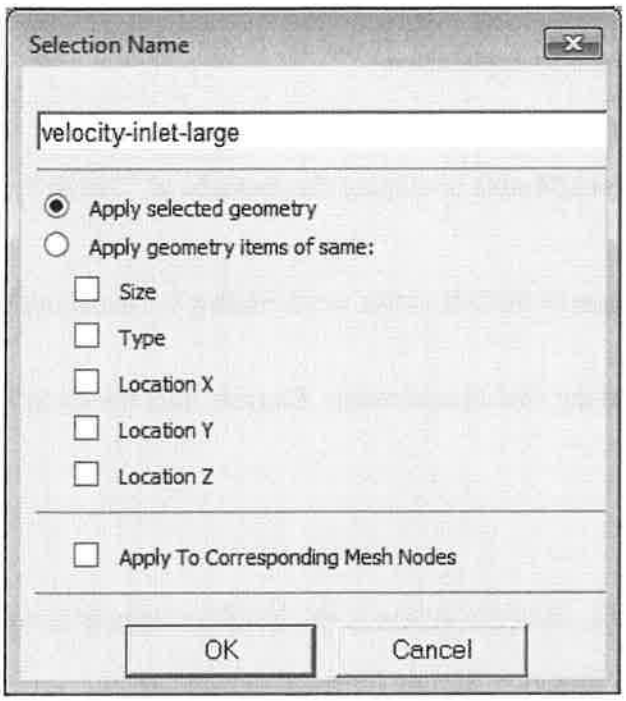
Figure 1.8: Selecting a Face to Name



This displays the **Selection Name** dialog box.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.9: Applying a Name to a Selected Face



c. In the **Selection Name** dialog box, enter velocity-inlet-large for the name and click **OK**.

d. Perform the same operations for:

- The small inlet (velocity-inlet-small)
- The large outlet (pressure-outlet)
- The symmetry plane (symmetry).

The named selections that you have created appear under the **Named Selections** item in the **Outline** view.

3. Create a named selection for the fluid body.

a. Change the selection filter to **Body** in the **Graphics Toolbar** ( )

b. Click the elbow in the graphics display to select it.

c. Right-click, select the **Create Named Selection** option and name the body Fluid.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Set basic meshing parameters for the ANSYS Meshing application.

   For this analysis, you will adjust several meshing parameters to obtain a finer mesh.

   a. In the **Outline** view, select **Mesh** under **Project/Model** to display the **Details of "Mesh"** view below the **Outline** view.

   b. Expand the **Sizing** node by clicking the "+" sign to the left of the word **Sizing** to reveal additional sizing parameters.

      i. Change **Relevance Center** to **Fine** by clicking the default value, **Coarse**, and selecting **Fine** from the drop-down list.

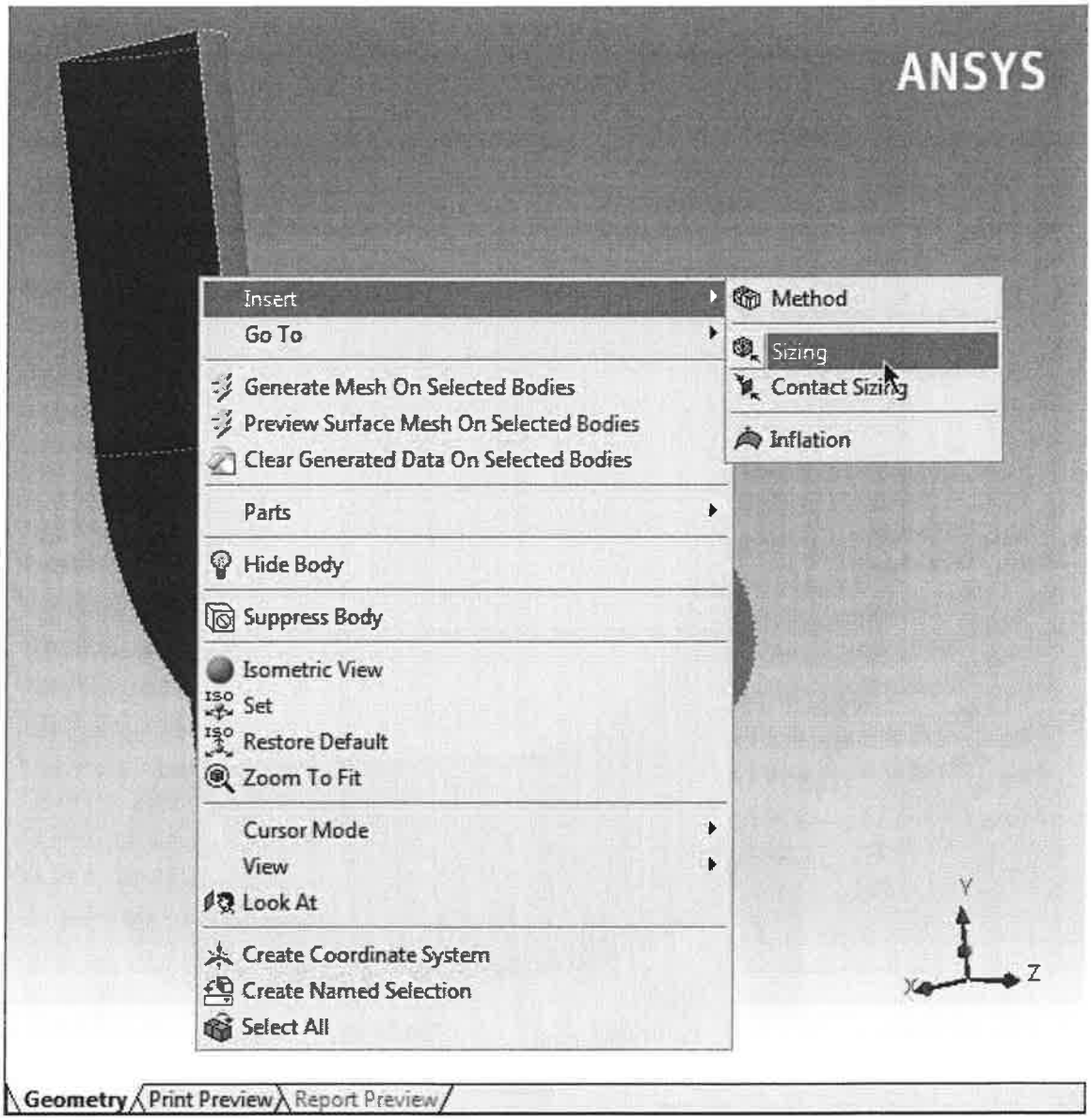
      ii. Change **Smoothing** to **High**

   c. Add a **Body Sizing** control.

      i. With **Mesh** still selected in the **Outline** tree, click the elbow in the graphics display to select it.

      ii. Right click in the graphics area and select **Insert → Sizing** from the context menu.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



*A new **Body Sizing** entry appears under **Mesh** in the project **Outline** tree*

iii. In the **Outline** tree, click the new **Body Sizing** control.

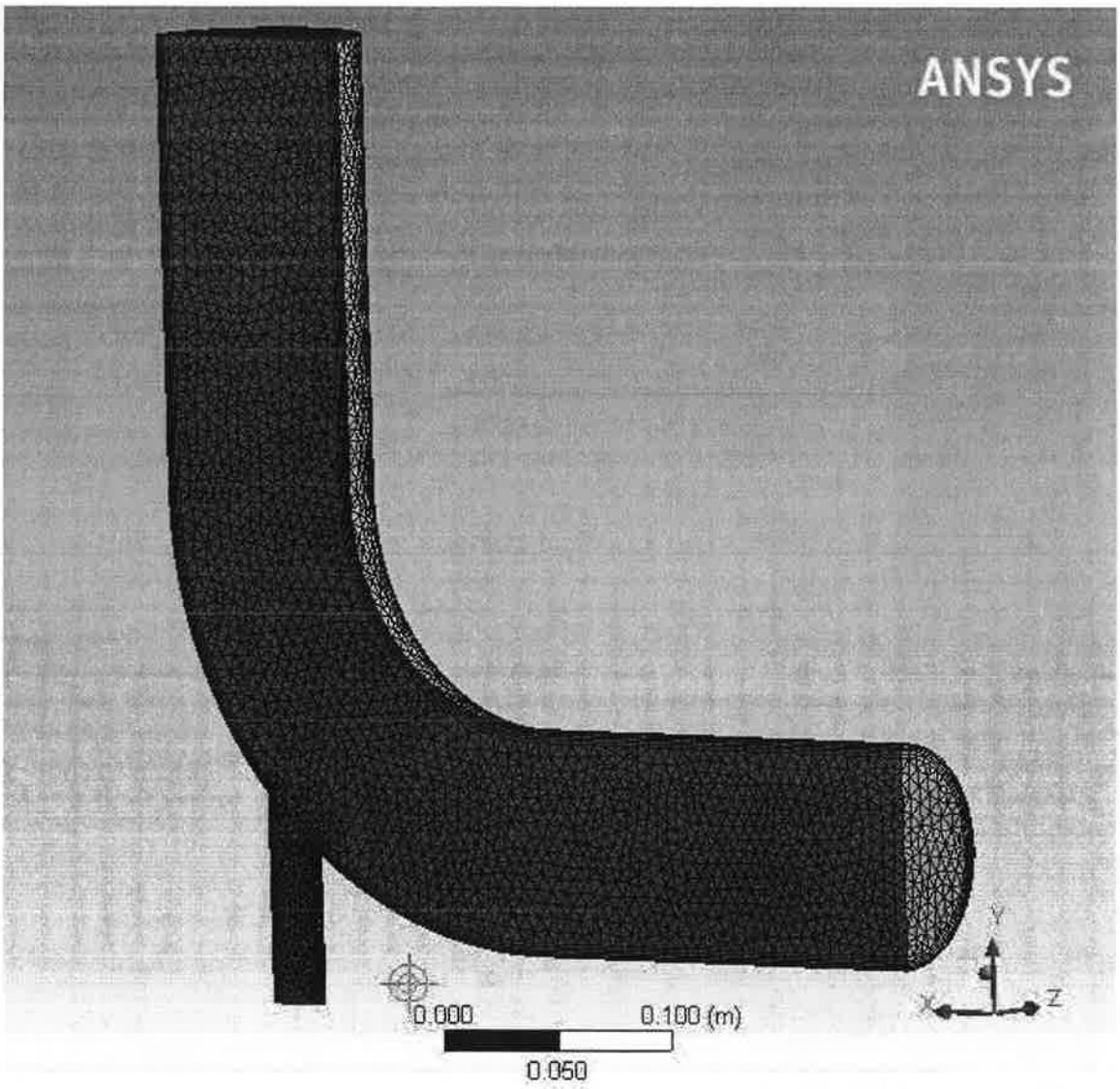
iv. Enter 6e-3 for **Element Size** and press **Enter**.

d. Click again **Mesh** in the **Outline** view and in the **Details of "Mesh"** view, expand the **Inflation** node to reveal additional inflation parameters. Change **Use Automatic Inflation** to **Program Controlled**.

5. Generate the mesh.

Right-click **Mesh** in the project **Outline** tree, and select **Update** in the context menu.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important

Using the **Generate Mesh** option creates the mesh, but does not actually create the relevant mesh files for the project and is optional if you already know that the mesh is acceptable. Using the **Update** option automatically generates the mesh, creates the relevant mesh files for your project, and updates the ANSYS Workbench cell that references this mesh.

Note

Once the mesh is generated, you can view the mesh statistics by opening the **Statistics** node in the **Details of "Mesh"** view. This will display information such as the number of nodes and the number of elements.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Close the ANSYS Meshing application.

   You can close the ANSYS Meshing application without saving it because ANSYS Workbench automatically saves the mesh and updates the **Project Schematic** accordingly. The **Refresh Required** icon in the **Mesh** cell has been replaced by a check mark, indicating that there is a mesh now associated with the fluid flow analysis system.

7. View the list of files generated by ANSYS Workbench.

   View → Files

Figure 1.11: ANSYS Workbench Files View for the Project After Mesh Creation

Files

	A	B	C	D	E	
1	Name	Cell ID	Size	Type	Date Modified	Location
2	elbow-workbench.wbpj		122 KB	ANSYS Project File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01
3	FFF.agdb	A2	1 MB	Geometry File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
4	FFF.msh	A3	8 MB	Fluent Mesh File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
5	FFF.mshdb	A3	2 MB	Mesh Database Files	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo
6	designPoint.wbdp		25 KB	Design Point File	XX/XX/20XX 0:00:00 AM	C:\Tutorial_01\elbow-wo

Note the addition of the mesh files (FFF.msh and FFF.mshdb) to the list of files. The FFF.msh file is created when you update the mesh, and the FFF.mshdb file is generated when you close the ANSYS Meshing application.

1.4.4. Setting Up the CFD Simulation in ANSYS Fluent

Now that you have created a computational mesh for the elbow geometry, you will set up a CFD analysis using ANSYS Fluent, then review the list of files generated by ANSYS Workbench.

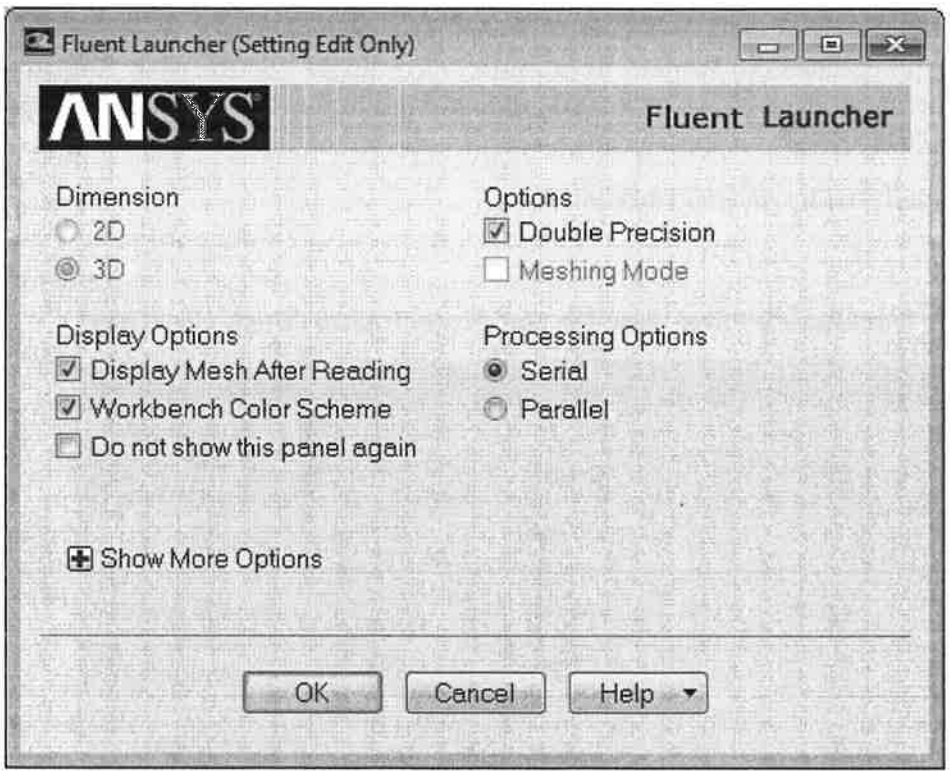
1.4.4.1. Starting ANSYS Fluent

1. In the ANSYS Workbench **Project Schematic**, double-click the **Setup** cell in the elbow fluid flow analysis system. You can also right-click the **Setup** cell to display the context menu where you can select the **Edit...** option.

   When ANSYS Fluent is first started, the Fluent Launcher is displayed, enabling you to view and/or set certain ANSYS Fluent start-up options.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.12: Fluent Launcher



2. Ensure that the proper options are enabled.

   a. Ensure that **Serial** from the **Processing Options** list is enabled.

   b. Select **Double Precision** under **Options**.

   c. Ensure that the **Display Mesh After Reading** and **Workbench Color Scheme** options are enabled.

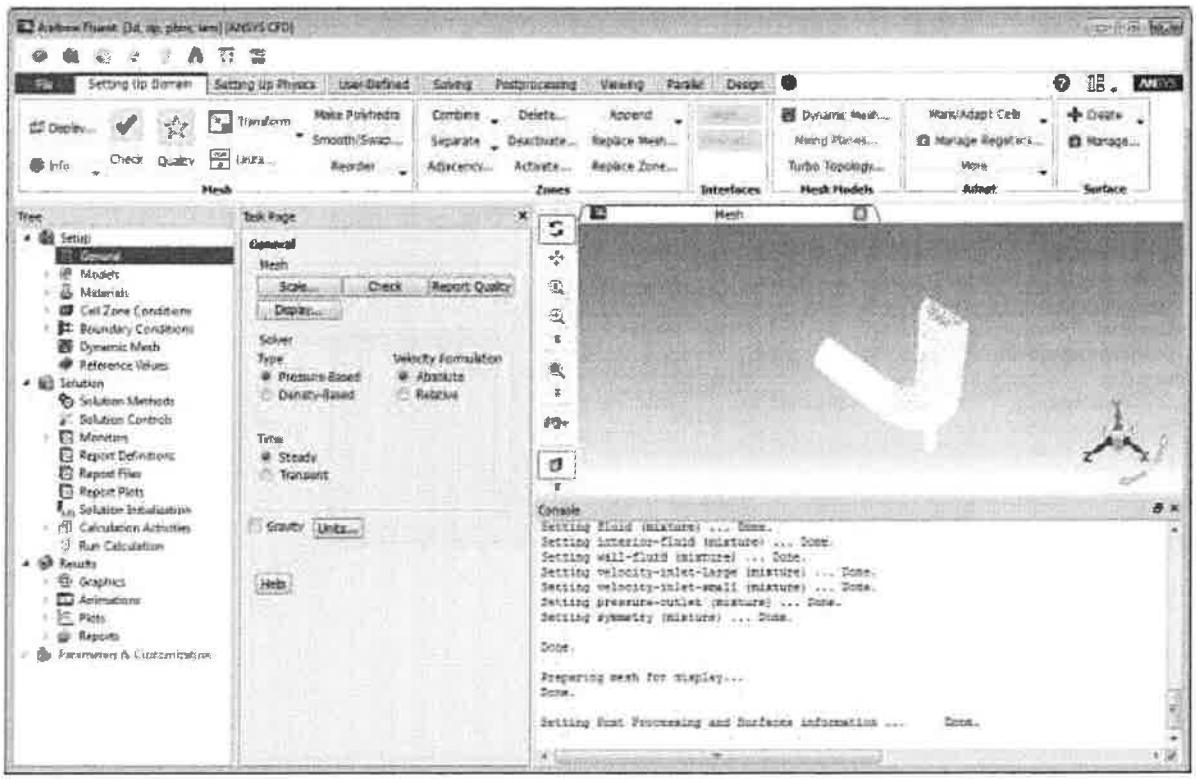
Note

An option is enabled when there is a check mark in the check box, and disabled when the check box is empty. To change an option from disabled to enabled (or vice versa), click the check box or the text.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

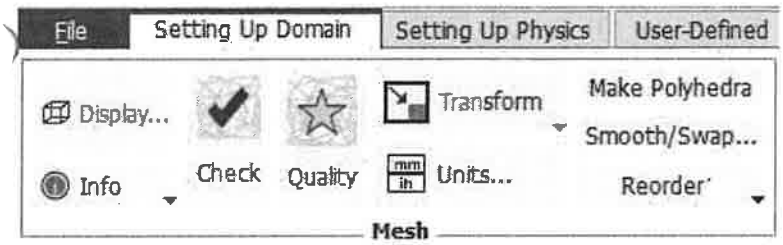
3. Click **OK** to launch ANSYS Fluent.

Figure 1.13:The ANSYS Fluent Application



1.4.4.2. Setting Up Domain

In this step, you will perform the mesh-related activities using the **Setting Up Domain** ribbon tab (**Mesh** group).



Note

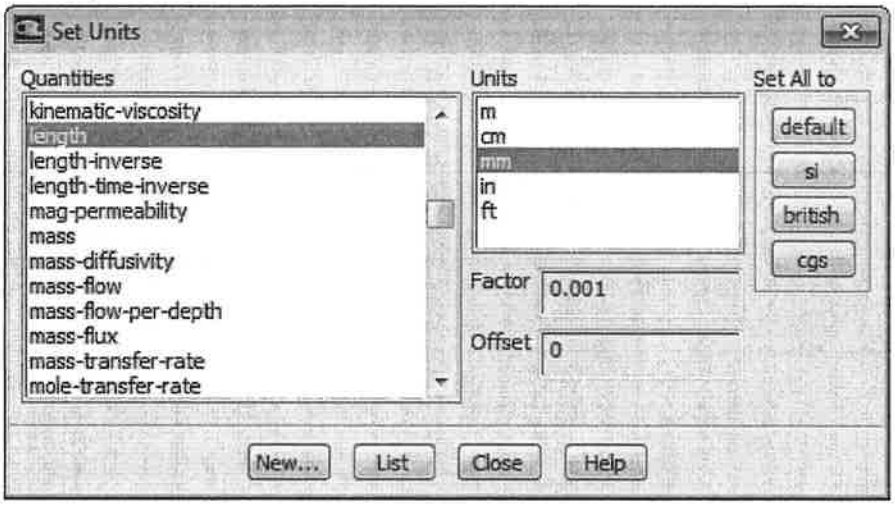
These controls are also available in the **General** task page that can be accessed by clicking the **Setup/General** tree item.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Change the units for length.

   Setting Up Domain → Mesh → Units...

   This displays the **Set Units** dialog box.



   a. Select **length** in the **Quantities** list.

   b. Select **mm** in the **Units** list.

   c. Close the dialog box.

   Note

   Now, all subsequent inputs that require a value based on a unit of length can be specified in millimeters rather than meters.

2. Check the mesh.

   Setting Up Domain → Mesh → Check

   Note

   ANSYS Fluent will report the results of the mesh check in the console.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Domain Extents:
  x-coordinate: min (m) = -2.000000e-01, max (m) = 2.000000e-01
  y-coordinate: min (m) = -2.250000e-01, max (m) = 2.000000e-01
  z-coordinate: min (m) = 0.000000e+00, max (m) = 5.000000e-02
Volume statistics:
  minimum volume (m3): 1.144763e-10
  maximum volume (m3): 5.871098e-08
    total volume (m3): 2.511309e-03
Face area statistics:
  minimum face area (m2): 2.051494e-07
  maximum face area (m2): 3.429518e-05
Checking mesh.....
Done.
```

3. Review the mesh quality.



Setting Up Domain → Mesh → Quality

Note

ANSYS Fluent will report the results of the mesh quality below the results of the mesh check in the console.

```
Mesh Quality:
Minimum Orthogonal Quality = 1.98483e-01
(Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.)
Maximum Ortho Skew = 7.19694e-01
(Ortho Skew ranges from 0 to 1, where values close to 1 correspond to low quality.)
Maximum Aspect Ratio = 2.05643e+01
```

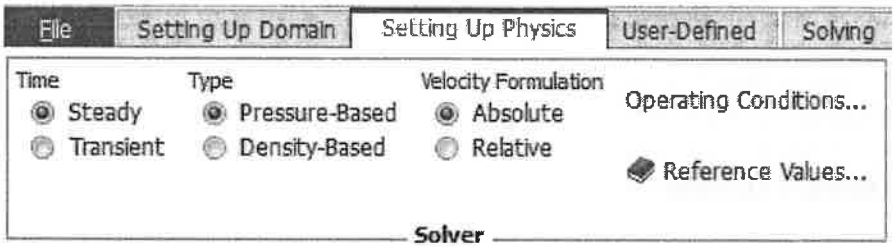
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.4.3. Setting Up Physics

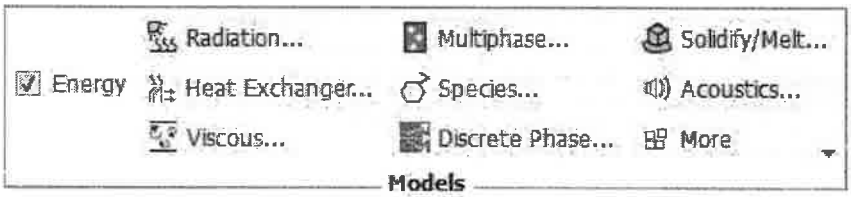
In the steps that follow, you will select a solver and specify physical models, material properties, and zone conditions for your simulation using the **Setting Up Physics** ribbon tab.

1. In the **Solver** group of the **Setting Up Physics** ribbon tab, retain the default selection of the steady pressure-based solver.

   Setting Up Physics → Solver



2. Set up your models for the CFD simulation using the **Models** group of the **Setting Up Physics** ribbon tab.



Note

You can also use the **Models** task page that can be accessed by clicking the **Setup/Models** tree item.

---

   a. Enable heat transfer by activating the energy equation.

      In the **Setting Up Physics** ribbon tab, select **Energy** (**Models** group).

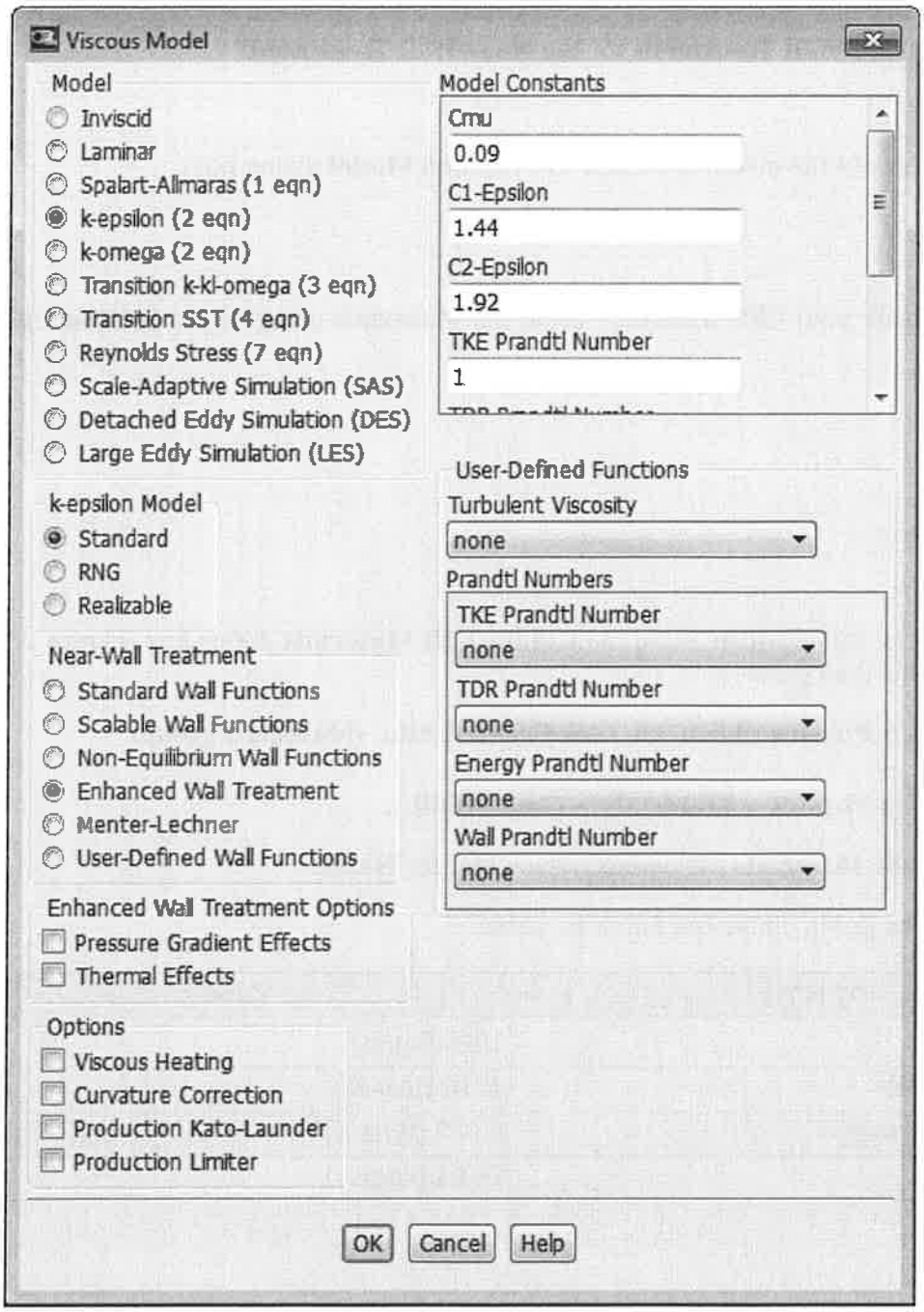
      Setting Up Physics → Models → Energy

   b. Enable the k-ε turbulence model.

      i. In the **Setting Up Physics** ribbon tab, click **Viscous...** (**Models** group).

         Setting Up Physics → Models → Viscous...

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ii. In the **Viscous Model** dialog box, select **k-epsilon** from the **Model** list.

iii. Use the default **Standard** from the **k-epsilon Model** group.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iv. Select **Enhanced Wall Treatment** for the **Near-Wall Treatment**.

v. Click **OK** to accept the model and close the **Viscous Model** dialog box.

3. Set up the materials for your CFD simulation using the **Materials** group of the **Setting Up Physics** ribbon tab.



Create a new material called **water** using the **Create/Edit Materials** dialog box (Figure 1.14: The Create/Edit Materials Dialog Box).

a. In the **Setting Up Physics** ribbon tab, click **Create/Edit...** (**Materials** group).

Setting Up Physics → Materials → Create/Edit...

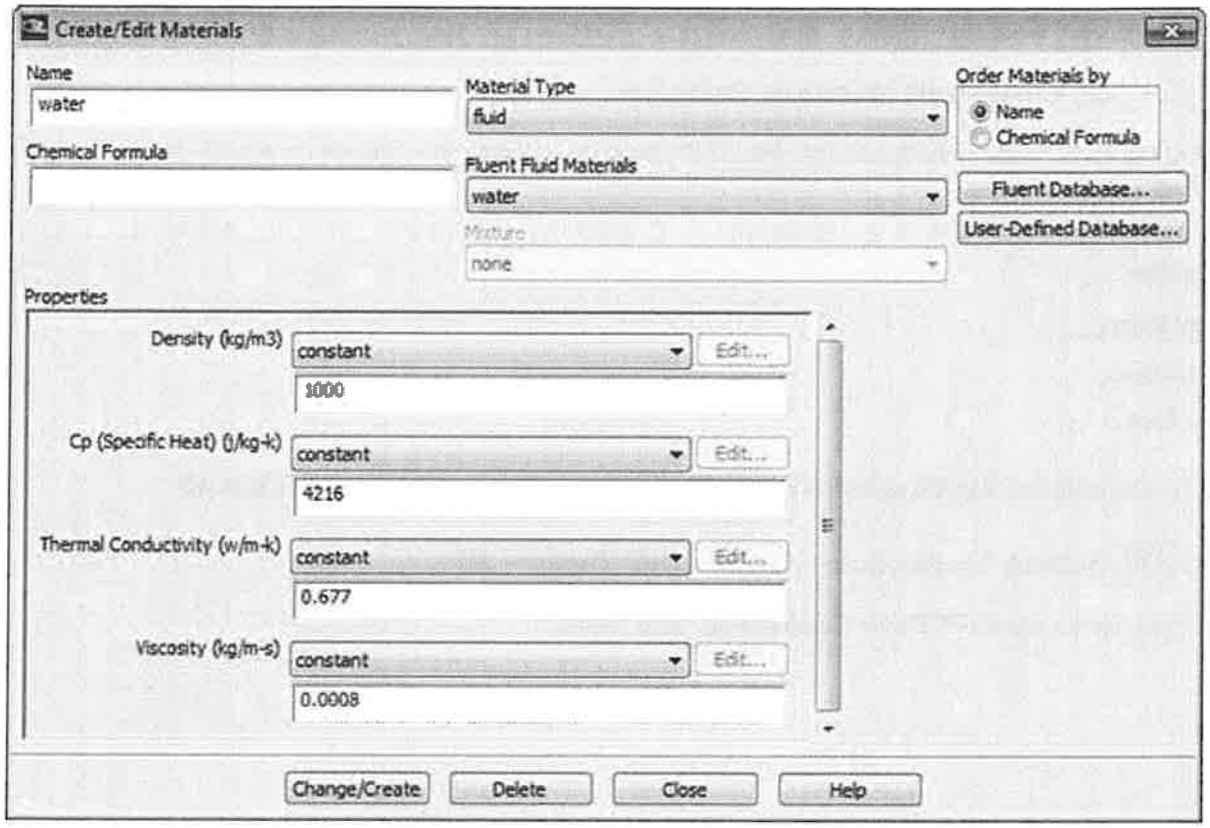
b. In the **Create/Edit Materials** dialog box, type water for **Name**.

c. In the **Properties** group, enter the following values:

Property	Value
Density	1000 [kg/m^3]
c_p (Specific Heat)	4216 [J/kg-K]
Thermal Conductivity	0.677 [W/m-K]
Viscosity	8e-04 [kg/m-s]

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

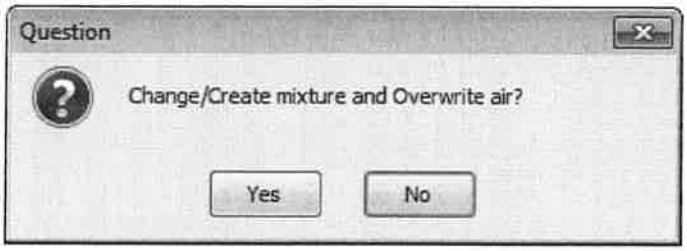
Figure 1.14: The Create/Edit Materials Dialog Box



d. Click **Change/Create**.

Note

A **Question** dialog box will open, asking if you want to overwrite air. Click **No** so that the new material **water** is added to the **Fluent Fluid Materials** list of materials that originally contained only **air**.

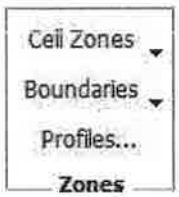


Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Ensure that there are now two materials (water and air) defined locally by examining the **Fluent Fluid Materials** drop-down list.

f. Close the **Create/Edit Materials** dialog box.

4. Set up the cell zone conditions for the CFD simulation using the **Zones** group of the **Setting Up Physics** ribbon tab.

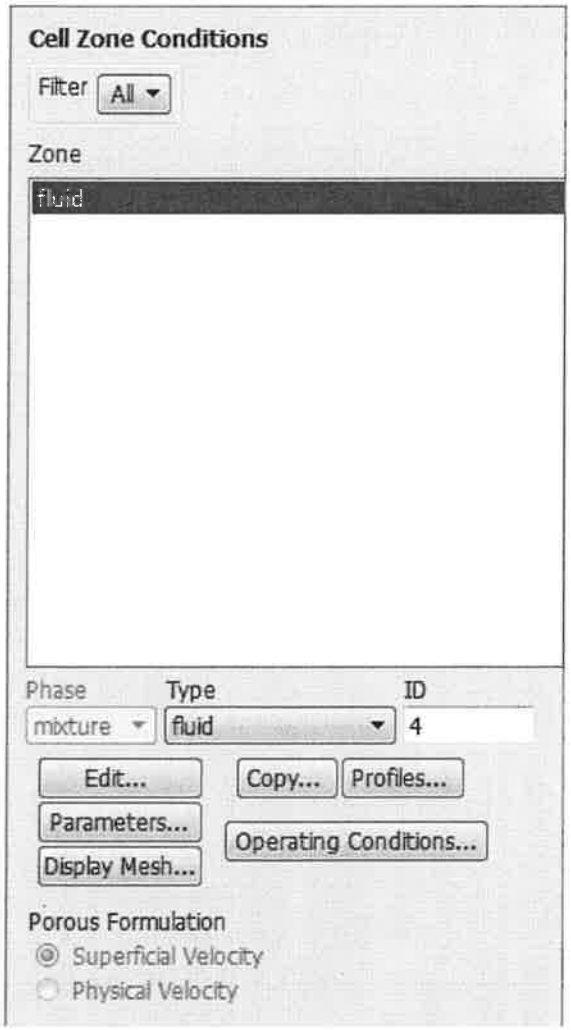


a. In the **Setting Up Physics** tab, click **Cell Zones** (**Zones** group) and select **All**.

Setting Up Physics → Zones → Cell Zones → All

This opens the **Cell Zone Conditions** task page.

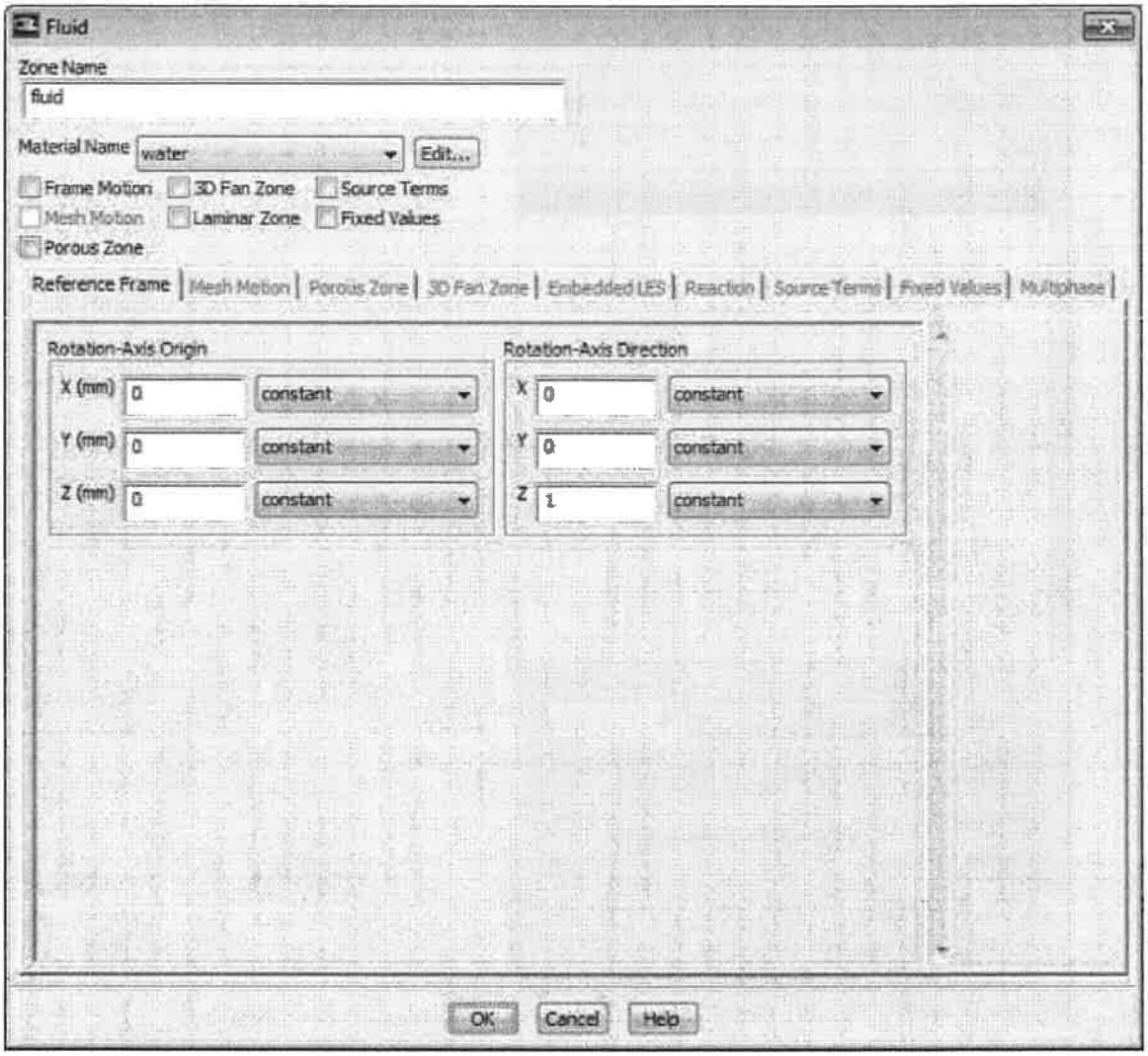
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Set the cell zone conditions for the fluid zone.

   i. In the **Cell Zone Conditions** task page, in the **Zone** list, select **fluid** and click **Edit...** to open the **Fluid** dialog box.

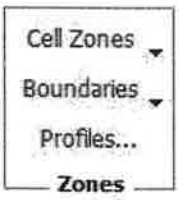
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ii. In the **Fluid** dialog box, select **water** from the **Material Name** drop-down list.

iii. Click **OK** to close the **Fluid** dialog box.

5. Set up the boundary conditions for the CFD analysis using the **Zones** group of the **Setting Up Physics** ribbon tab.

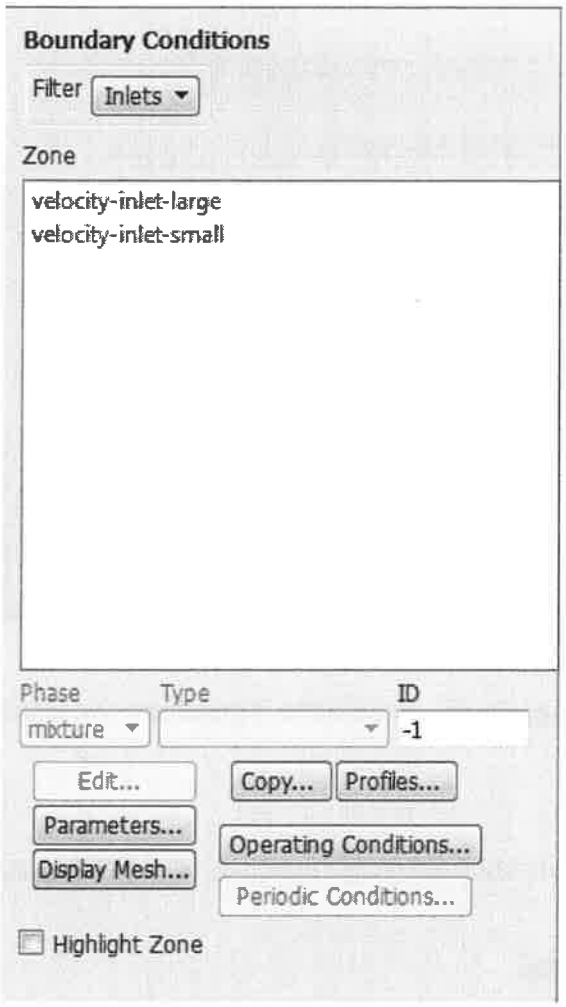


a. In the **Setting Up Physics** tab, click **Boundaries** (**Zones** group) and select **Inlets** from the drop-down list.

Setting Up Physics → Zones → Boundaries → Inlets

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

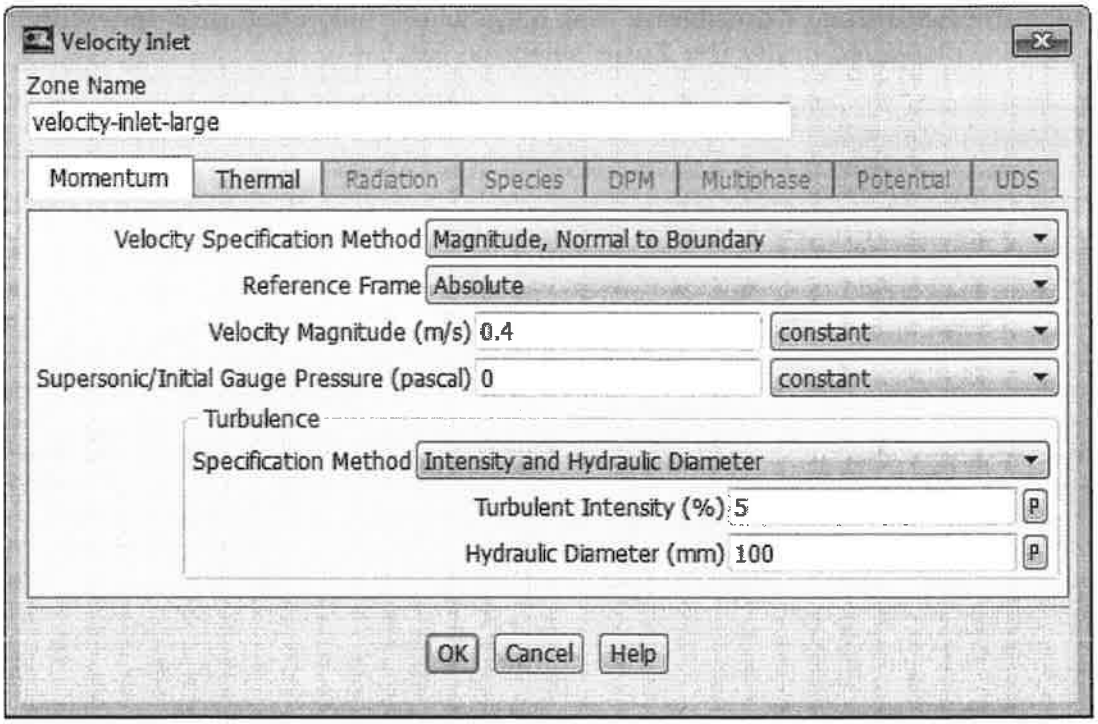
This opens the **Boundary Conditions** task page where only inlet boundaries defined in your simulation are displayed under the **Zone** selection list.



b. Set the boundary conditions at the cold inlet (**velocity-inlet-large**).

   i. From the **Zone** selection list, select **velocity-inlet-large** and click **Edit...**.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



ii. In the **Velocity Inlet** dialog box, ensure that **Magnitude, Normal to Boundary** are selected for **Velocity Specification Method**.

iii. Enter 0.4 m/s for **Velocity Magnitude**.

iv. In the **Turbulence** group, from the **Specification Method** drop-down list, select **Intensity and Hydraulic Diameter**.

v. Retain the default of 5% for **Turbulent Intensity**.

vi. Enter 100 mm for **Hydraulic Diameter**.

Note

The hydraulic diameter D_h is defined as:

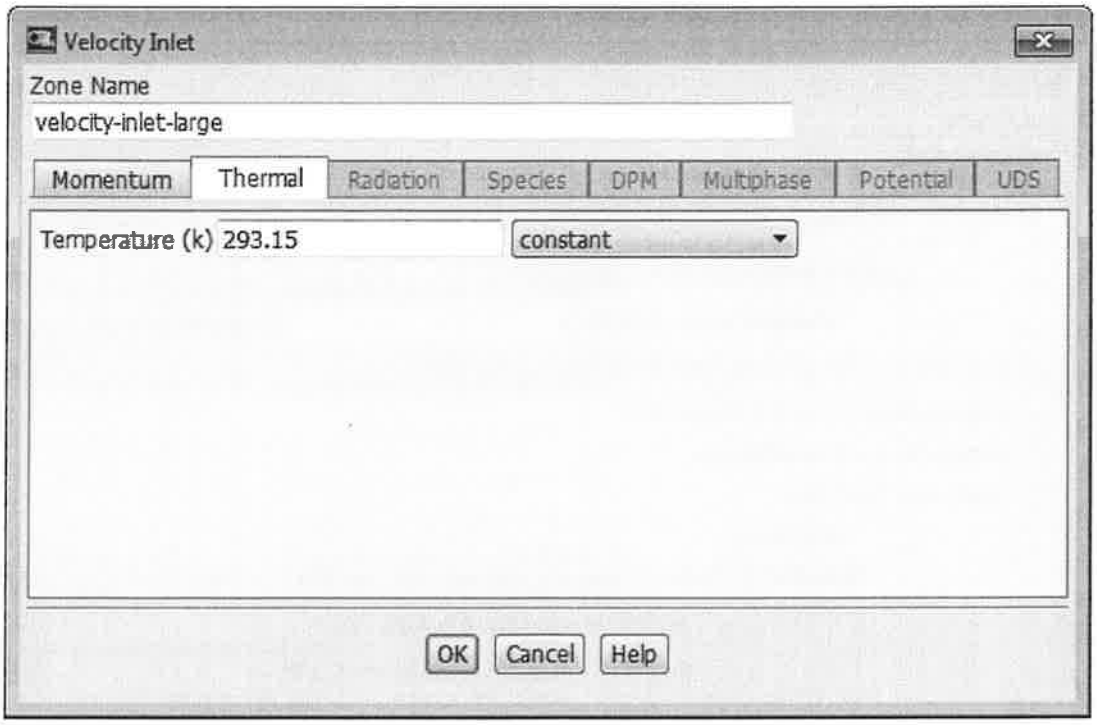
$$D_h = \frac{4A}{P_w}$$

where A is the cross-sectional area and P_w is the wetted perimeter.

---

vii. Click the **Thermal** tab.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



viii. Enter 293.15 K for **Temperature**.

ix. Click **OK** to close the **Velocity Inlet** dialog box.

c. In a similar manner, set the boundary conditions at the hot inlet (**velocity-inlet-small**), using the values in the following table:

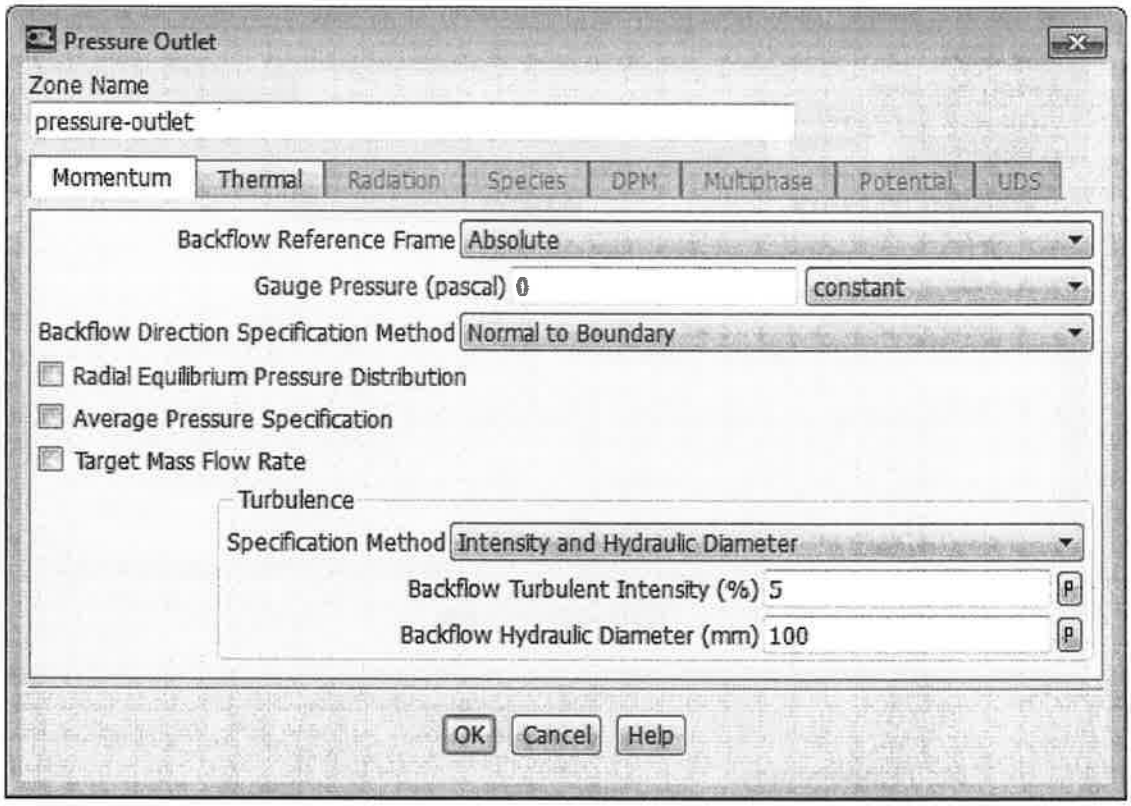
Velocity Specification Method	Components
Magnitude, Normal to Boundary	1.2 [m/s]
Specification Method	Intensity & Hydraulic Diameter
Turbulent Intensity	5%
Hydraulic Diameter	25 [mm]
Temperature	313.15[K]

d. In the **Boundary Conditions** task page, change **Filter** to **All**.

All boundaries are now listed in the **Zone** selection list.

e. Double-click **pressure-outlet** and set the boundary conditions at the outlet, as shown in the **Pressure Outlet** dialog box.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1.4.5. Solving

In the steps that follow, you will set up and run the calculation using the **Solving** ribbon tab.

Note

You can also use the task pages listed under the **Solution** branch in the tree to perform solution-related activities.

---

1. Set up solution parameters for the CFD simulation.

   a. Change the Gradient method.

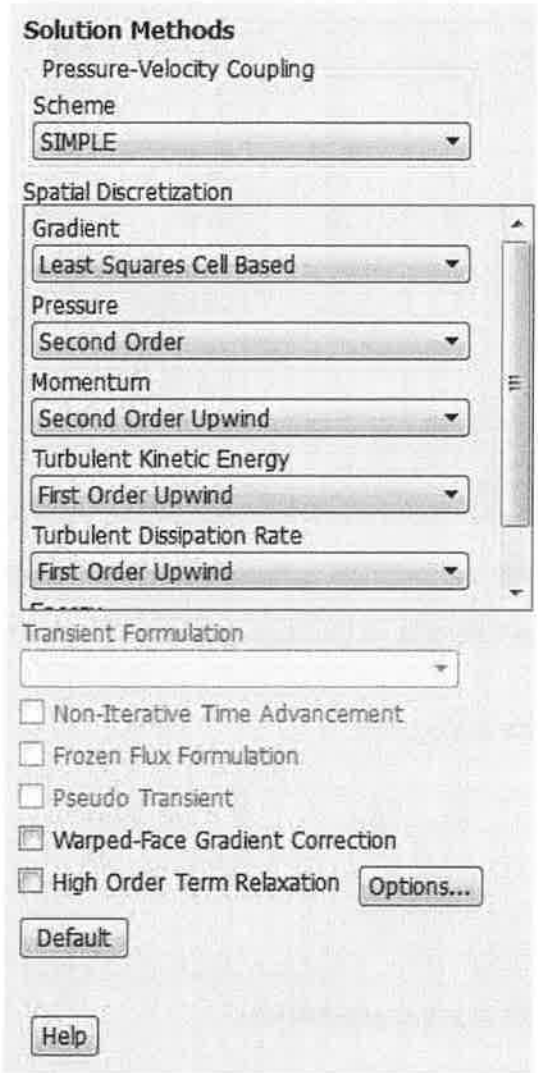
      i. In the **Solving** ribbon tab, click **Methods...** (**Solution** group).

      Solving → Solution

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This will open the **Solution Methods** task page.



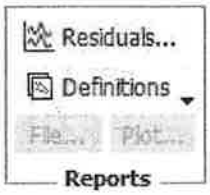
ii. In the **Spatial Discretization** group of the **Solution Methods** task page, change the **Gradient** to **Green-Gauss Node Based**. This gradient method is suggested for tetrahedral meshes.

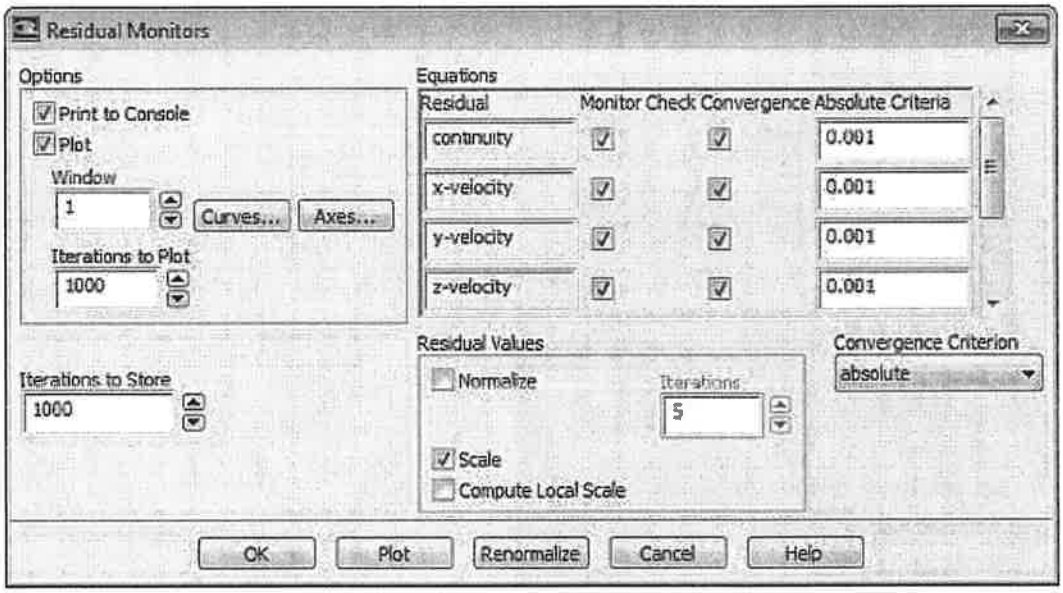
b. Enable the plotting of residuals during the calculation.

i. In the **Solving** ribbon tab, click **Residuals...** (**Reports** group).

Solving → Reports → Residuals...

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





In the **Residual Monitors** dialog box, ensure that **Plot** is enabled in the **Options** group.

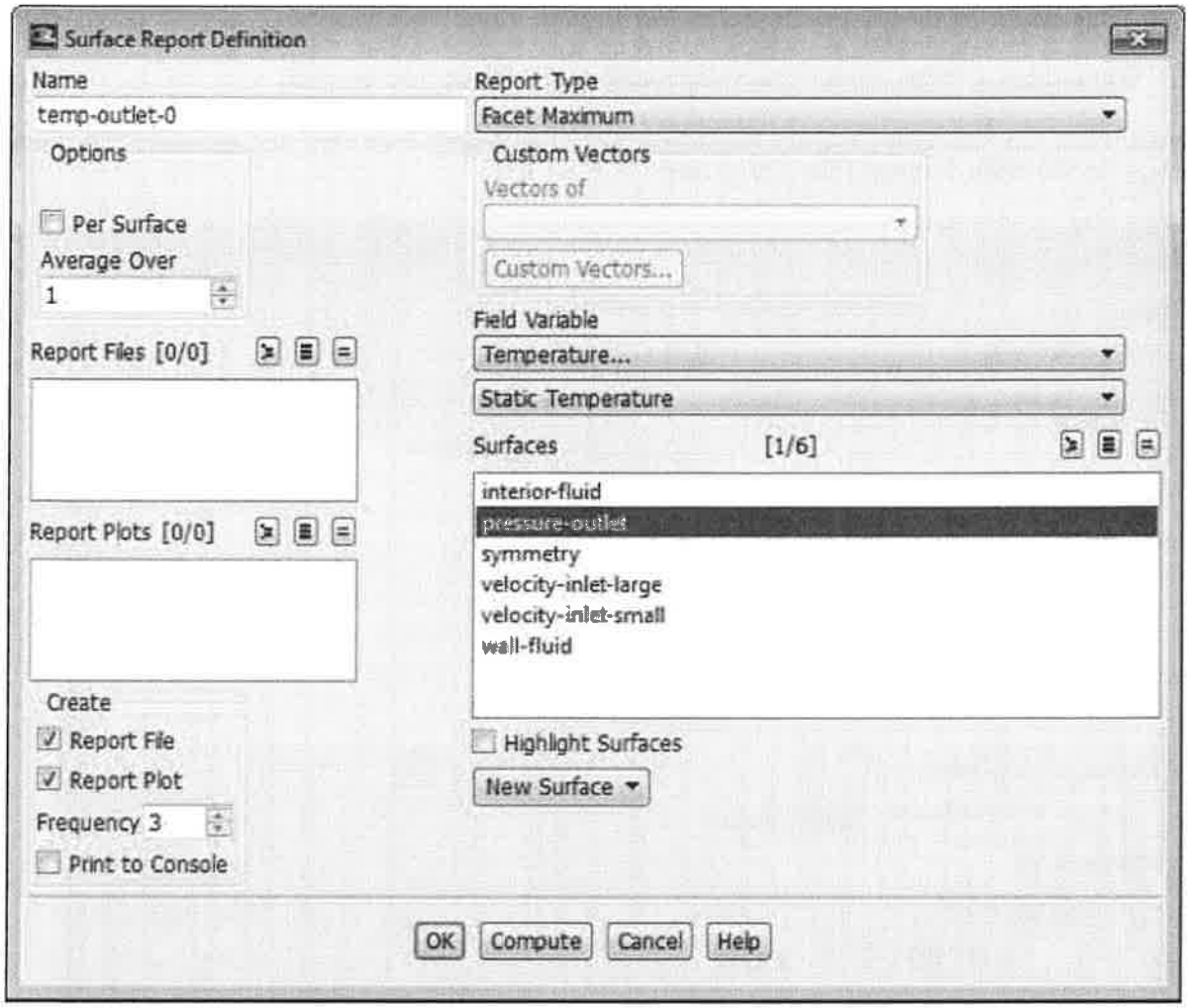
ii. Keep the default values for the **Absolute Criteria** of the **Residuals**, as shown in the **Residual Monitors** dialog box.

iii. Click **OK** to close the **Residual Monitors** dialog box.

c. Create a surface report definition at the outlet (**pressure-outlet**).

Solving → Reports → Definitions → New → Surface Report → Facet Maximum

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. In the **Surface Report Definition** dialog box, enter temp-outlet-0 for the **Name**.

ii. Under the **Create** group, enable **Report File** and **Report Plot**.

During a solution run, ANSYS Fluent will write solution convergence data in a report file and plot the solution convergence history in a graphics window.

It is good practice to monitor physical solution quantities in addition to equation residuals when assessing convergence.

iii. Set **Frequency** to 3 by clicking the up-arrow button.

This setting instructs ANSYS Fluent to update the plot of the surface report and write data to a file after every 3 iterations during the solution.

iv. Select **Temperature...** and **Static Temperature** from the **Field Variable** drop-down lists.

v. Select **pressure-outlet** from the **Surfaces** selection list.

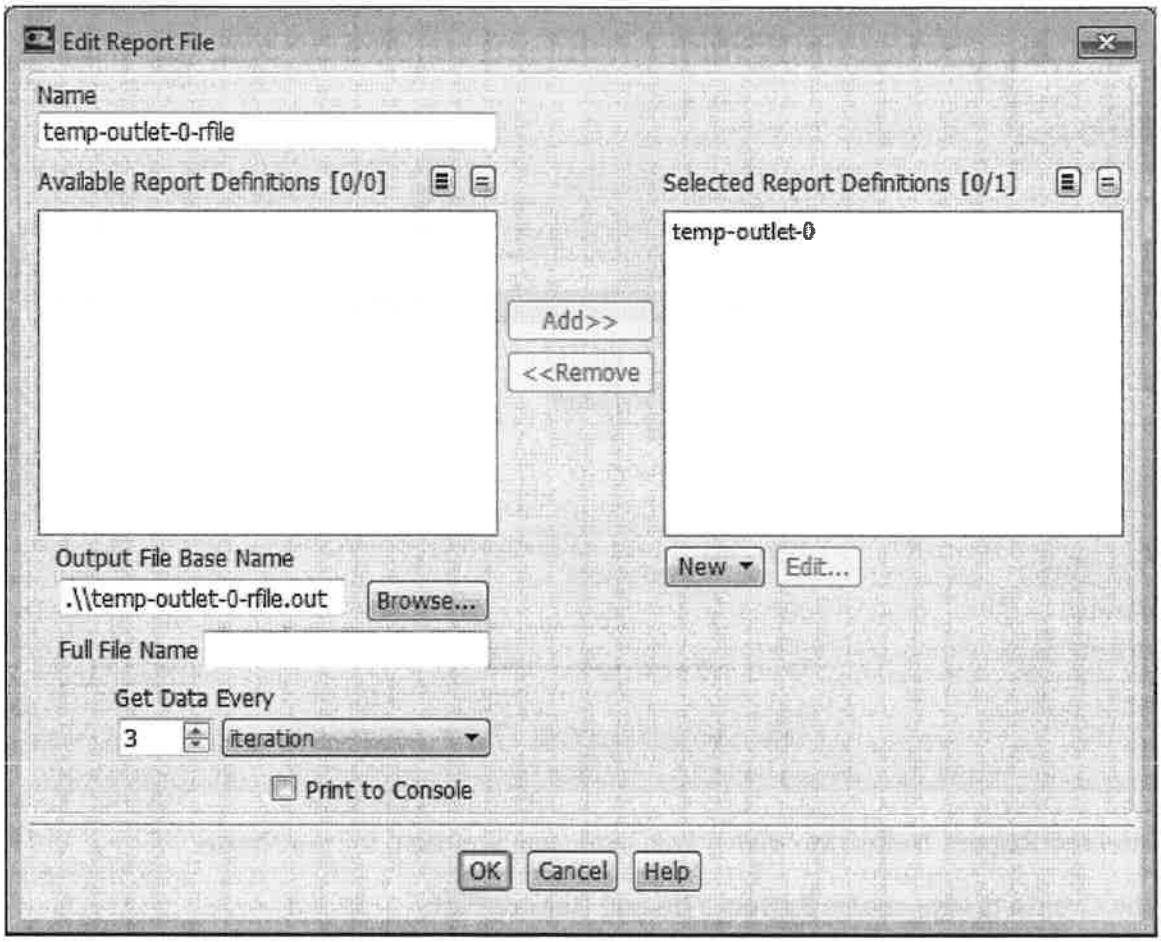
vi. Click **OK** to save the surface report definition and close the **Surface Report Definition** dialog box.

The new surface report definition temp-outlet-0 appears under the **Solution/Report Definitions** tree item. ANSYS Fluent also automatically creates the following items:

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- temp-outlet-0-rfile (under the **Solution/Report Files** tree branch)
- temp-outlet-0-rplot (under the **Solution/Report Plots** tree branch)

d. Double click the **Solution/Report Files/temp-outlet-0-rfile** tree item and examine the report file ettings in the **Edit Report File** dialog box.

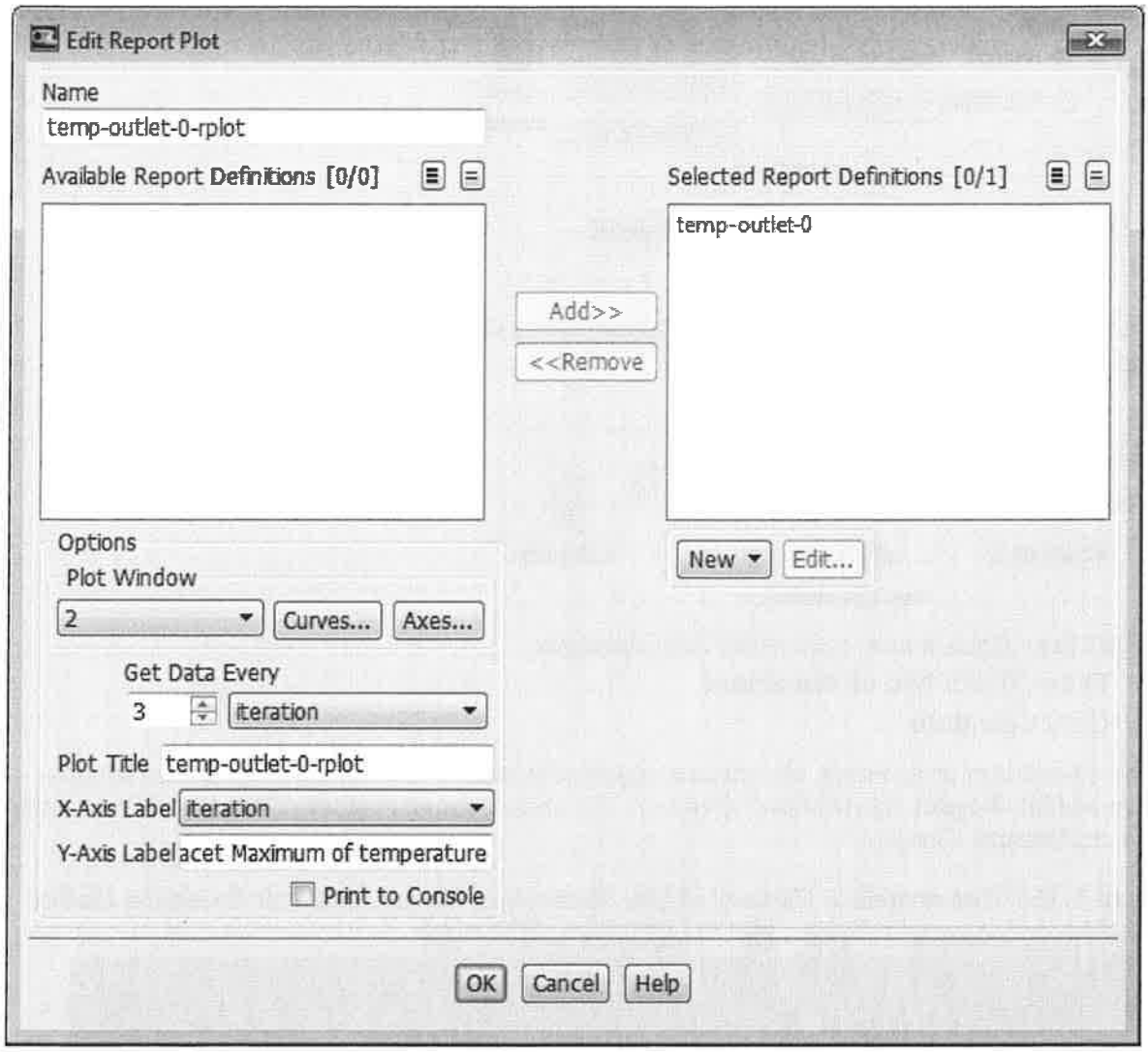


The dialog box is automatically populated with data from the temp-outlet-0 report definition. The report that will be written in a report file during a solution is listed under **Selected Report Definitions**.

Retain the default output file name and click **OK**.

e. Double click the **Solution/Report Plots/temp-outlet-0-rplot** tree item and examine the report plot settings in the **Edit Report Plot** dialog box.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The dialog box is automatically populated with data from the temp-outlet-0 report definition. As the solution progresses, the report that is listed under **Selected Report Definitions** will be plotted in a graphics tab window with the title specified in **Plot Title**.

Retain the default names for **Plot Title** and **Y-Axis Label** and click **OK**.

f. Initialize the flow field using the **Initialization** group of the **Solving** ribbon tab.

Solving → Initialization

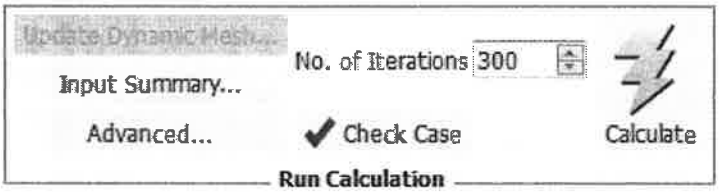
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



i. Keep the **Method** at the default of **Hybrid**.
ii. Click **Initialize**.

2. Calculate a solution using the **Run Calculation** group of the **Solving** tab.

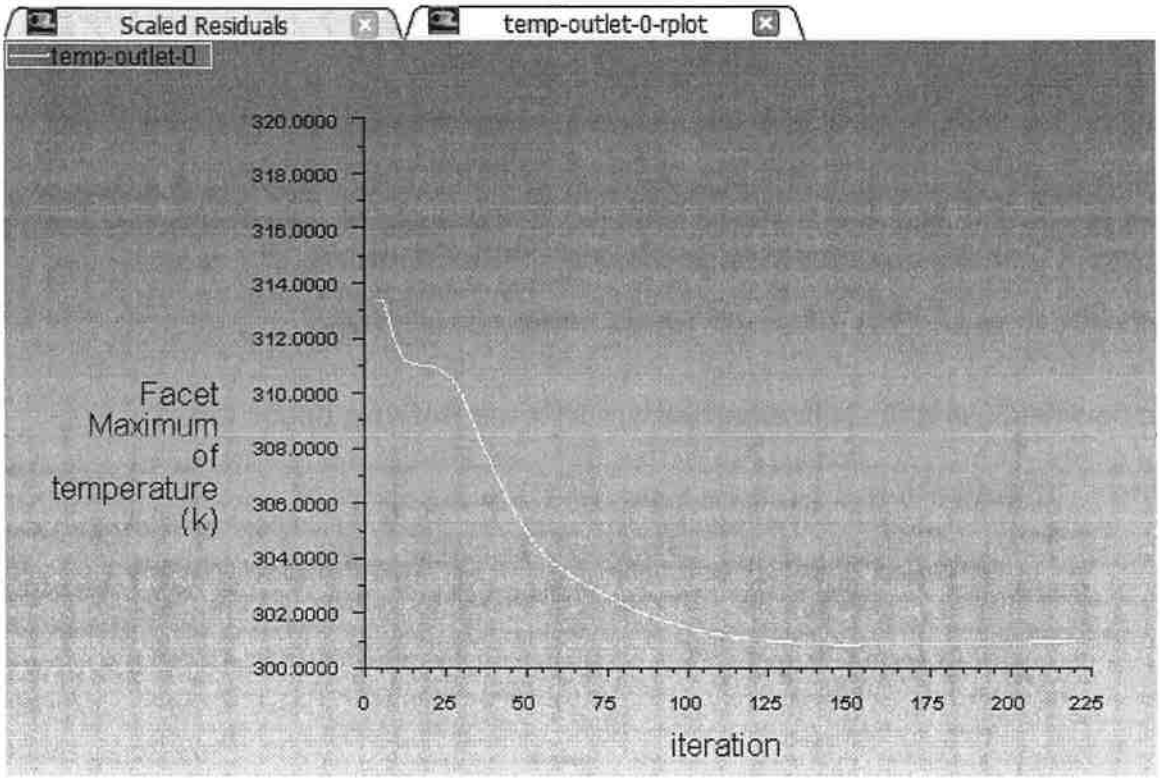
Solving → Run Calculation



a. Start the calculation by requesting 300 iterations.
   i. Enter 300 for **No. of Iterations**.
   ii. Click **Calculate**.

As the calculation progresses, the surface report plot will be plotted in the graphics window (**temp-outlet-0-rplot** tab window) (Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet).

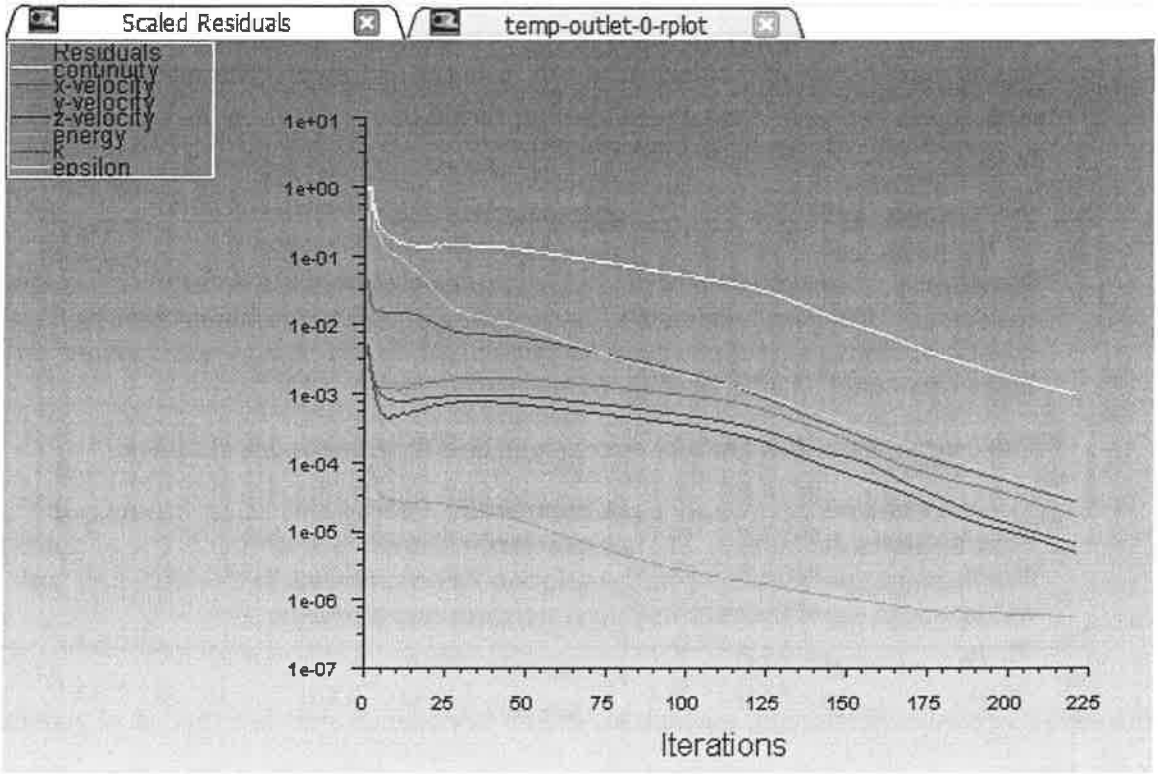
Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet



The residuals history will be plotted in the **Scaled Residuals** tab window (Figure 1.16: Residuals for the Converged Solution).

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.16: Residuals for the Converged Solution



b. Examine the plots for convergence (Figure 1.16: Residuals for the Converged Solution and Figure 1.15: Convergence History of the Maximum Temperature at Pressure Outlet).

Note

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities and checking for mass and energy balances.

There are three indicators that convergence has been reached:

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The residuals have decreased to a sufficient degree.

  The solution has converged when the **Convergence Criterion** for each variable has been reached. The default criterion is that each residual will be reduced to a value of less than 10^{-3}, except the **energy** residual, for which the default criterion is 10^{-6}.

- The solution no longer changes with more iterations.

  Sometimes the residuals may not fall below the convergence criterion set in the case setup. However, monitoring the representative flow variables through iterations may show that the residuals have stagnated and do not change with further iterations. This could also be considered as convergence.

- The overall mass, momentum, energy, and scalar balances are obtained.

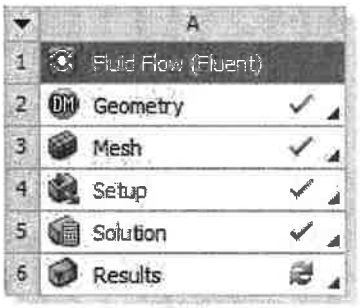
  You can examine the overall mass, momentum, energy and scalar balances in the **Flux Reports** dialog box. The net imbalance should be less than 0.2% of the net flux through the domain when the solution has converged. In the next step you will check to see if the mass balance indicates convergence.

3. With ANSYS Fluent still running, go back to ANSYS Workbench and view the list of generated files.

   View → Files

   Note the addition of the surface report definition file temp-outlet-0.out to the list of files.

   The status of the **Solution** cell is now up-to-date.



elbow

1.4.6. Displaying Results in ANSYS Fluent

In this step, you will display the results of the simulation in ANSYS Fluent, display the results in CFD-Post, and then review the list of files generated by ANSYS Workbench.

1. Display results in ANSYS Fluent using the **Postprocessing** ribbon tab.

   In ANSYS Fluent, you can perform a simple evaluation of the velocity and temperature contours on the symmetry plane.

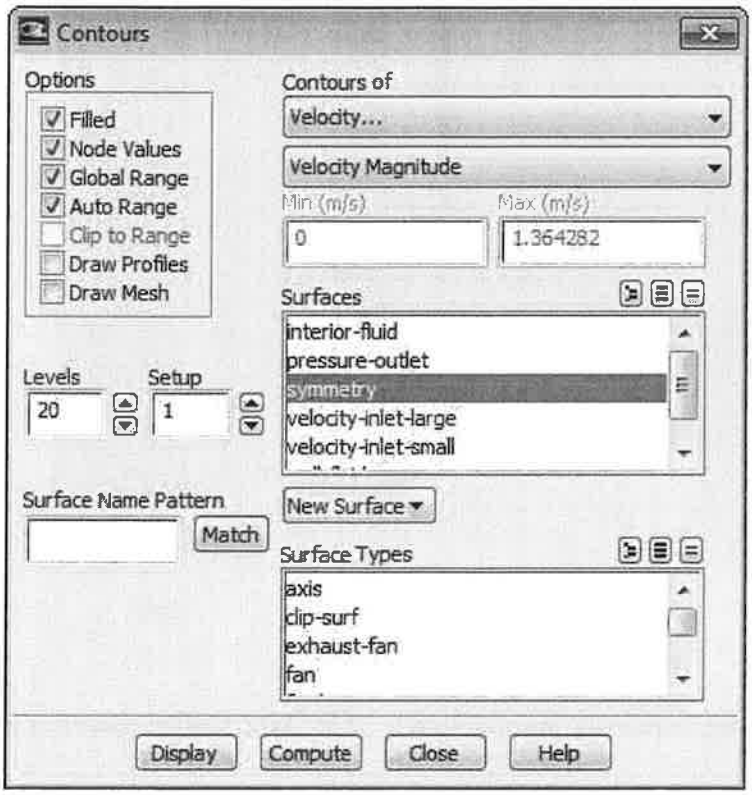
Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Display filled contours of velocity magnitude on the symmetry plane (Figure 1.17: Velocity Distribution Along Symmetry Plane).

Postprocessing → Graphics → Contours → Edit...

Note

You can also double-click the **Results/Graphics/Contours** tree item.



i. In the **Contours** dialog box, in the **Options** group, enable **Filled**.

ii. Ensure that **Node Values** is enabled in the **Options** group.

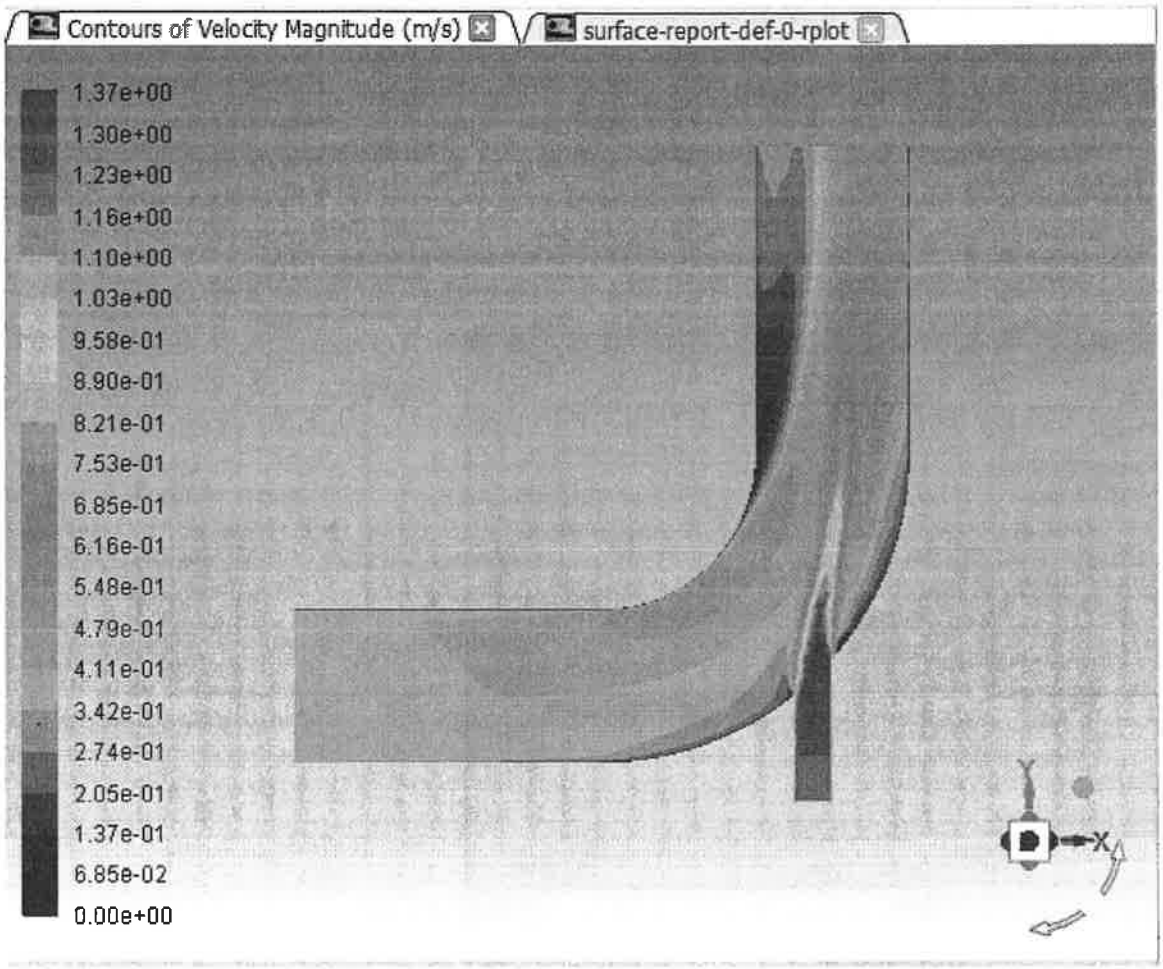
iii. From the **Contours of** drop-down lists, select **Velocity...** and **Velocity Magnitude**.

iv. From the **Surfaces** selection list, deselect all items by clicking and then select **symmetry**.

v. Click **Display** to display the contours in the active graphics window.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

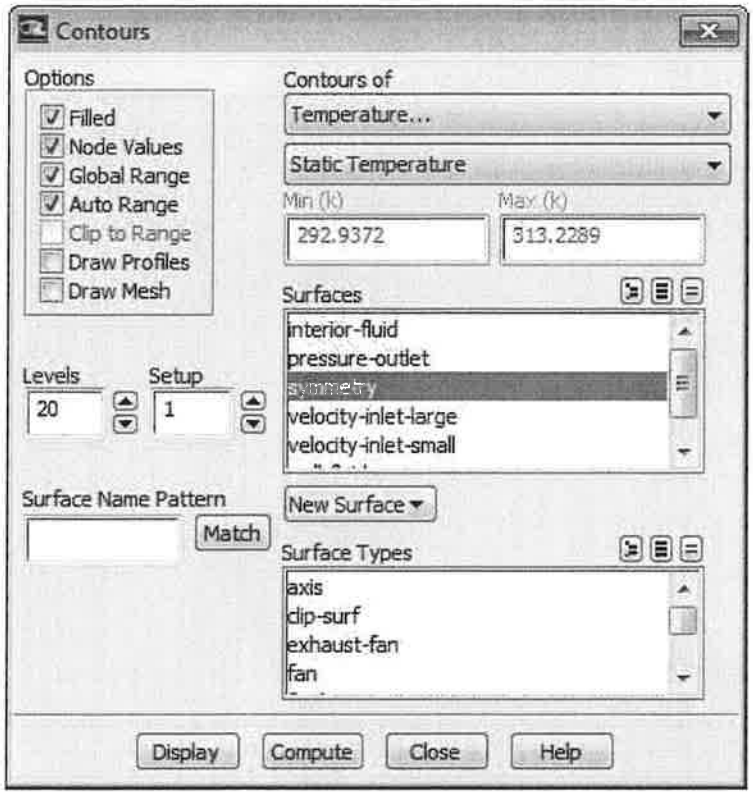
Figure 1.17: Velocity Distribution Along Symmetry Plane



b. Display filled contours of temperature on the symmetry plane (Figure 1.18: Temperature Distribution Along Symmetry Plane).

Postprocessing → Graphics → Contours → Edit...

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

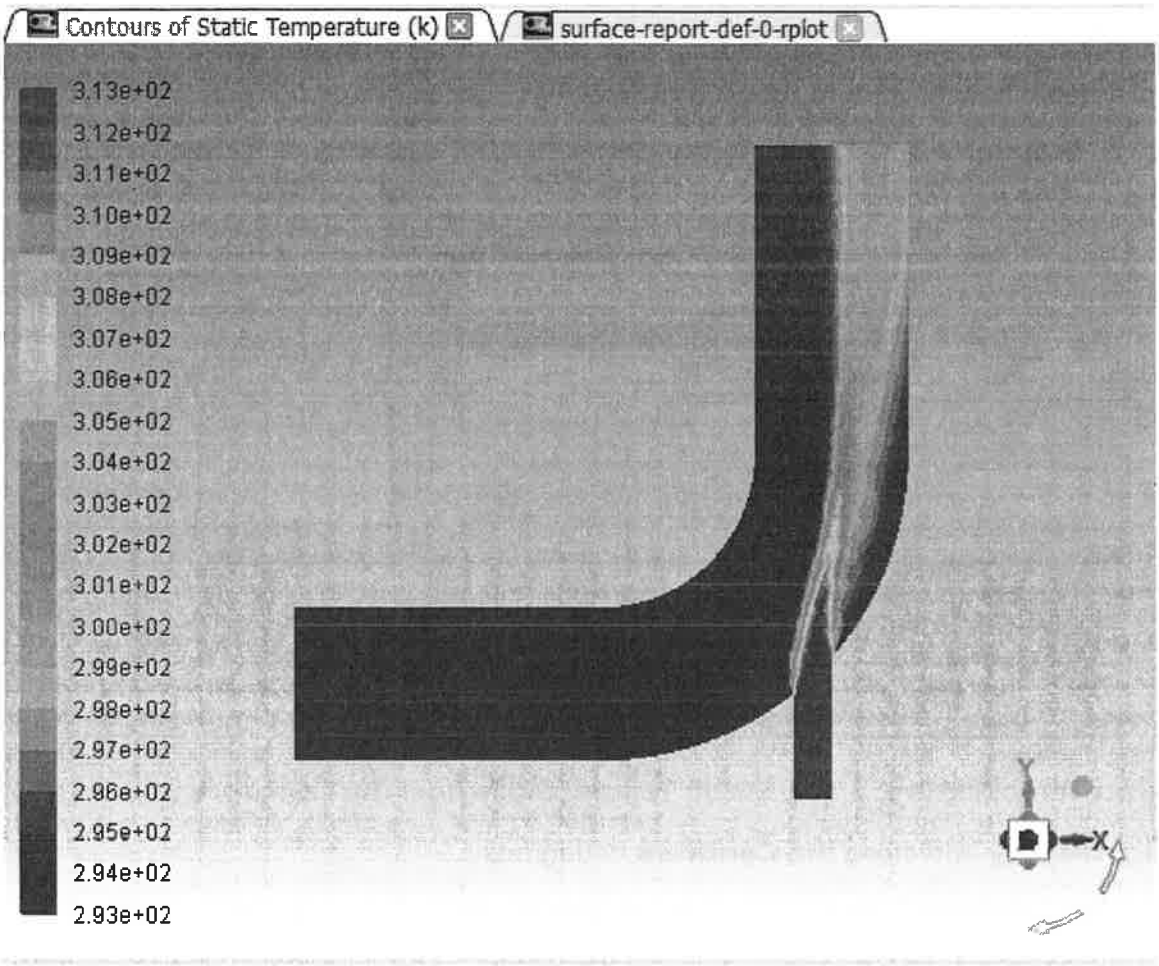


i. Select **Temperature...** and **Static Temperature** from the **Contours of** drop-down lists.

ii. Click **Display** and close the **Contours** dialog box.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.18: Temperature Distribution Along Symmetry Plane



c. Close the ANSYS Fluent application.

File → Close Fluent

Important

Note that the ANSYS Fluent case and data files are automatically saved when you exit ANSYS Fluent and return to ANSYS Workbench.

---

d. View the list of files generated by ANSYS Workbench.

View → Files

Note the addition of the compressed ANSYS Fluent case and data files to the list of files. These will have names like FFF-1.cas.gz and FFF-1-00222.dat.gz. Note that the digit(s) following FFF may be different if you have had to restart the meshing or calculation steps for any reason and that the name of the data file is based on the number of iterations. Thus your file names may be slightly different than those shown here.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

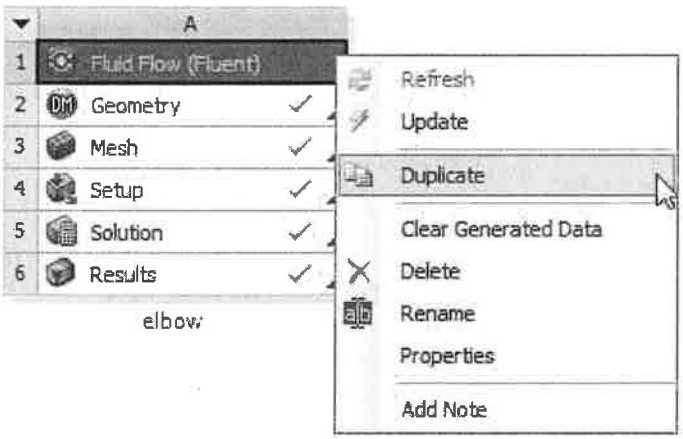
1.4.7. Duplicating the Fluent-Based Fluid Flow Analysis System

At this point, you have a completely defined fluid flow system that has a geometry, a computational mesh, a CFD setup and solution, and corresponding results. In order to study the effects upon the flow field that may occur if you were to alter the geometry, another fluid flow analysis is required. One approach would be to use the current system and change the geometry, however you would overwrite the data from your previous simulation. A more suitable and effective approach would be to create a copy, or duplicate, of the current system, and then make the appropriate changes to the duplicate system.

In this step, you will create a duplicate of the original Fluent-based fluid flow system, then review the list of files generated by ANSYS Workbench.

1. In the **Project Schematic**, right-click the title cell of the **Fluid Flow (Fluent)** system and select **Duplicate** from the context menu.

Figure 1.22: Duplicating the Fluid Flow System



Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.23: The Original Fluid Flow System and Its Duplicate



Note

Notice that in the duplicated system, the state of the **Solution** cell indicates that the cell requires an update while the state of the **Results** cell indicates that the cell requires attention. This is because when a system is duplicated, the case and data files are not copied to the new system, therefore, the new system does not yet have solution data associated with it.

2. Rename the duplicated system to new-elbow.
3. Save the elbow-workbench project in ANSYS Workbench.

1.4.8. Changing the Geometry in ANSYS DesignModeler

Now that you have two separate, but equivalent, Fluent-based fluid flow systems to work from, you can make changes to the second system without impacting the original system. In this step, you will make a slight alteration to the elbow geometry in ANSYS DesignModeler by changing the diameter of the smaller inlet, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS DesignModeler.

   Double-click the **Geometry** cell of the new-elbow system (cell B2) to display the geometry in ANSYS DesignModeler.

2. Change the diameter of the small inlet (velocity-inlet-small).

   a. In the **Tree Outline**, select **Cylinder1** to open the **Details View** of the small inlet pipe.

   b. In the **Details View**, change the **FD10, Radius (>0)** value from 12.5 millimeters to 19 millimeters.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

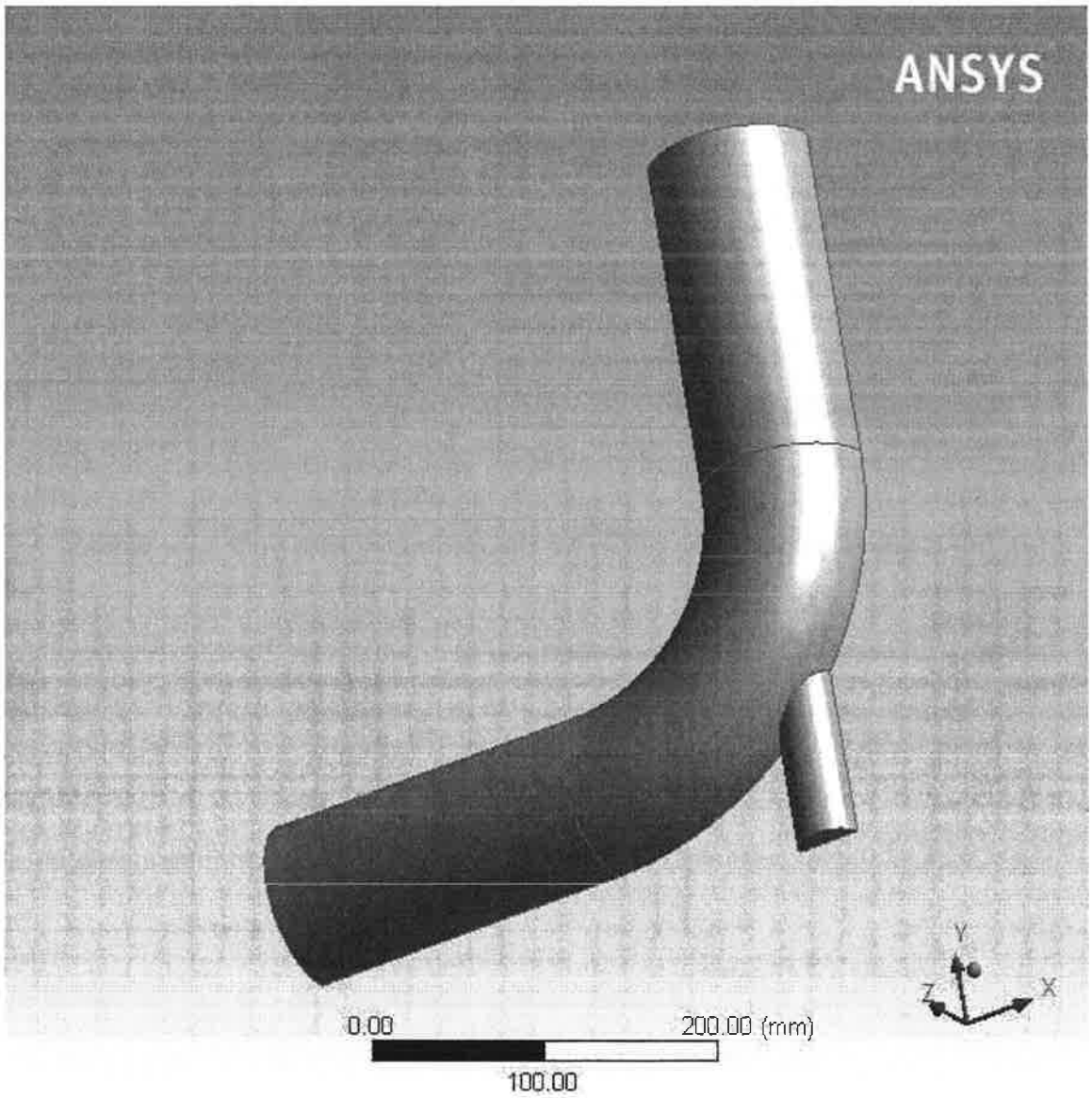
Details View

Details of Cylinder1	
Cylinder	Cylinder1
Base Plane	XYPlane
Operation	Add Material
Origin Definition	Coordinates
FD3, Origin X Coordinate	137.5 mm
FD4, Origin Y Coordinate	-225 mm
FD5, Origin Z Coordinate	0 mm
Axis Definition	Components
FD6, Axis X Component	0 mm
FD7, Axis Y Component	125 mm
FD8, Axis Z Component	0 mm
FD10, Radius (>0)	19 mm
As Thin/Surface?	No

c. Click the **Generate** button to generate the geometry with your new values.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.19: Changing the Diameter of the Small Inlet in ANSYS DesignModeler



3. Close ANSYS DesignModeler.
4. View the list of files generated by ANSYS Workbench.

   View → Files

   Note the addition of the geometry, mesh, and ANSYS Fluent settings files now associated with the new, duplicated system.

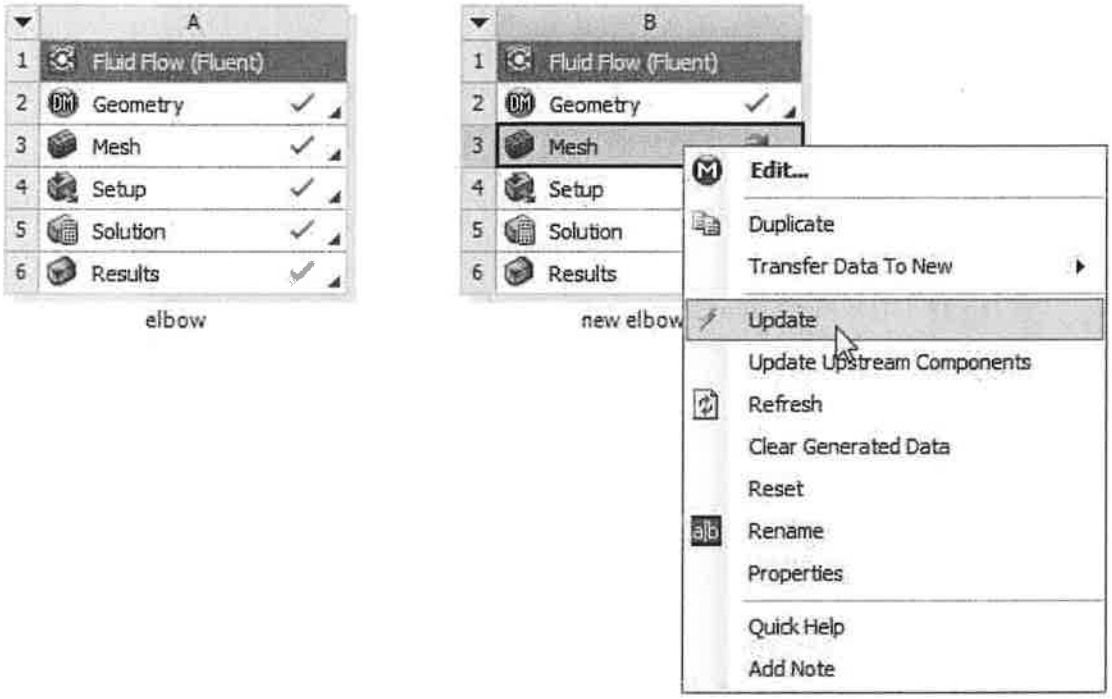
1.4.9. Updating the Mesh in the ANSYS Meshing Application

The modified geometry now requires a new computational mesh. The mesh settings for the duplicated system are retained in the duplicated system. In this step, you will update the mesh based on the mesh settings from the original system, then review the list of files generated by ANSYS Workbench.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project Schematic**, right-click the **Mesh** cell of the new-elbow system (cell B3) and select **Update** from the context menu. This will update the mesh for the new geometry based on the mesh settings you specified earlier in the ANSYS Meshing application without having to open the ANSYS Meshing editor to regenerate the mesh.

Figure 1.25: Updating the Mesh for the Changed Geometry

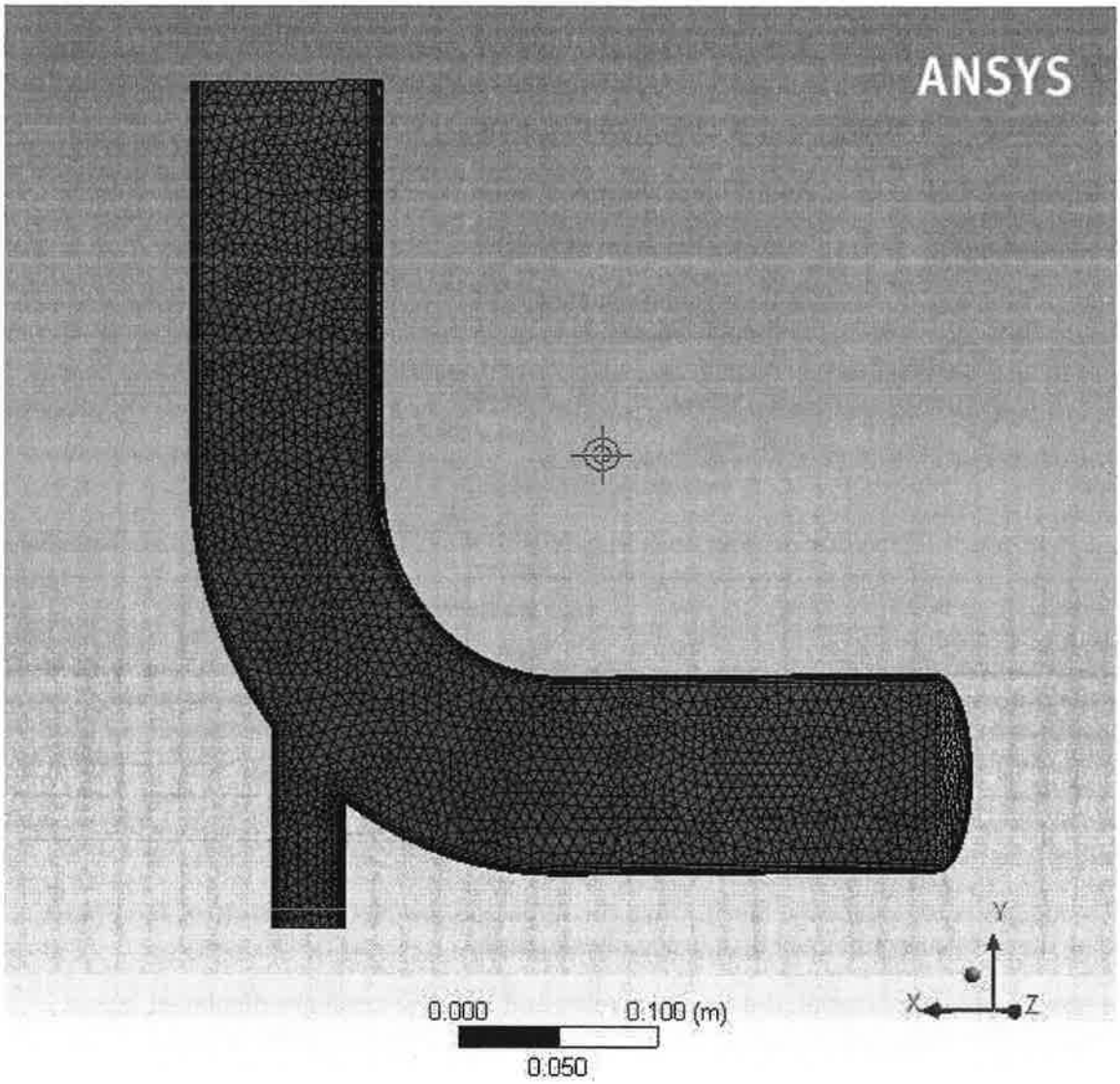


It will take a few moments to update the mesh. Once the update is complete, the state of the **Mesh** cell is changed to up-to-date, symbolized by a green check mark.

For illustrative purposes of the tutorial, the new geometry and the new mesh are displayed below.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.20: The Updated Geometry and Mesh in the ANSYS Meshing Application



Inspecting the files generated by ANSYS Workbench reveals the updated mesh file for the duplicated system.

View → Files

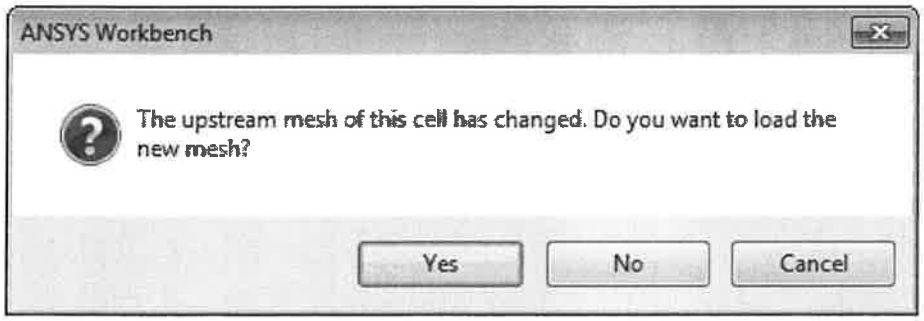
1.4.10. Calculating a New Solution in ANSYS Fluent

Now that there is an updated computational mesh for the modified geometry in the duplicated system, a new solution must be generated using ANSYS Fluent. In this step, you will revisit the settings within ANSYS Fluent, calculate another solution, view the new results, then review the list of files generated by ANSYS Workbench.

1. Open ANSYS Fluent.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Project Schematic**, right-click the **Setup** cell of the new-elbow system (cell B4) and select **Edit...** from the context menu. Since the mesh has been changed, you are prompted as to whether you want to load the new mesh into ANSYS Fluent or not.



Click **Yes** to continue, and click **OK** when Fluent Launcher is displayed in order to open ANSYS Fluent.

2. Ensure that the unit of length is set to millimeters.

   Setting Up Domain → Mesh → Units...

3. Check the mesh (optional).

   Setting Up Domain → Mesh → Check

4. Revisit the boundary conditions for the small inlet.

   Setup → Boundary Conditions → velocity-inlet-small Edit...

   In the **Velocity Inlet** dialog box for **velocity-inlet-small**, you must set the hydraulic diameter to 38 mm based on the new dimensions of the small inlet.

5. Re-initialize the solution.

   Solving → Initialization

   Keep the **Method** at the default of **Hybrid** and click **Initialize**.

6. Recalculate the solution.

   Solving → Run Calculation → Calculate

   Keep the **No. of Iterations** set to 300 and click **Calculate**.

7. Close ANSYS Fluent.

8. Revisit the results of the calculations in ANSYS Fluent.

   Double-click the **Solution** cell of the new-elbow fluid flow system to re-open ANSYS Fluent where you can review the results of the new solution.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

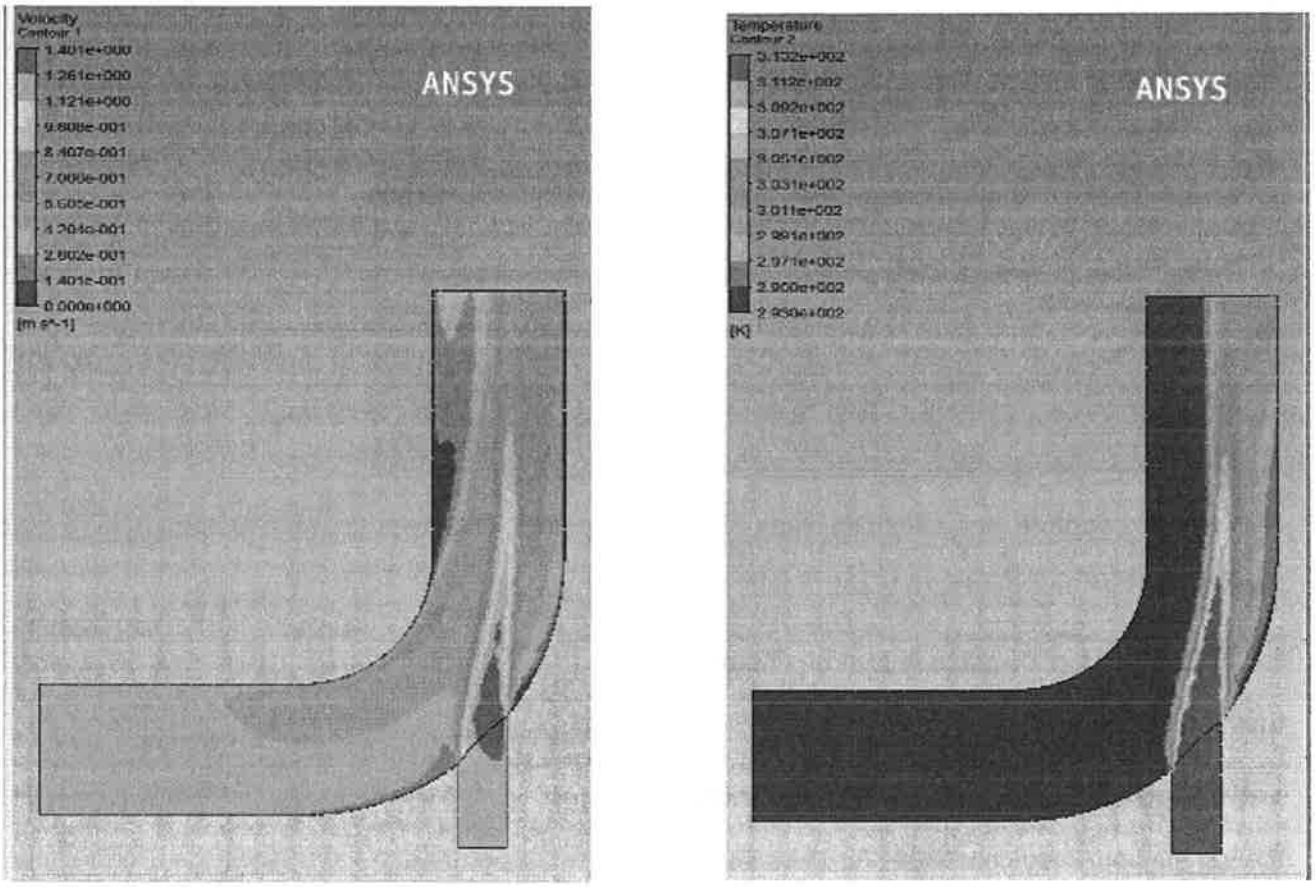


Figure 1.21: CFD-Post Displaying Velocity and Temperature Contours on the Symmetry Plane for the New Geometry

9. Close ANSYS Fluent.

10. Save the elbow-workbench project in ANSYS Workbench.

11. View the list of files generated by ANSYS Workbench.

    View → Files

    Note the addition of the solution files now associated with new duplicated system.

1.5. Summary

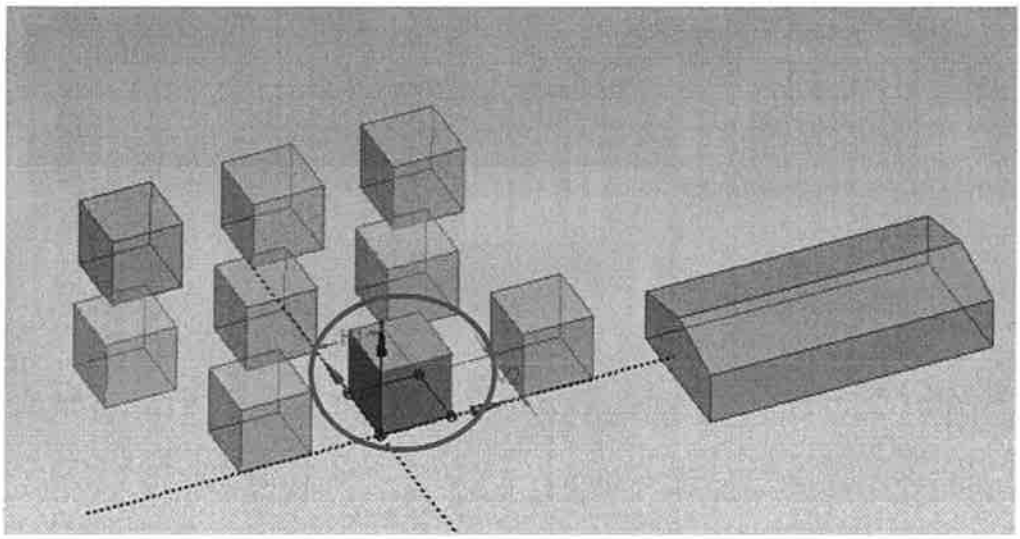
In this tutorial, portions of ANSYS Workbench were used to compare the fluid flow through two slightly different geometries. ANSYS DesignModeler was used to create a mixing elbow geometry, ANSYS Meshing was used to create a computational mesh, ANSYS Fluent was used to calculate the fluid flow throughout the geometry using the computational mesh, and ANSYS Fluent was used to analyze the results. In addition, the geometry was altered, a new mesh was generated, and a new solution was calculated. Finally, ANSYS Workbench was set up so that the results of both calculations were compared.

Release 17.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tutorial 2 Simulation of Natural Ventilation around and across a Silsoe Cube

Abstract

This tutorial simply demonstrates the procedure of CFD simulation of natural ventilation around and across a Silsoe Cube, subject to the CFD methodology outlined in BCA Green Mark 2015 for non-residential building (NRB).



In summary, the tutorial covers details about the creation of geometrical model (in Design Modeler), the generation of computational grid (in ANSYS Meshing), numerical settings and computation (ANSYS FLUENT) and post processing of results (ANSYS FLUENT).

Please note that the tutorial has been drafted using ANSYS Workbench 17.2.

Pre-requisite:

- Familiar with the workbench working environment (complete Tutorial 1 using workbench)
- One external geometry file:
  - surrounding-buildings.stp – for surrounding buildings
- One user define function:
  - UDF_N_2.c – log-law wind profiles (north wind)

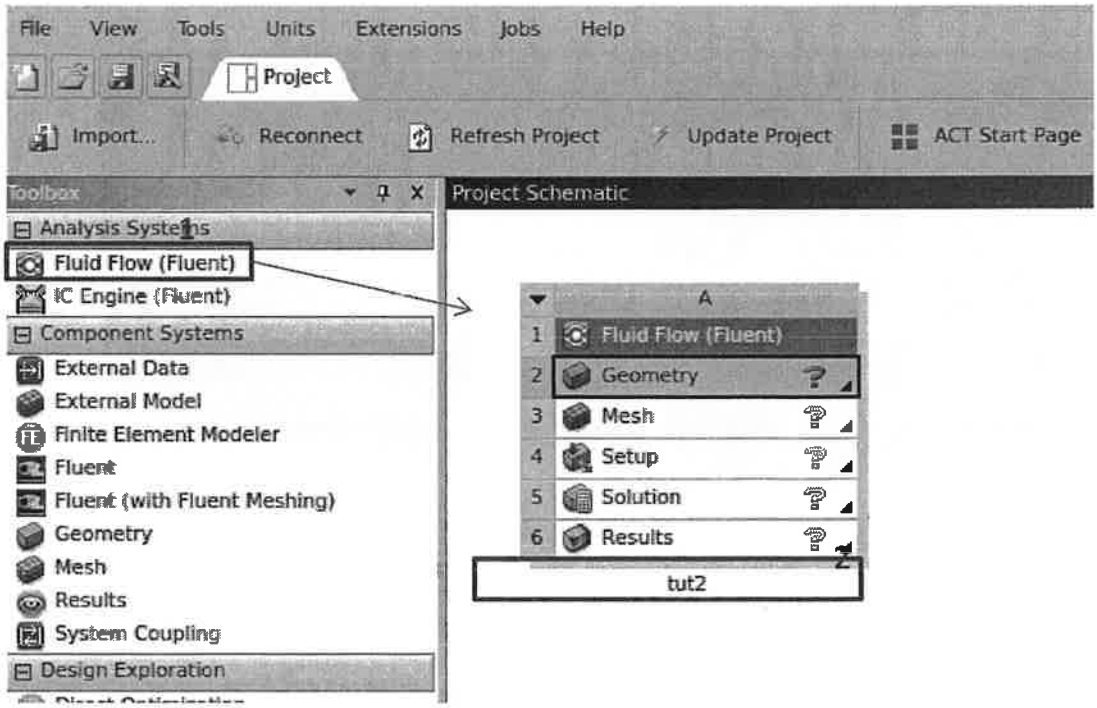
Launch ANSYS Workbench

1. From the Windows **Start** menu, select **Start > All Programs > ANSYS 17.0 > Workbench 17.0** to start a new ANSYS Workbench session.

2. Create a new **Fluent fluid flow analysis** system by double-clicking the **Fluid Flow (Fluent)** option under **Analysis Systems** in the Toolbox.

3. Double-click the **Fluid Flow (Fluent)** label underneath the analysis system (if it is not already highlighted) to name the analysis to be "tut2".

4. Select **Save** under the File menu in ANSYS Workbench to save the project. In the pop-up window of **Save As** dialog box, browse to your working folder and enter a specific name (say for example, "tut2") for the ANSYS Workbench project.



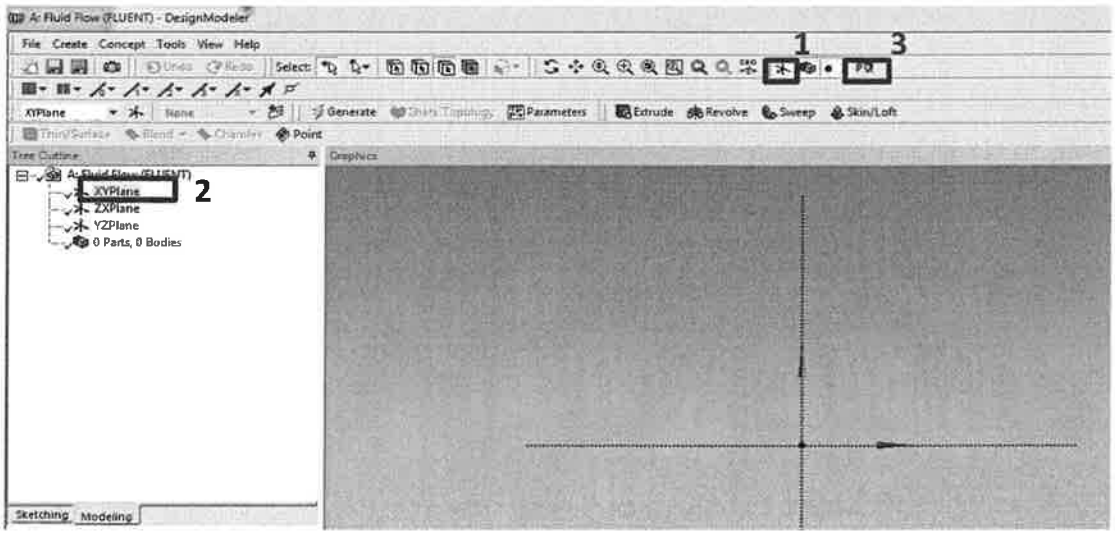
1. Geometry Creation in Design Modeler

Double click **Geometry** icon in the created project schematics to launch Design Modeler. Choose the default SI unit system (Meter).

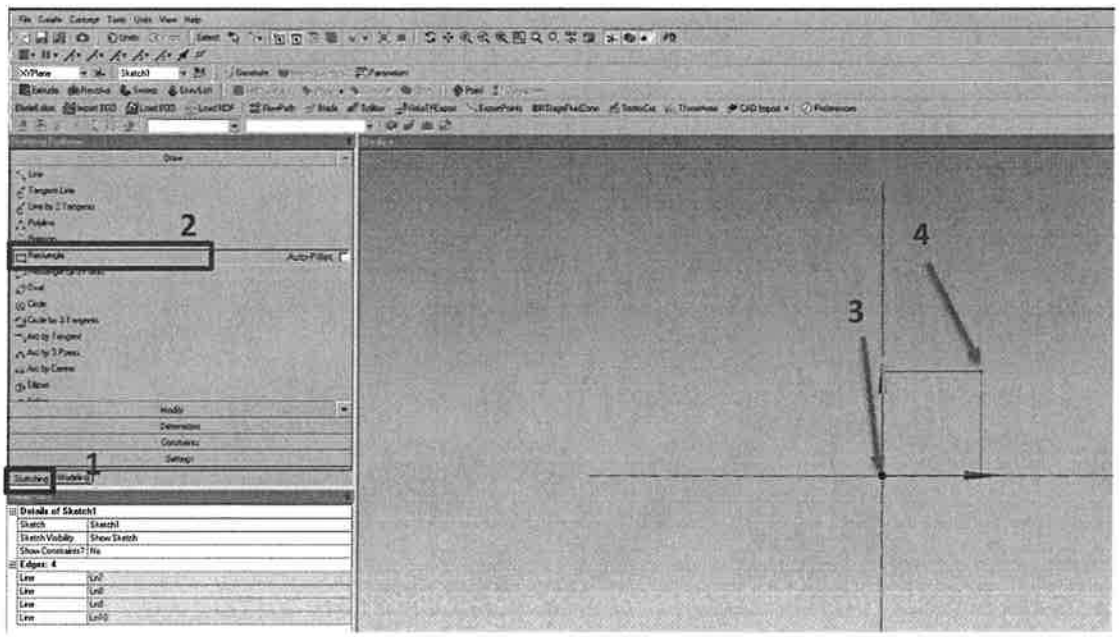
1.1 Creation of the Silsoe Cube of concern

1.1.1 Creation of a 2D sketch

Left click on the **Display Plane** icon to view the XYZ planes. Click **XYPlane**, and then choose the **Look At** icon to reorient the view.

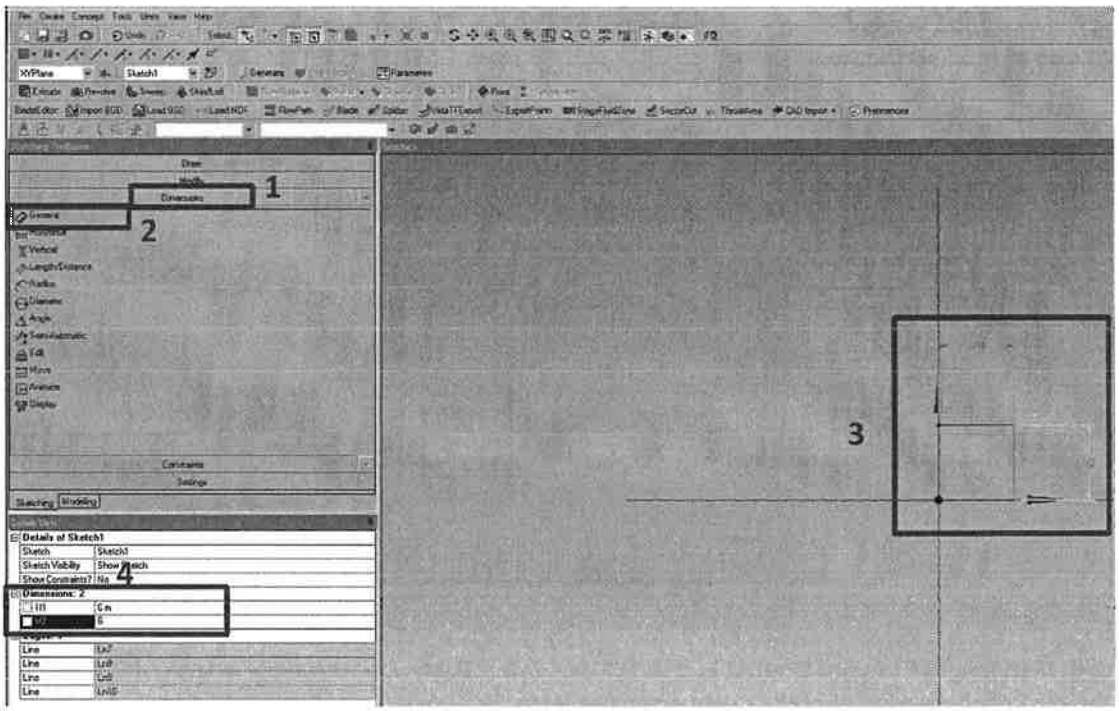


Click **Sketching** tab to draw **Sketch1** on the XYPlane. Select **Rectangle**, move the mouse cursor around the origin and then left click the origin when the letter "P" appears on the screen, and then click the second point along the diagonal direction to create a rectangle in the first quadrant.



Dimension the rectangle: H1 = 6, V2 = 6.

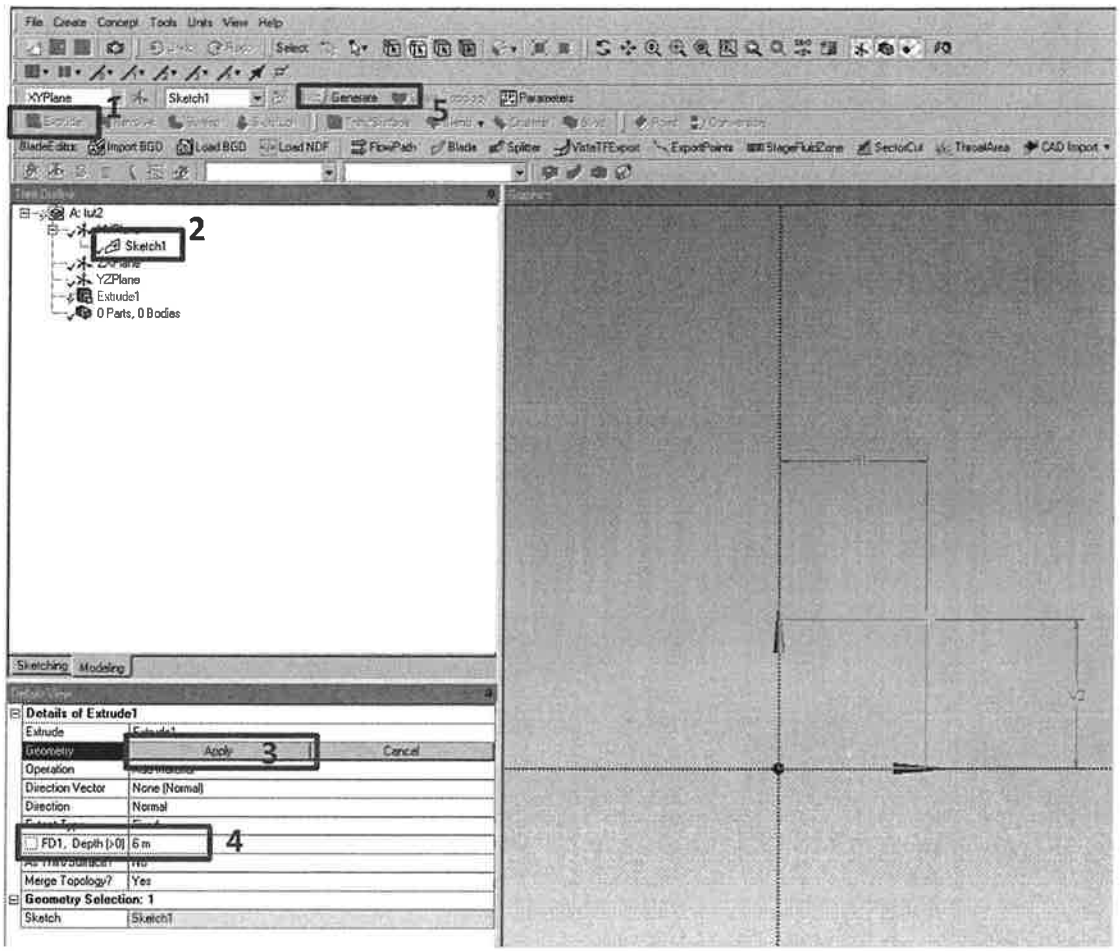
Under the sketching toolbox, choose ***Dimensions*** → ***General***, and then left click one of the horizontal edges of the rectangle, and then move the mouse away vertically and click the screen again. Then the dimension "H1" for the edge will be marked and created. Similarly, create the dimension of "V2" for any one of the vertical edges. Update the values: H1 = 6, V2 = 6. The sketch will be automatically updated to be a square.



1.1.2 Generation of the cube

Extrude the square along the +Z direction to generate Silsoe cube with the height of 6m.

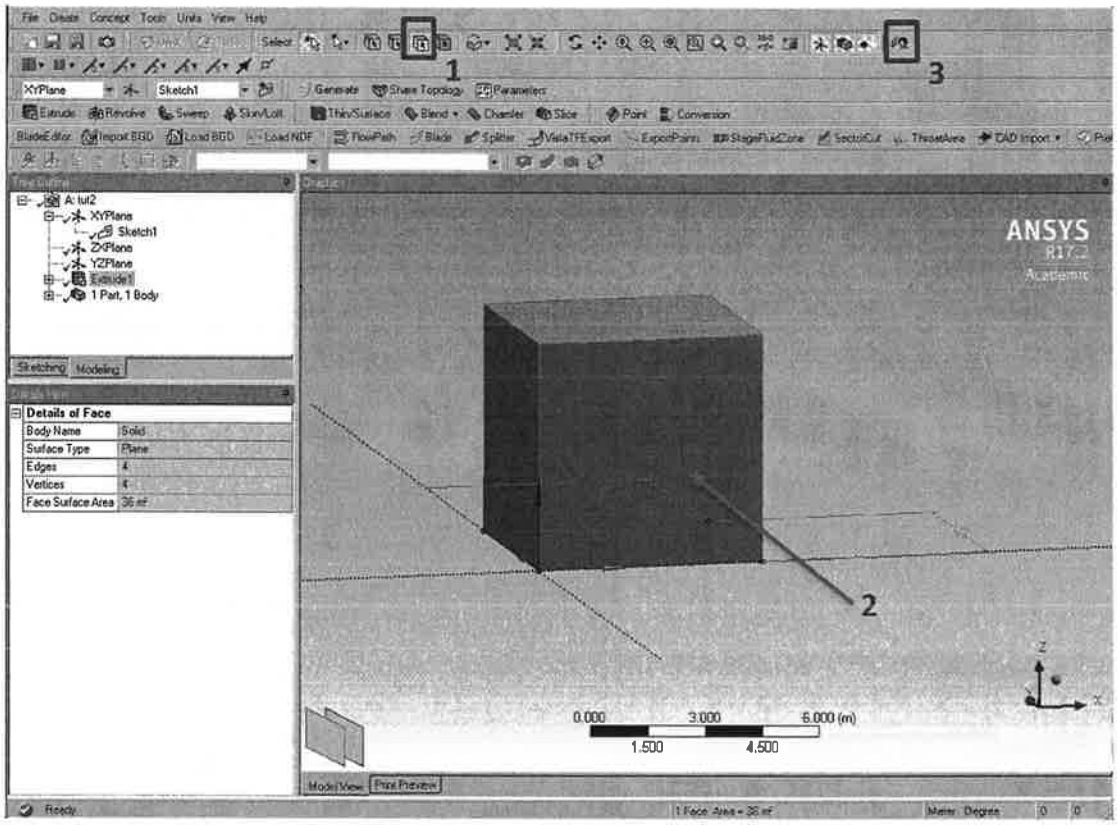
Click ***Extrude*** button on the menu bar, and then select "Sketch 1" as the geometry to be extrude with the height of 6m along the normal direction (+Z direction). Click **Generate** button to create the 3D extrusion.



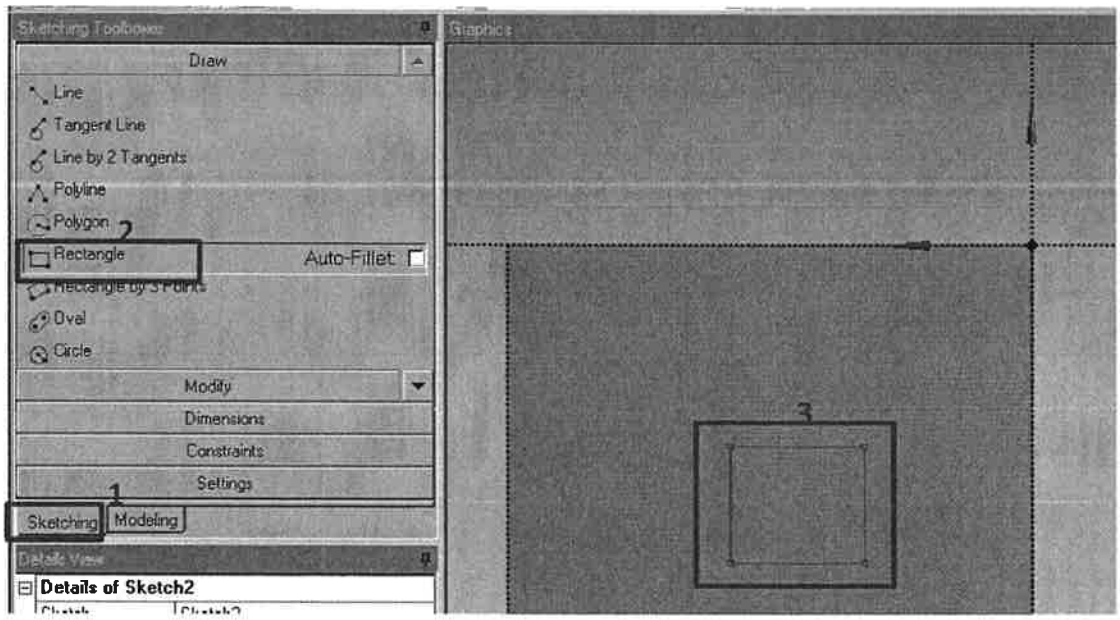
1.1.3 Creation of two openings

Create openings on the northern and southern sides of the Silsoe Cube.

Choose the "**face filter**" and then click the face on the -Y side of the cube for the opening generation and reorient the view by clicking **Look At** icon in the main menu bar.

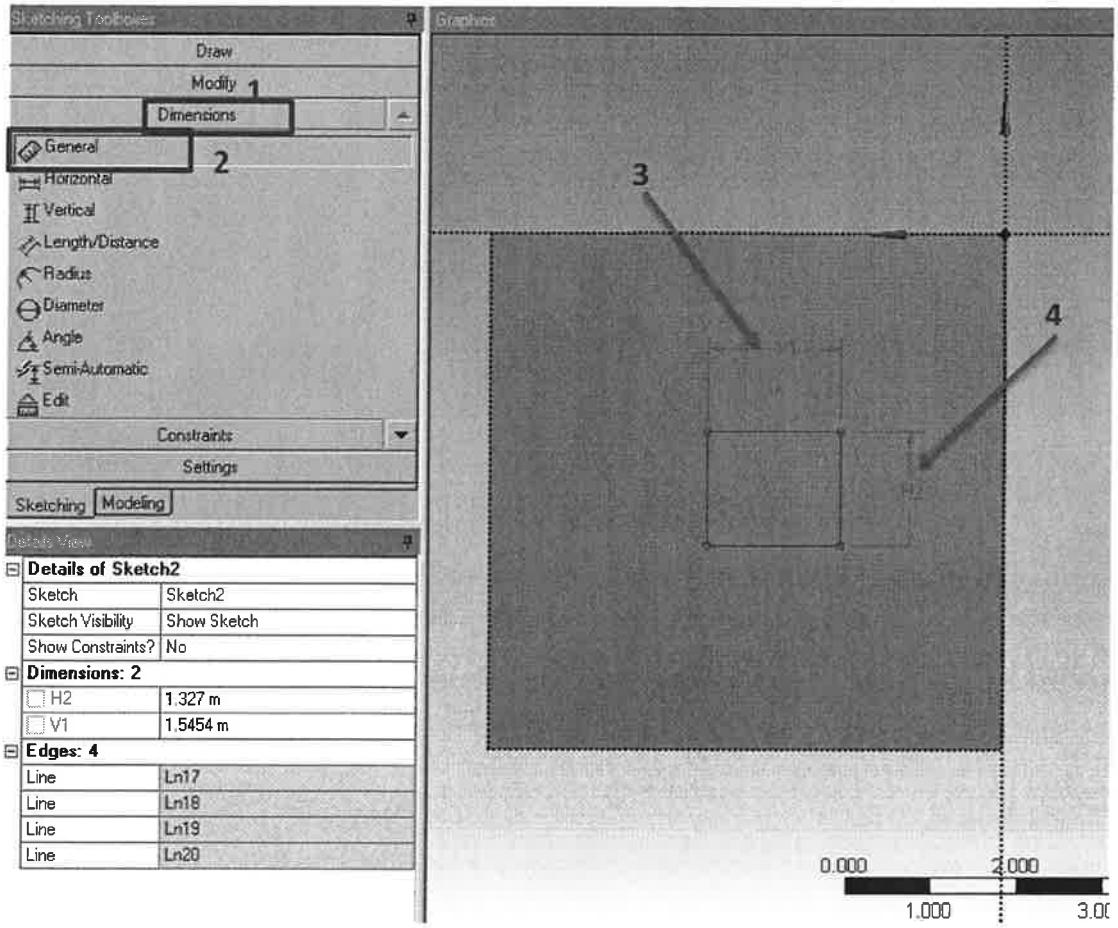


Create the opening (sketch 2) by following the procedure step by step as illustrated in the following figure. The operation is similar to the creation of Sketch 1.



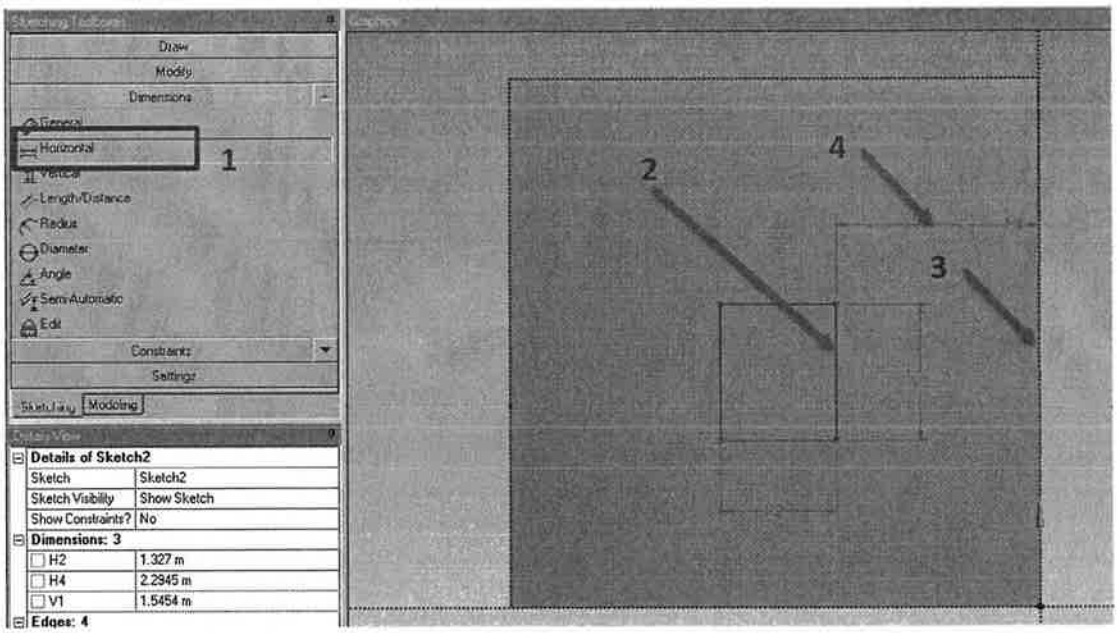
Create the dimensions for **Sketch 2**, which is located at the centre of the host face. The opening is 0.4m wide and 1.0m high. ***Extrude*** operation with the "***Imprint Face***" option is used to create the opening on the respective face.

Create the dimensions H1 and V1 using ***General*** dimension option.

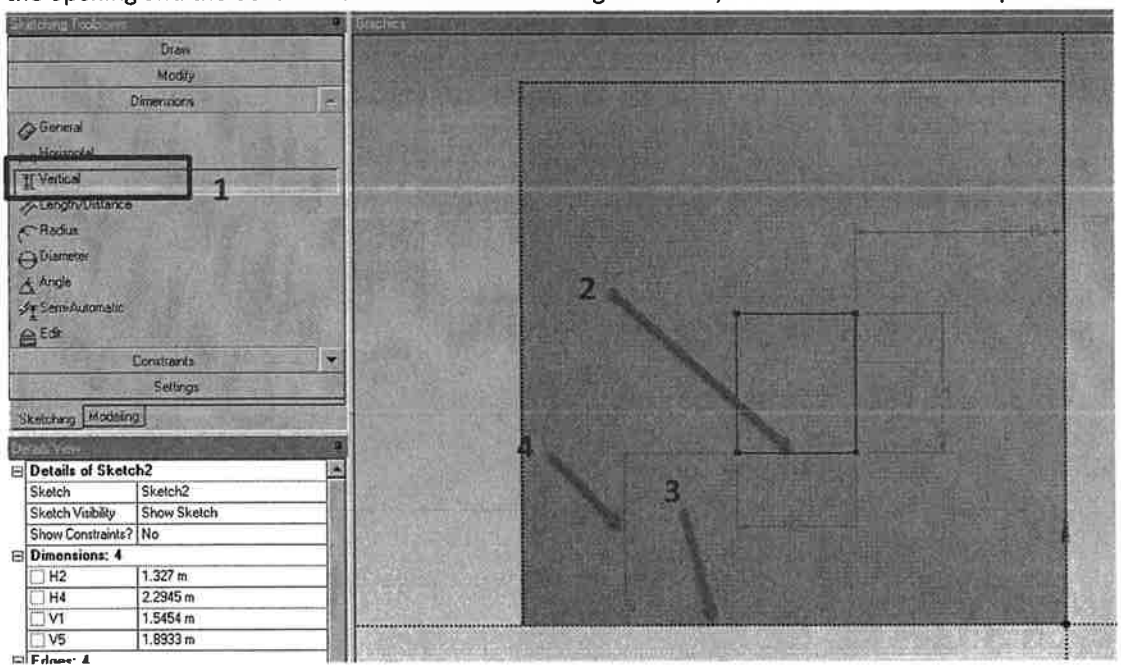


Details of Sketch2	
Sketch	Sketch2
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 2	
H2	1.327 m
V1	1.5454 m
Edges: 4	
Line	Ln17
Line	Ln18
Line	Ln19
Line	Ln20

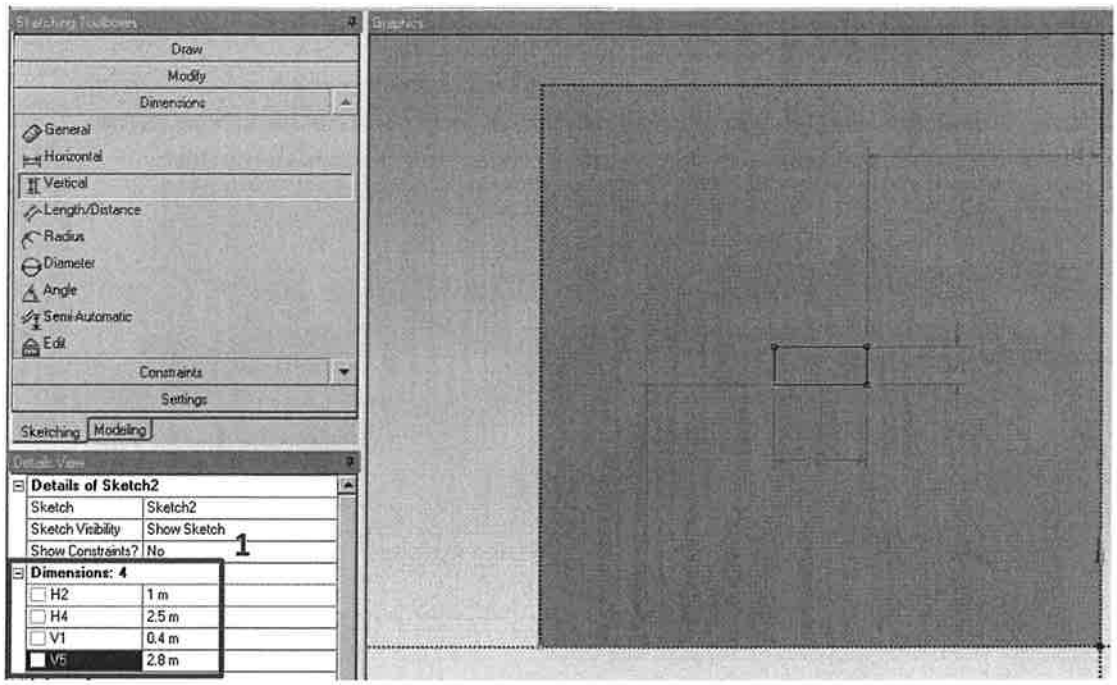
Create the dimension of H4 to quantify the distance from the bottom of the opening to the bottom of the cube, using the "***Horizontal***" dimension option. Within the working plane, the two vertical edges, as shown in the following picture, should be selected in sequence so as to create the horizontal dimension of H4.



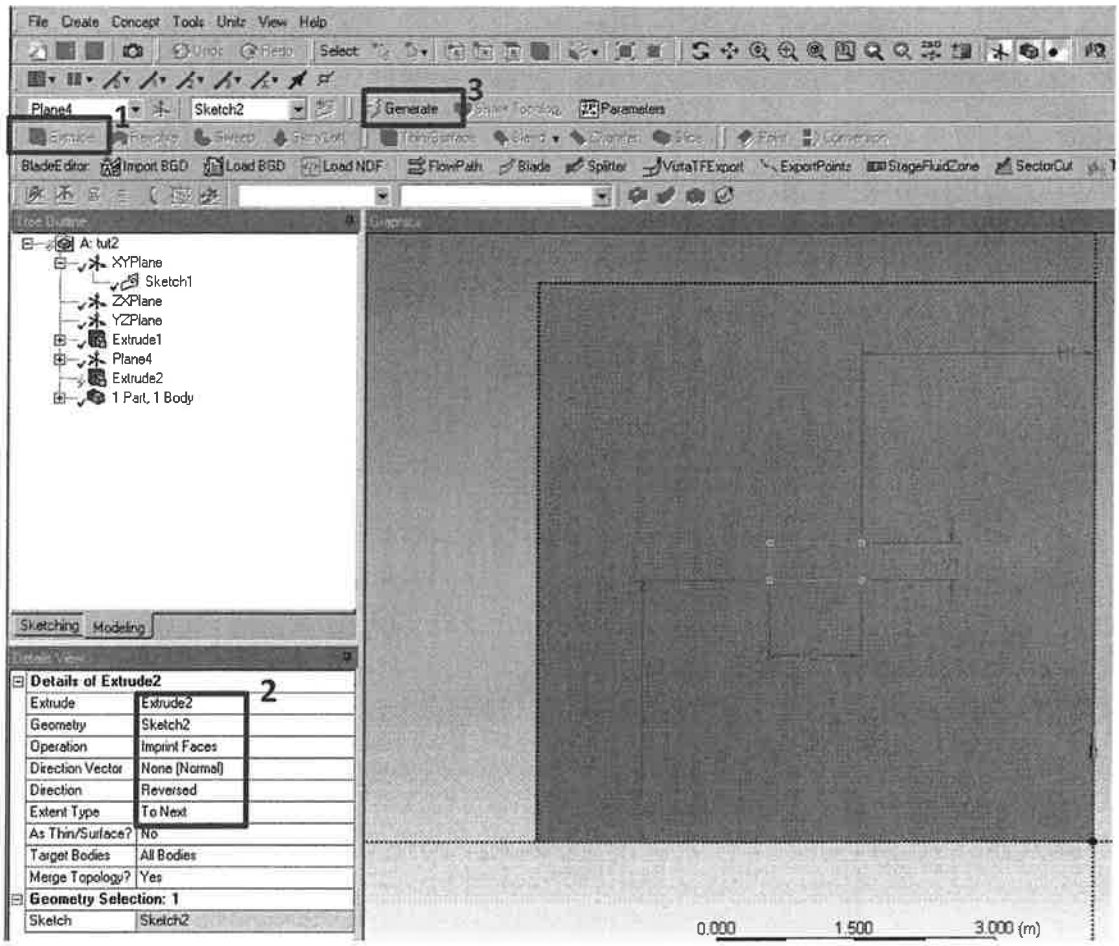
Similarly, the dimension of "V5" that quantifies the distance between the opening to the side of the cube, can be created using the "Vertical" dimension option. The two horizontal entries, one edge for the opening and the other for the cube as shown in Figure below, should be selected in sequence.



Update the dimensions as designed: H1 = 1, H4 = 2.5, V1 = 0.4, V5 = 2.8m. The settings ensure that the opening with 0.4m wide and 1.0m high is located at the centre of the face.

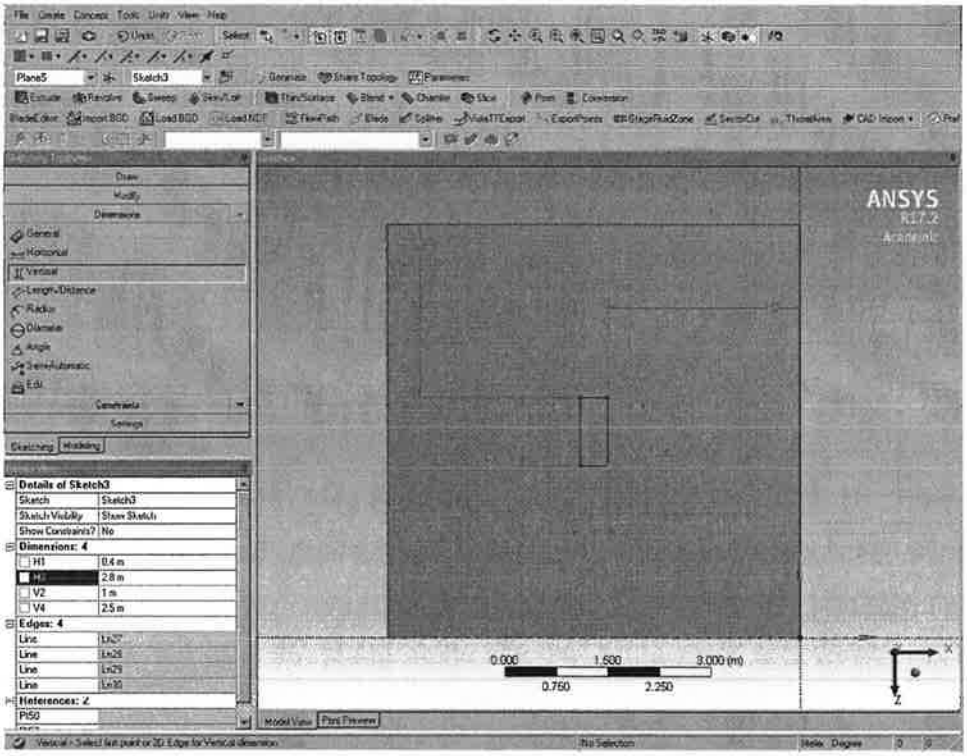


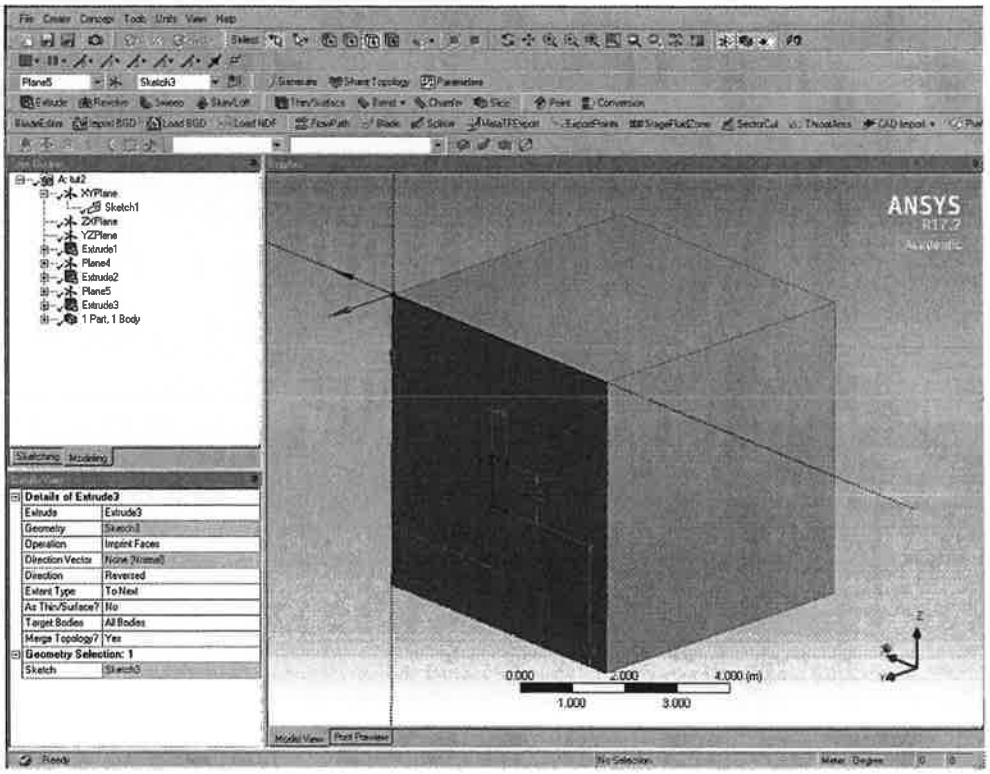
Create the opening by imprinting the *sketch 2* onto the cube face hosting the *Sketch 2*.



Repeat the procedure to create another opening on the opposite side.

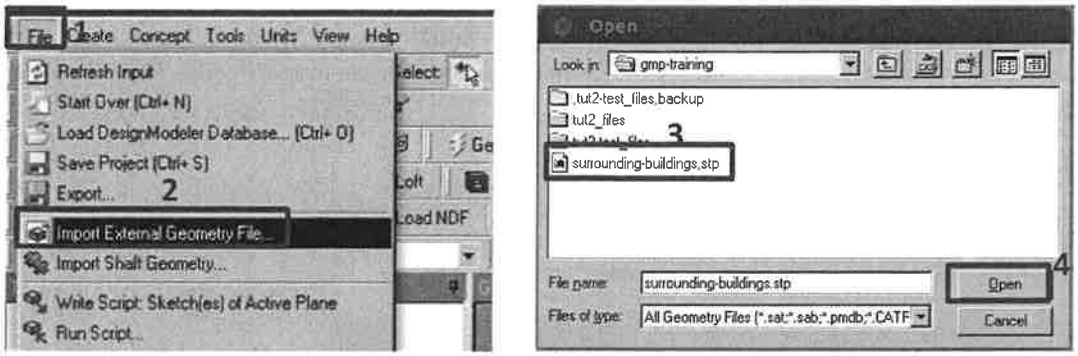
Select the opposite side face as the working plane. Apply the same dimension values to size the opening. Choose **Extrude** Operation with **Imprint Face** option to generate the opening on the opposite side face. Please make sure that the both openings have the same orientation.



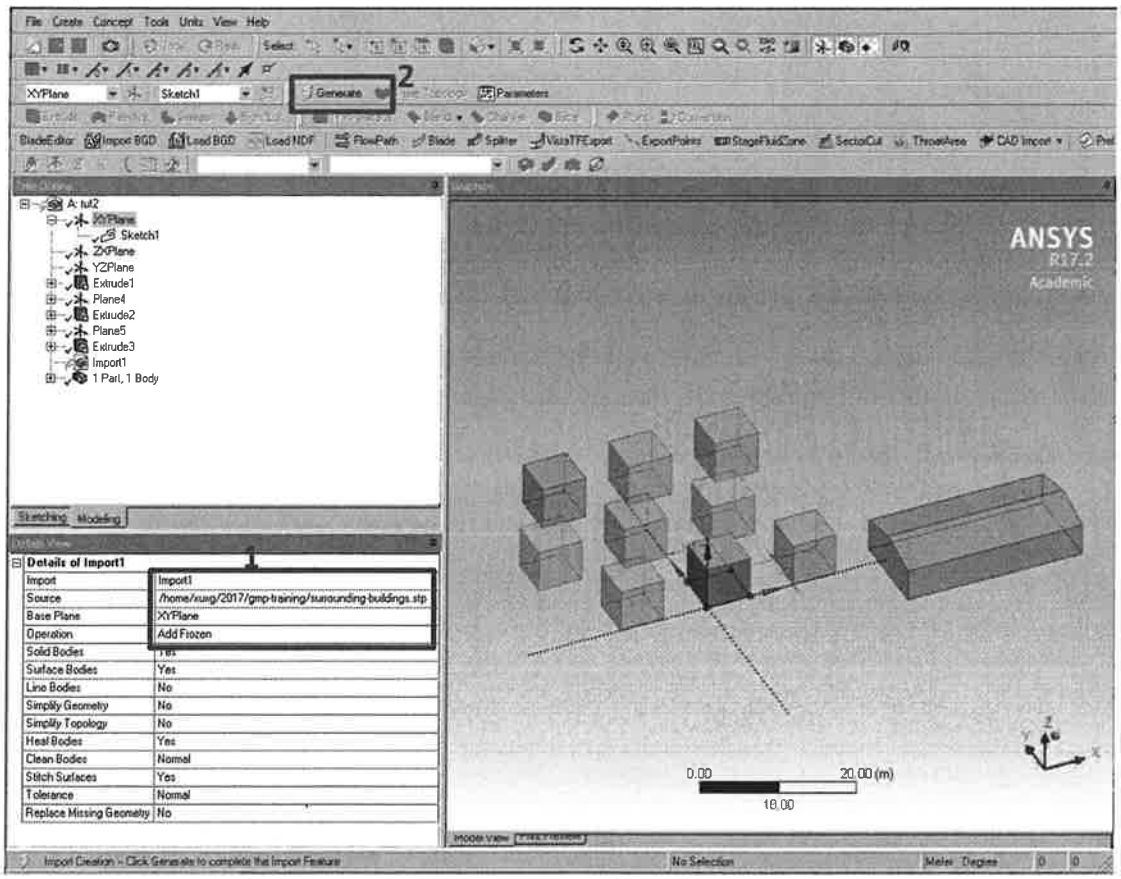


1.2 Creation of the surrounding buildings

Import the surrounding buildings by importing the external file "surrounding-buildings.stp".



Under the Detail View menu, make sure that the default XY Plane is chosen as the base plane for the assembly. Click the ***Generate*** button to load in the external buildings.



The external buildings consist of 8 cubic blocks with the same dimensions as the Silsoe Cube, and one warehouse building, as shown in the figure above.

1.3 Creation of the computational domain

In this tutorial, the prevailing wind from the north intends to be simulated. It should be noted that the computational domain is created according to the latest requirement as anticipated in GM 2015, i.e. the total blockage area due to the buildings are less than 3% of the overall cross-sectional area of the computational domain. Details about the CFD methodology can be found in GM 2015 for NRB published on BCA website.

1.3.1 Create enclosure

Choose ***Tools -> Enclosure***

Then update the settings in the Detail View side menu:

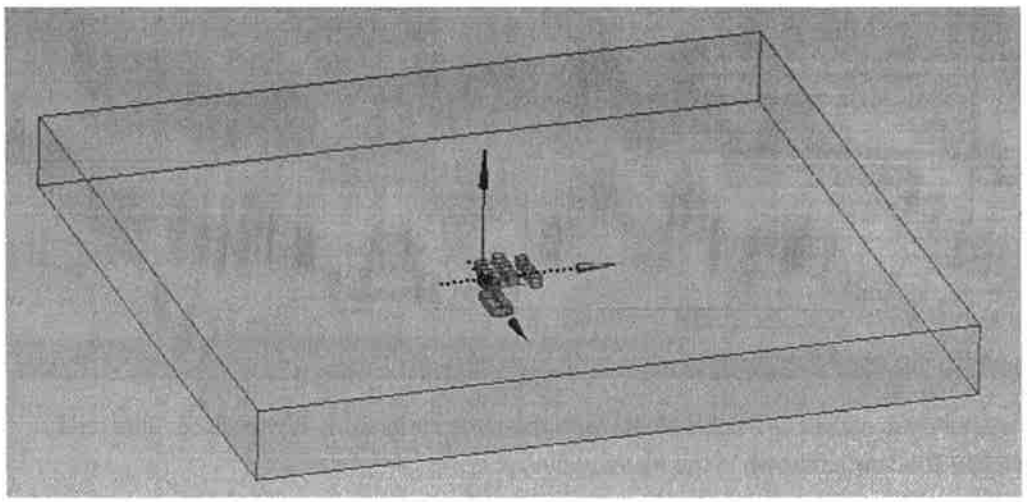
Details View

Details of Enclosure1	
Enclosure	Enclosure1
Shape	Box
Number of Planes	0
Cushion	Non-Uniform
FD1, Cushion +X value (>0)	165 m
FD2, Cushion +Y value (>0)	195 m
FD3, Cushion +Z value (>0)	36 m
FD4, Cushion -X value (>0)	165 m
FD5, Cushion -Y value (>0)	195 m
FD6, Cushion -Z value (>0)	1 m
Target Bodies	All Bodies
Export Enclosure	Yes

Finally click **Generate** button in main menu bar to create the enclosure.

Create the bounding box for all the building blocks that have been created. Note that the building blocks will be subtracted out of the enclosure with the conformal interfaces.

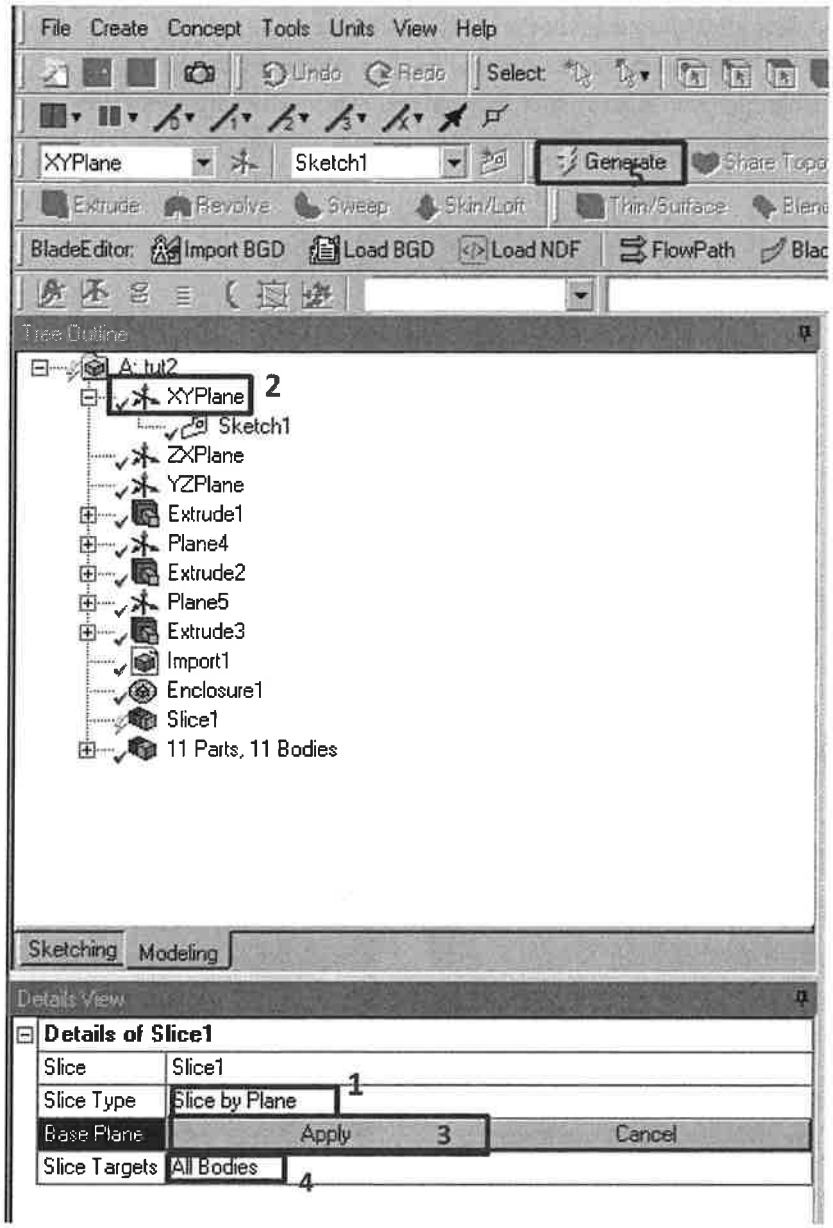
Here is what the enclosure looks like.



1.3.2 Trim the extra part (Z < 0) of the enclosure

Since the enclosure contains the lower portion (z<0), which is underneath the ground, it should be removed out of the air-filled computational domain that should be simulated in this example.

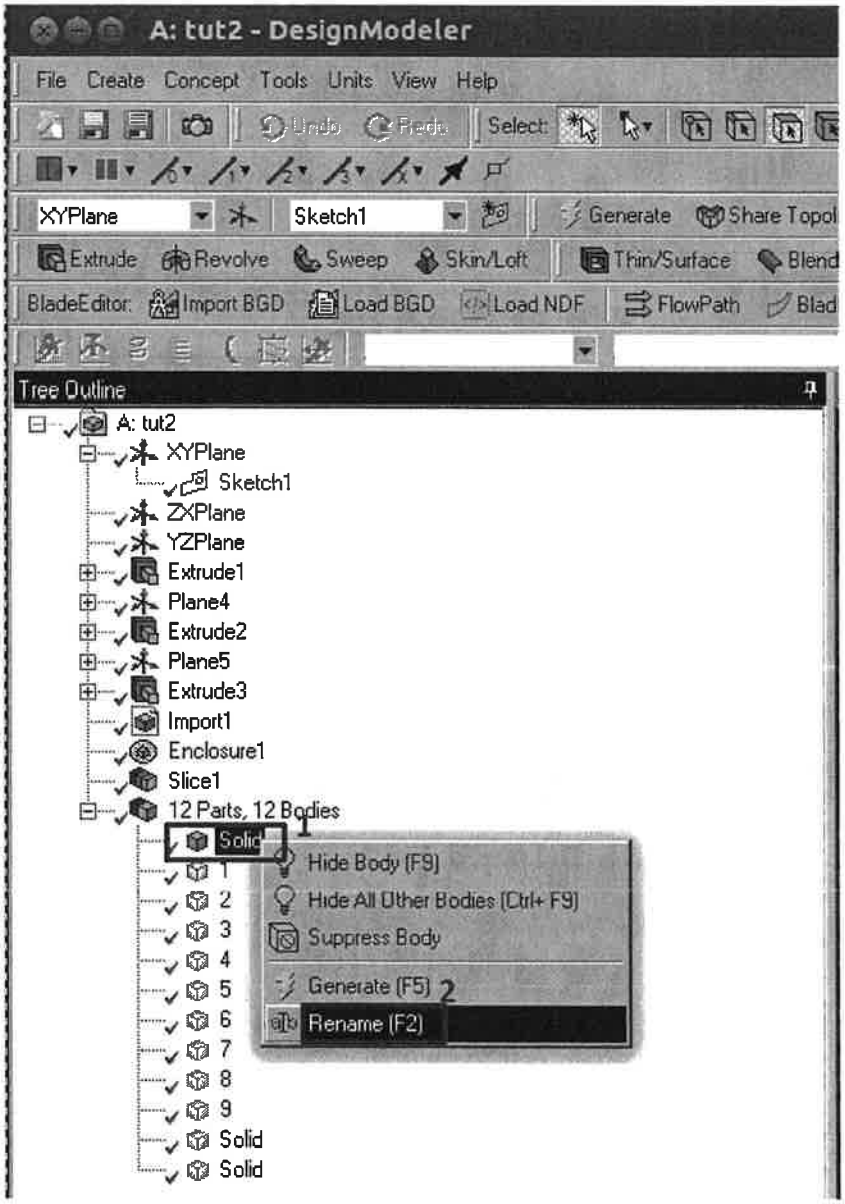
Under the drop-down main menu, choose ***Create*** → ***Slice***, then follow the setting as shown in the figure below to complete the operation.



1.3.3 Finalize the geometrical model

- Rename the inner and external computational domain (air-filled);
- Form a single part with multiple bodies for the sake of conformal meshing at the shared interfaces;
- Suppress the rest solid bodies so that they will not be meshed and simulated.

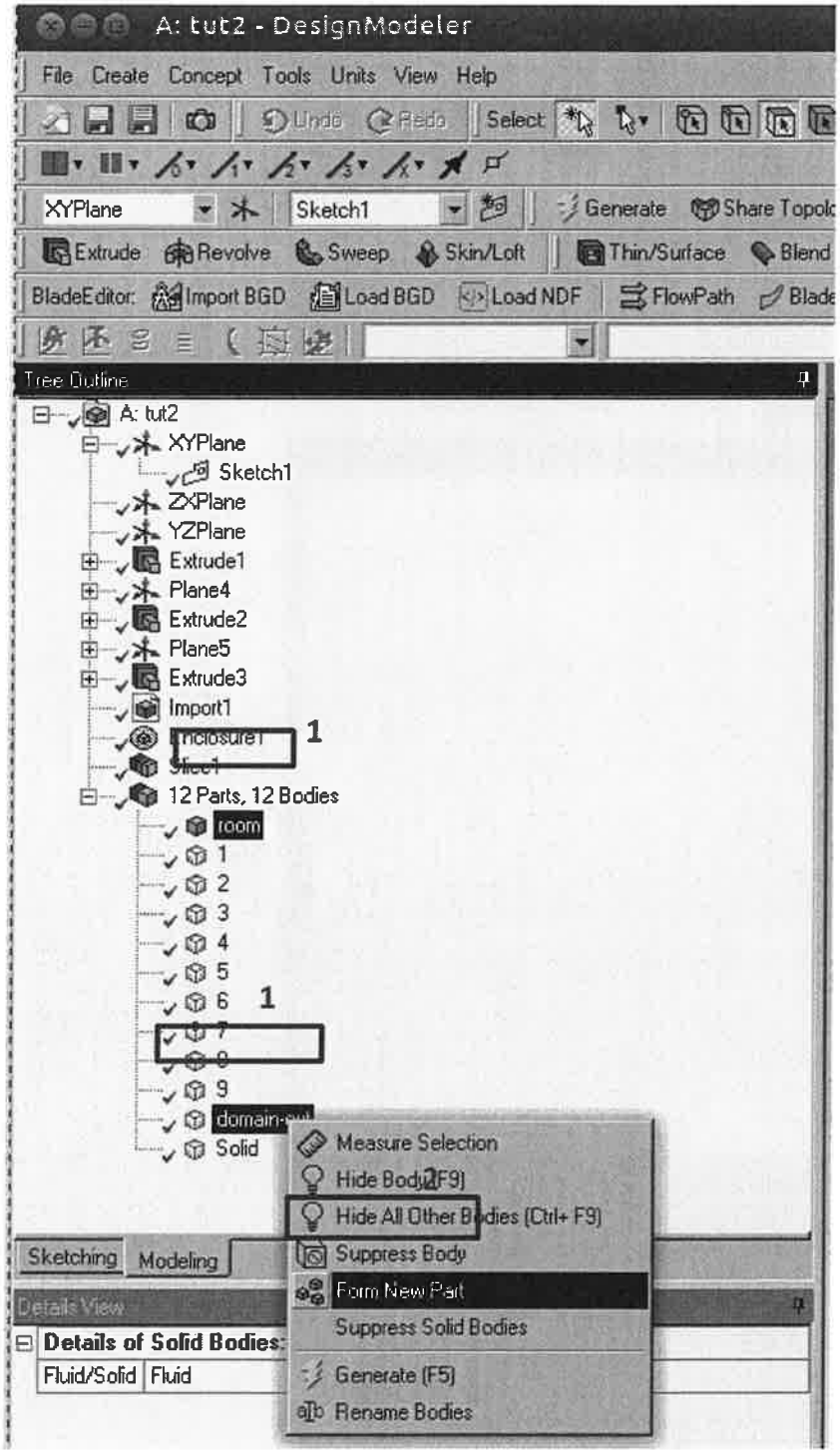
Change the name of cube to be "room". Use the left mouse buttion to click the "Solid" body in the feature tree menu, and then click the right mouse button to choose ***Rename*** function to update the name. **Please take note that the body names for your model may be different from what are shown in the tutorial, dependent on your operational proocedure. Please make sure that the correct solids that are representative to the air-filled volumes are selected and renamed accordingly.**



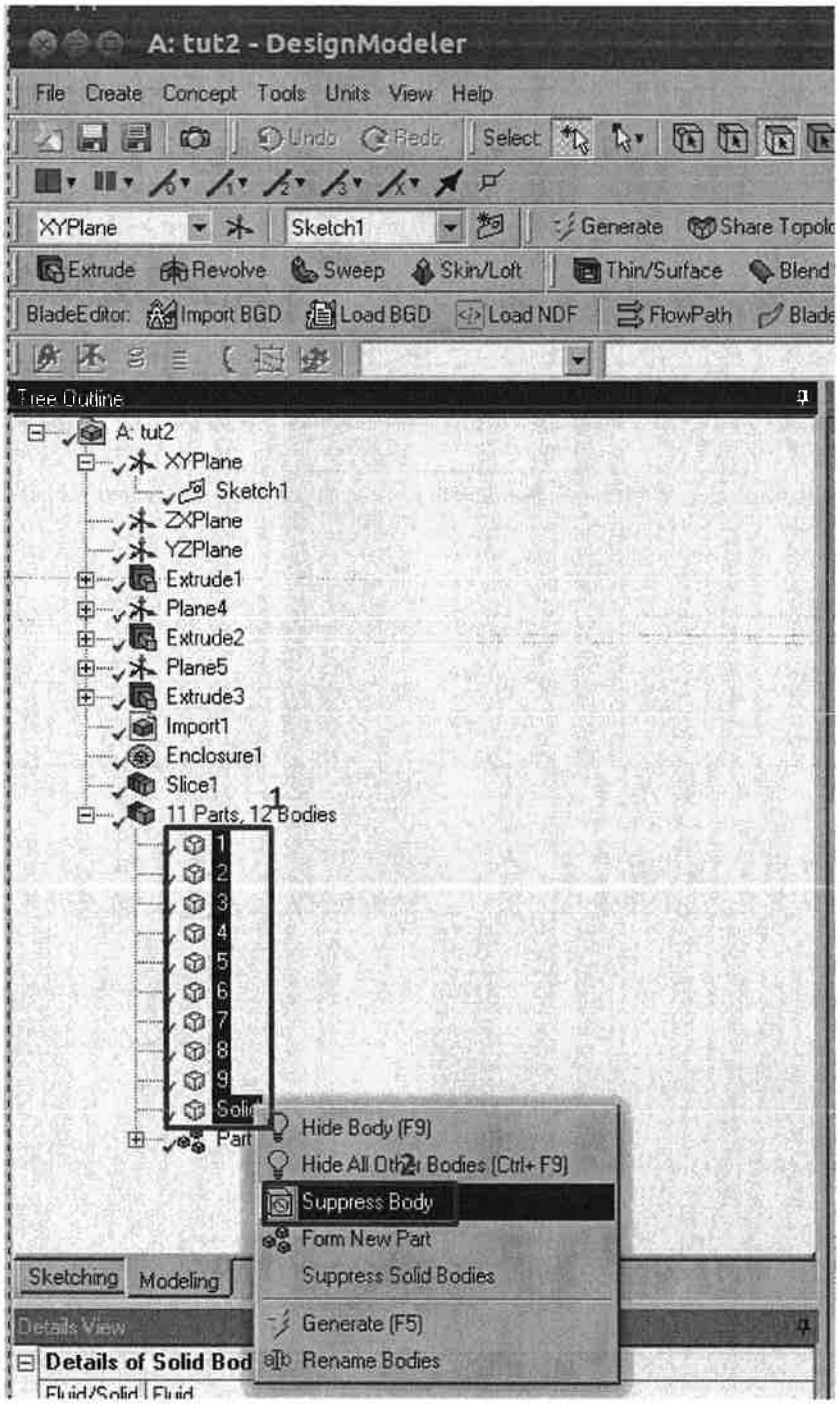
Furthermore, change the name of the sliced domain with Z ≥ 0 (larger body) to be "domain-ext".

Form a single part with multiple bodies ("room" and "domain-ext") to form the computational domain.

Selected the both body "room" and "domain-ext" with the left mouse button while holding down the **CTRL** key, then click the right mouse button to choose ***Form New Part*** option.

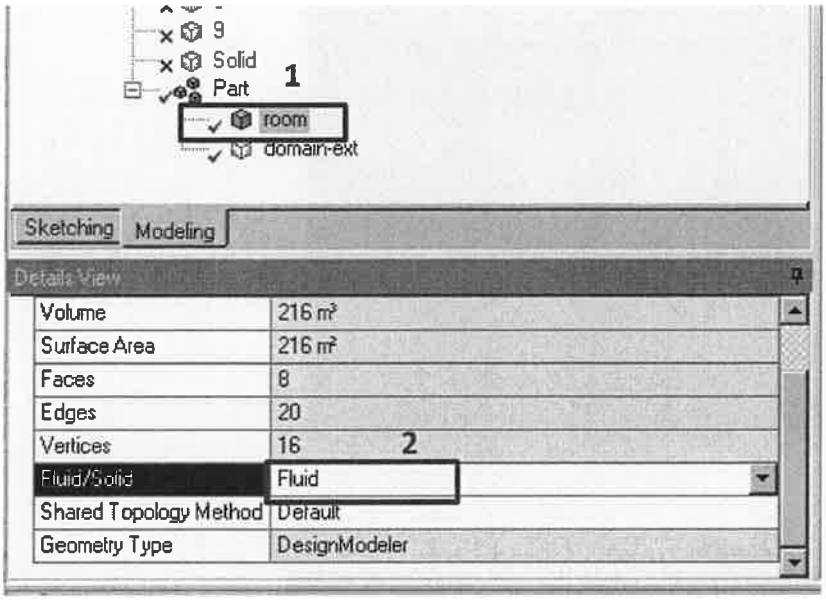


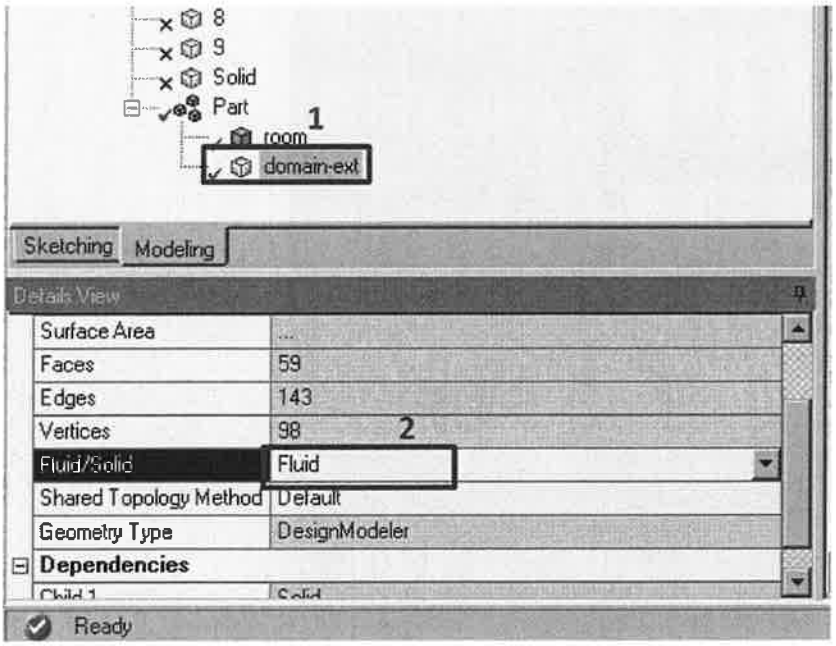
Suppress all bodies, except for the newly formed part. Under the feature tree side menu, holding CTRL key, left click all the rest bodies and then right click the mouse to choose ***Suppress Body*** option. **Please take note that the body names for your model may be different from what are shown in the tutorial, dependent on your operational proocedure.**



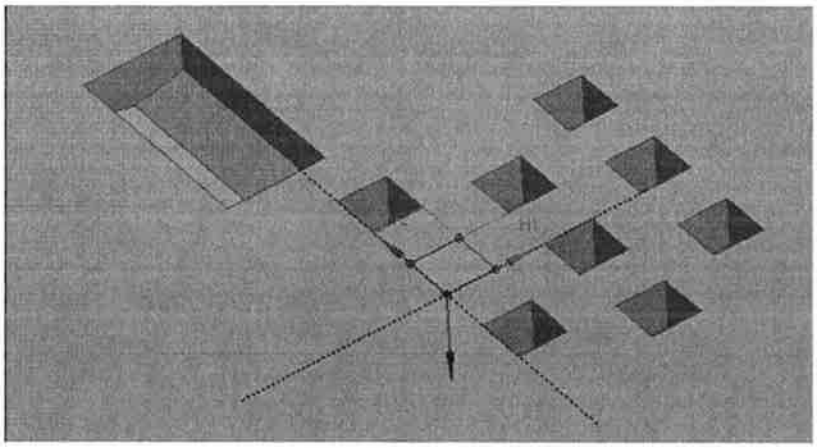
Please note that no mesh will then be created for the suppressed bodies (inside).

Change the both bodies "room" and "domain-ext" to be fluid-type.

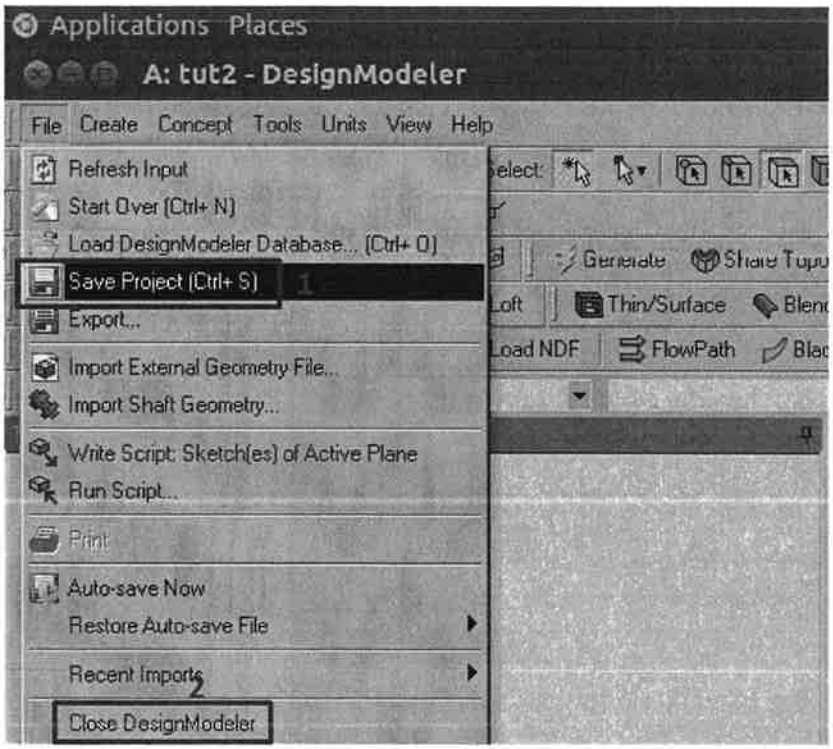




The final model should look like this (please play with the orientation to create the similar view).

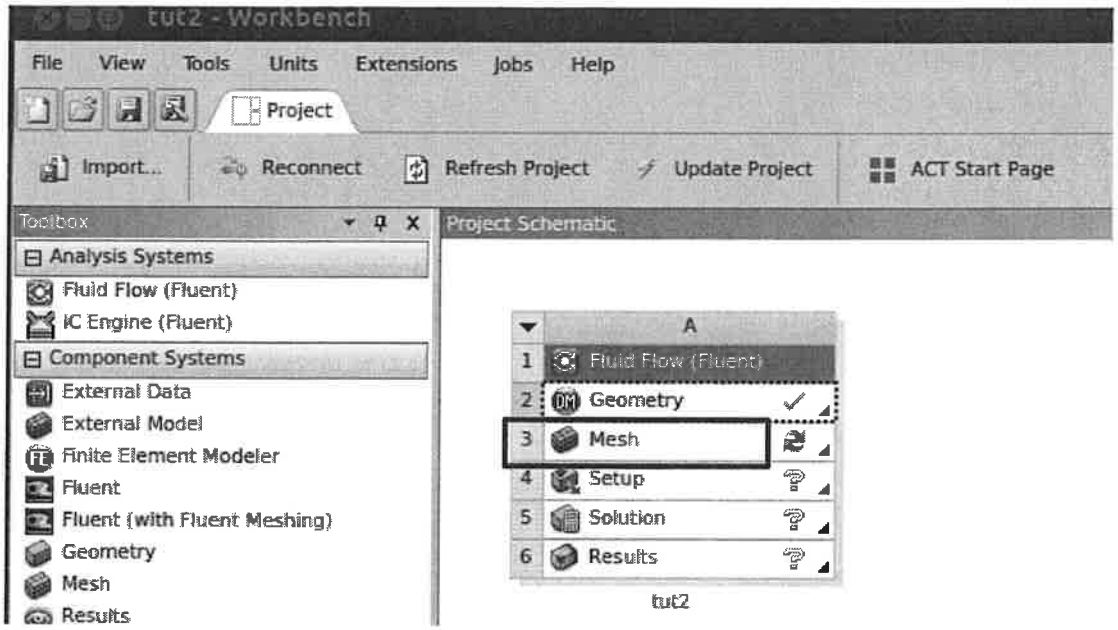


Save project and **Close DesignModeler**.



2. Mesh Generation in ANSYS Meshing

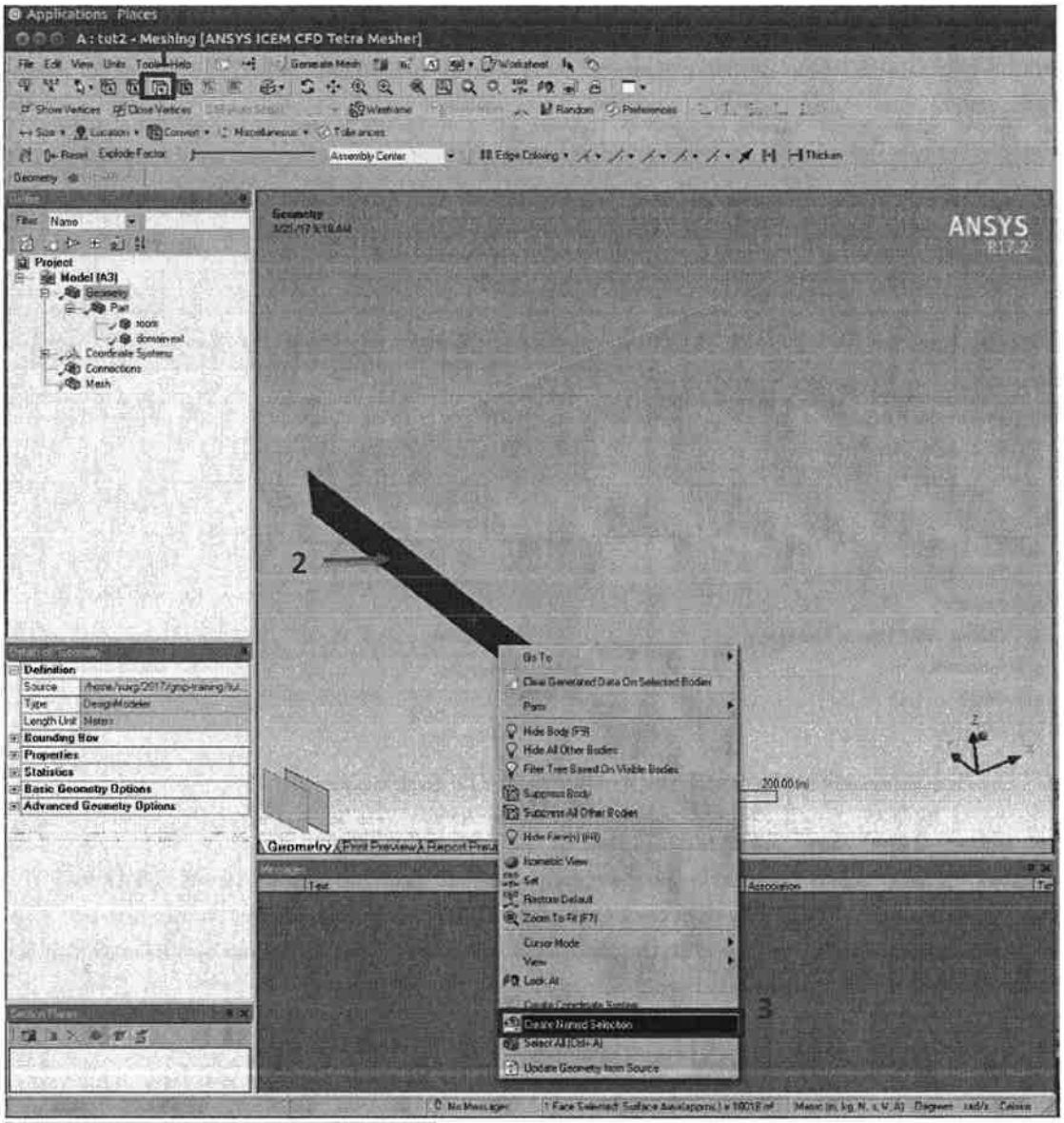
Double click Cell A3 (Mesh Icon) in the project schematic for tut2 to launch ANSYS Meshing for mesh generation.

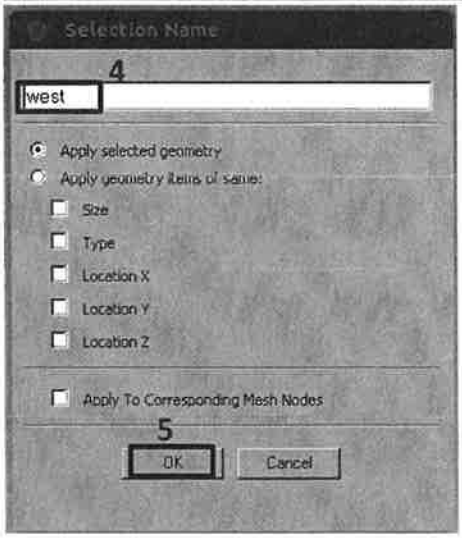


2.1 Create named selections for geometry boundaries

2.1.1 Named selections for external faces of "domain-ext" body

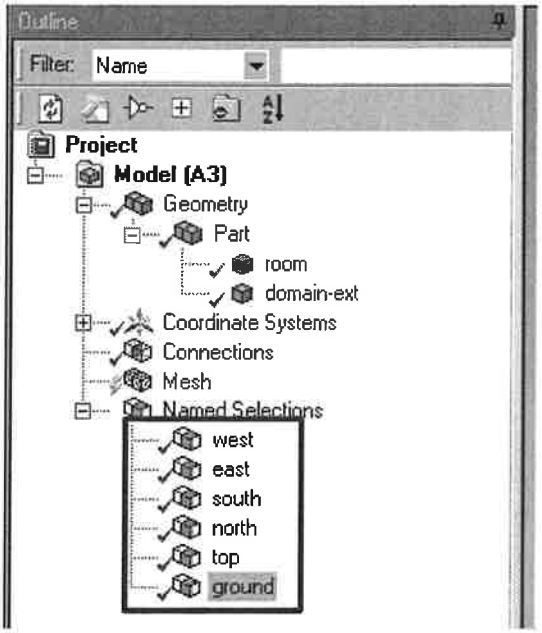
Ensure that Face-filter is selected in the main menu bar first. Click the face on the +X side of the body "domain-ext", followed by right click to choose "Create Named Selections". In the pop-up window, assign the name of "west" to the face. After that, "west" should appear in the list of Named Selections in the Tree Outline side menu.


Applications Places
A : tut2 - Meshing [ANSYS ICEM CFD Tetra Mesher]
ANSYS
R17.2
2
3
Go To
Hide Body (F3)
Hide All Other Bodies
Suppress Body
Suppress All Other Bodies
Isometric View
Reset
Restore Default
Zoom To Fit (F7)
Cursor Mode
View
Look At
Create Named Selection
Select All (Ctrl+ A)
Update Geometry from Source


Selection Name
4
west
Apply selected geometry
Apply geometry items of same:
Size
Type
Location X
Location Y
Location Z
Apply To Corresponding Mesh Nodes
5
OK
Cancel

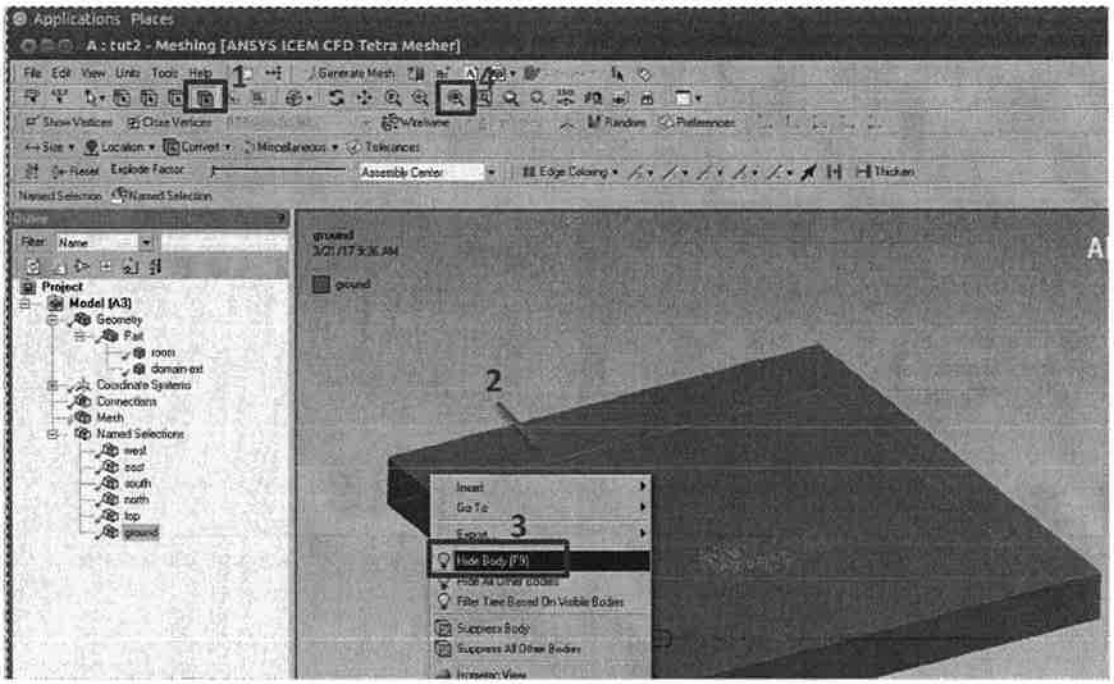
Repeat the process to create the following named selections for the external domain "domain-ext":

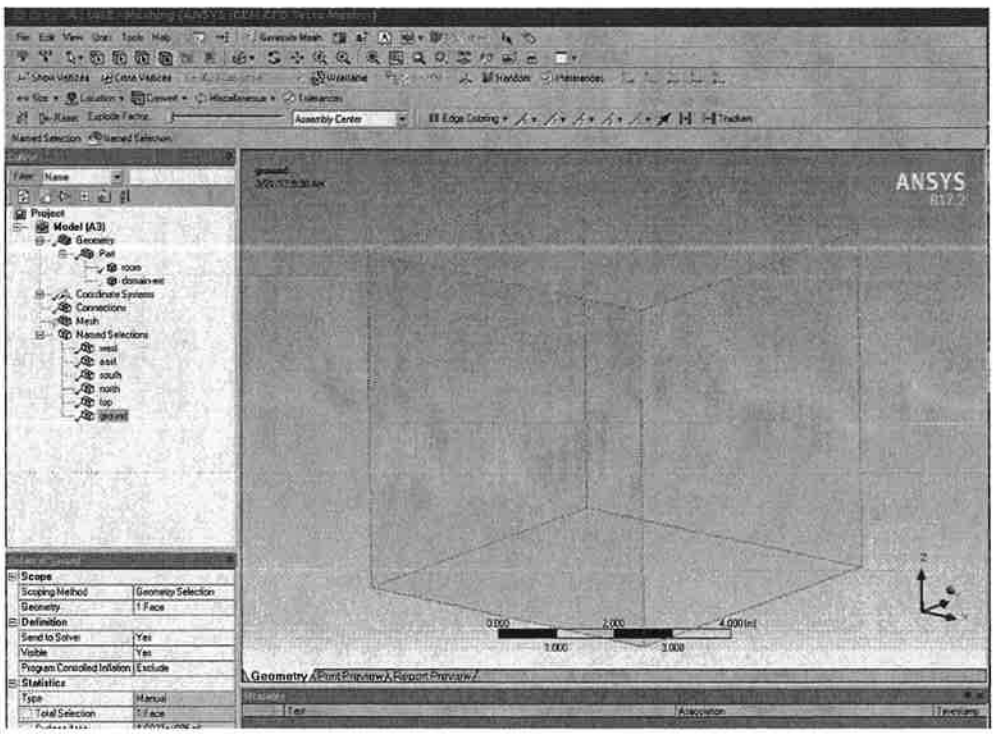
Serial number	Location	Named Selection
1	-X	east
2	+Y	north
3	-Y	south
4	+Z	top
5	-Z	ground



2.1.2 Named selections for the faces of "room" body

Hide Body "domain-ext" in order to easily access the faces of "room" body. Choose the "**Body-Selection**" filter, and then click anywhere of the external body and right click to choose **Hide Body**. Thus, only the body "room" is shown in the screen.





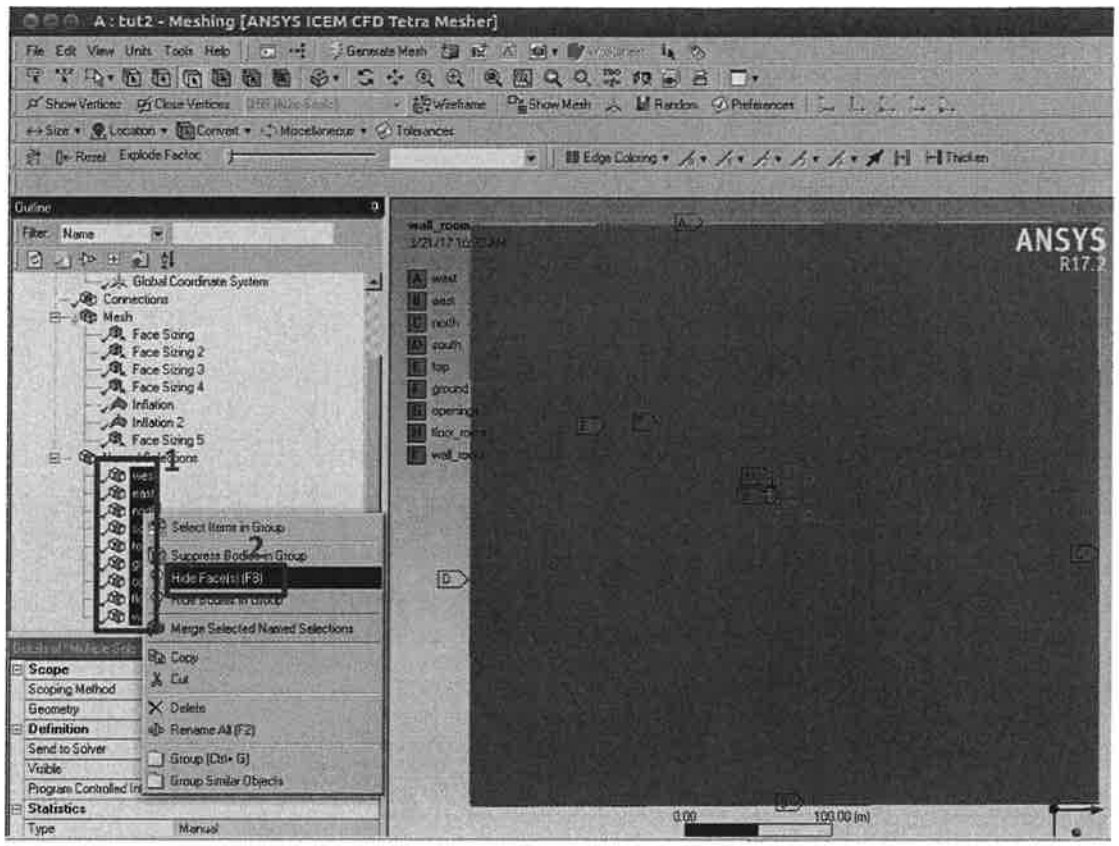
Similarly, name the respective face groups as follows:

Serial No	Locations	Name	Number of faces
1	openings	openings	2
2	-Z	floor_room	1
3	+Z, -X, +X, -Y, +Y	wall_room	5

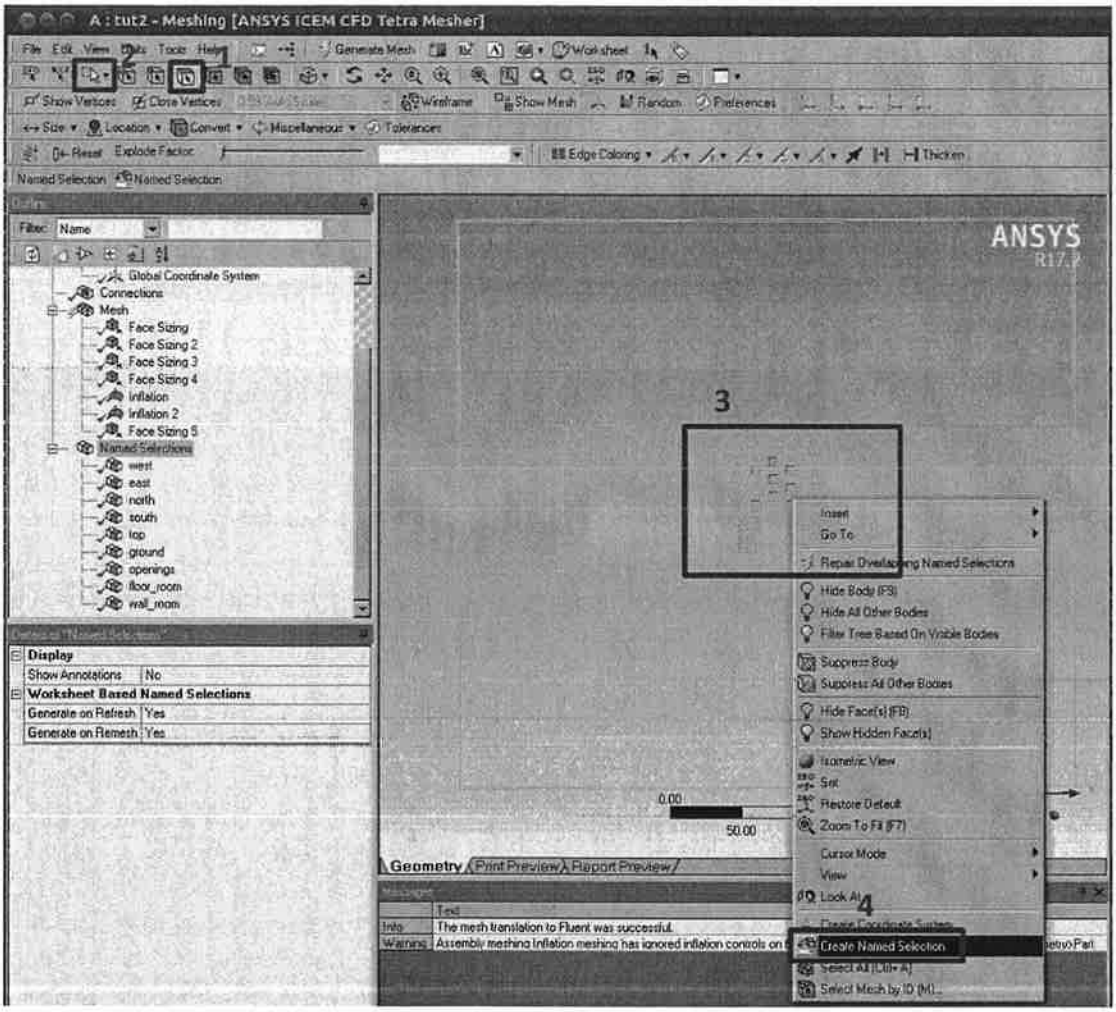
Please note that hold the **CTRL** key for multiple selections.

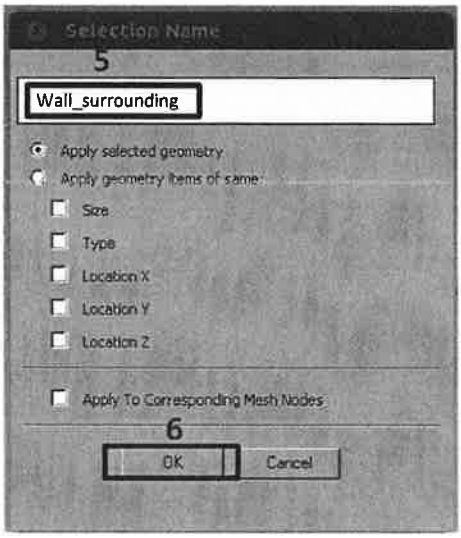
2.1.3 Definition of Named Selection for surfaces of surrounding buildings

Hide the defined named selections first. Select all the named selections in the Tree Outline side menu, with the help of **Shift** key. Right click mouse button to choose **Hide Face(s)** to hide the selected faces.



Fit the plan view (view from Z direction) within the graphic field, use window-selection mode to draw a rectangle box to contain all the faces of all surrounding buildings, and then right click the mouse button to define the named selection of "wall_surrounding".

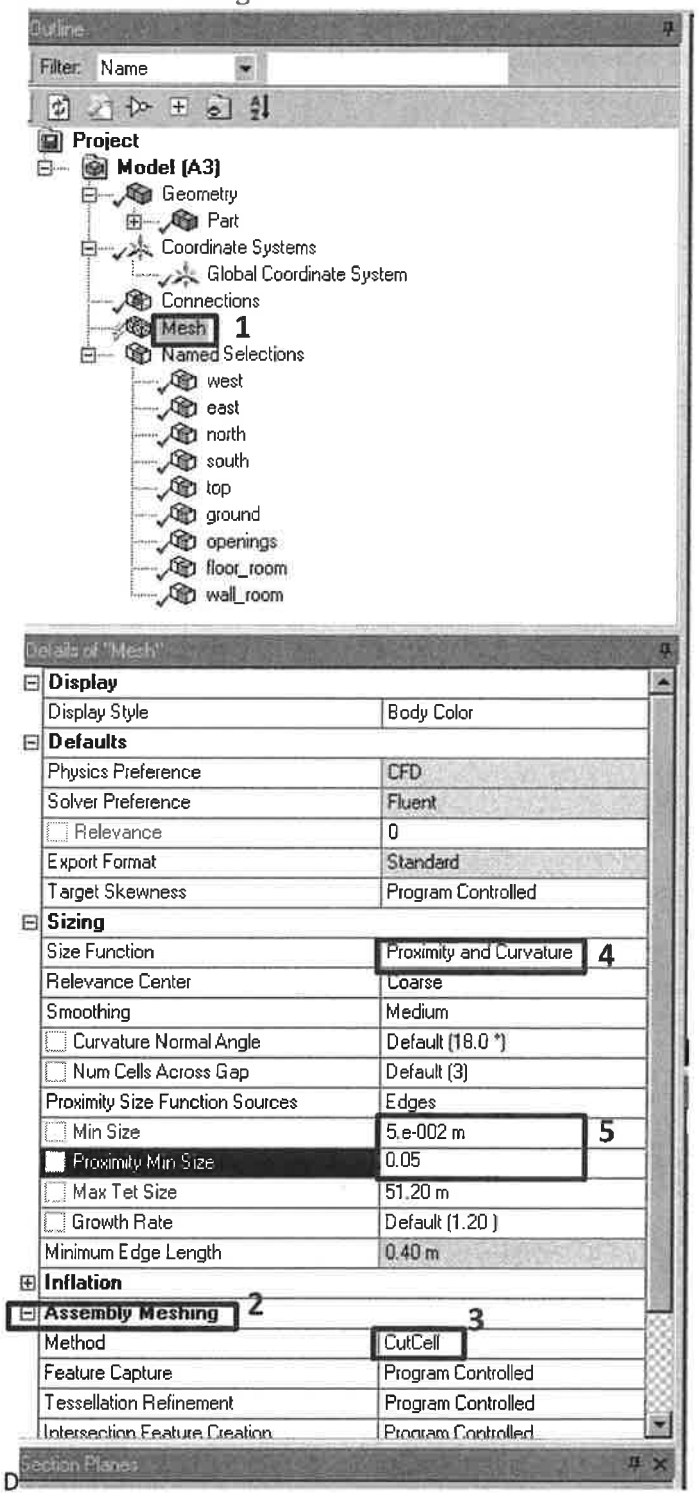




2.2 Mesh size control

Hexa-dominated Cutcell mesh is suggested to be used for such a large scale simulation, with the concerns of computational efficiency.

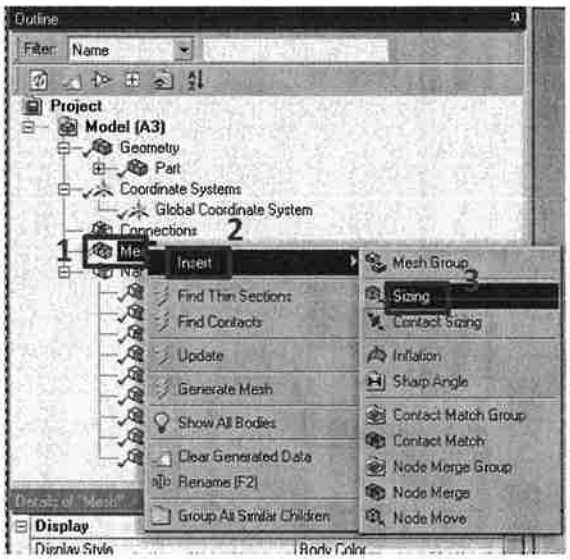
2.2.1 Global settings

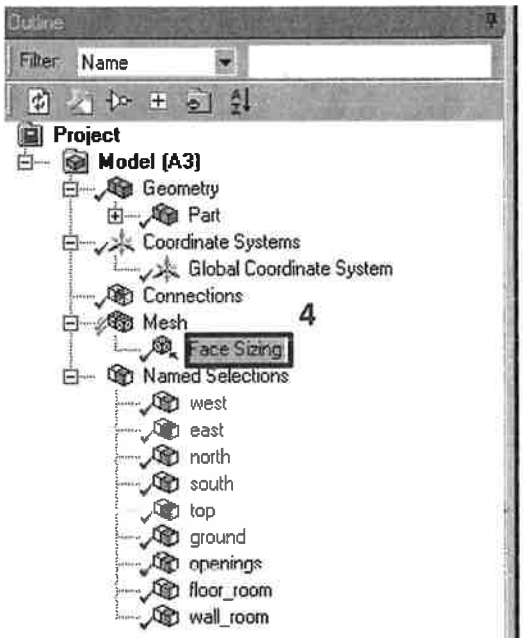


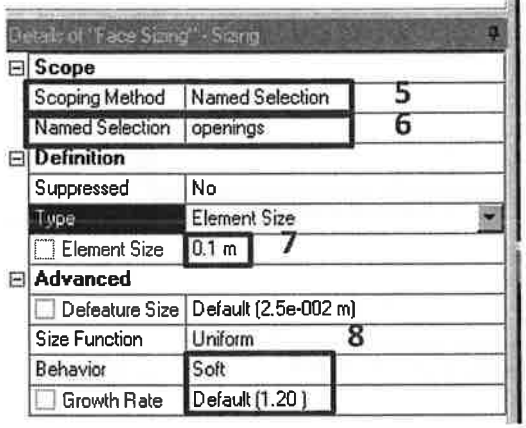
2.2.2 Face size functions

Green Mark Code has various mesh sizing requirements for different design features, thus, it is handy to adopt size functions to achieve the controls of meshing size distribution spatially.

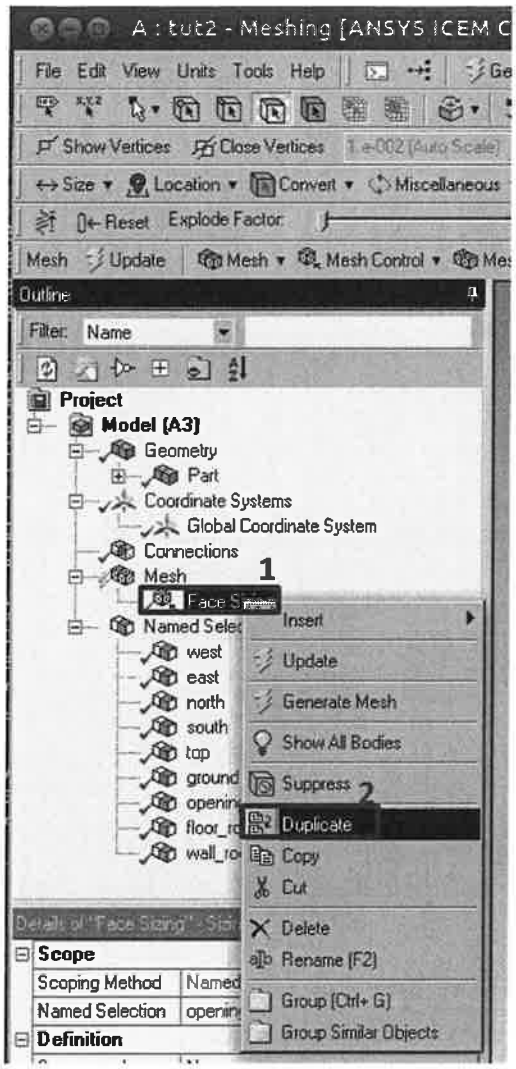
Under the **Outline** side menu, choose **Mesh** and then right click the mouse button to choose **Insert -> Sizing**. Within the menu for **Details of "Face Sizing"**, set the element size to be 0.1m for "openings" Named Selection, by choosing **soft**-type size function behaviour, with which the size of surface elements to be generated on the selected faces should be no more than 0.1m.

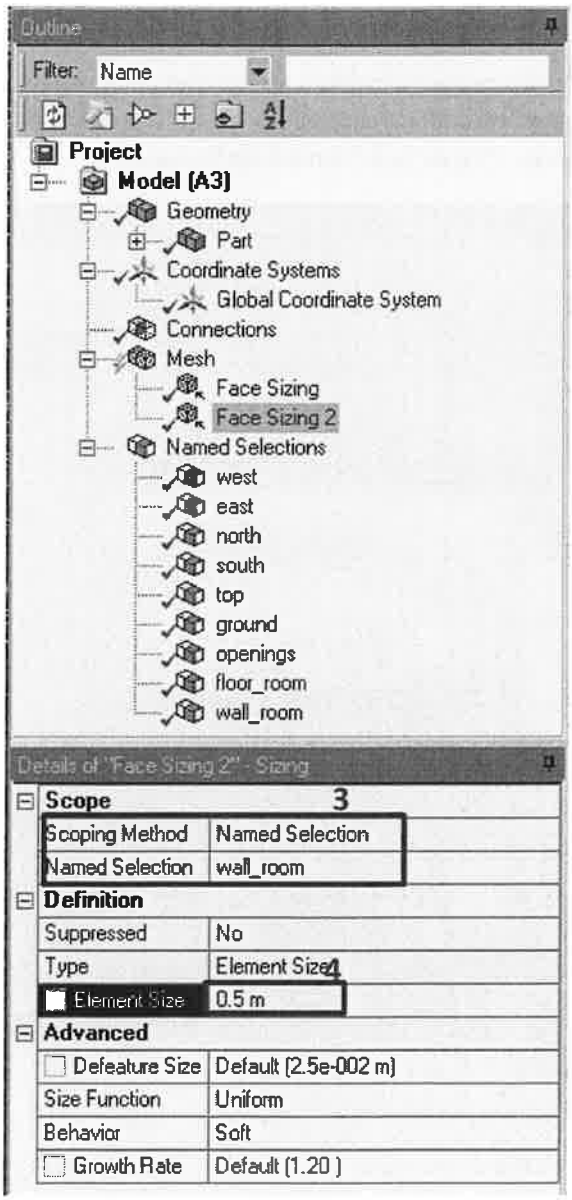






Continue to create the sizing functions for other design features with the help of **Duplicate** operation. Right click the "Face Sizing" just defined to choose **Duplicate** operation. Then a new size function called "Face Sizing 2" will be added in the list of size functions. Set the surface meshing size of 0.5m for Named Selection "wall_room".





Continue to use **Duplicate** operation to set the meshing sizes for Named Selections of

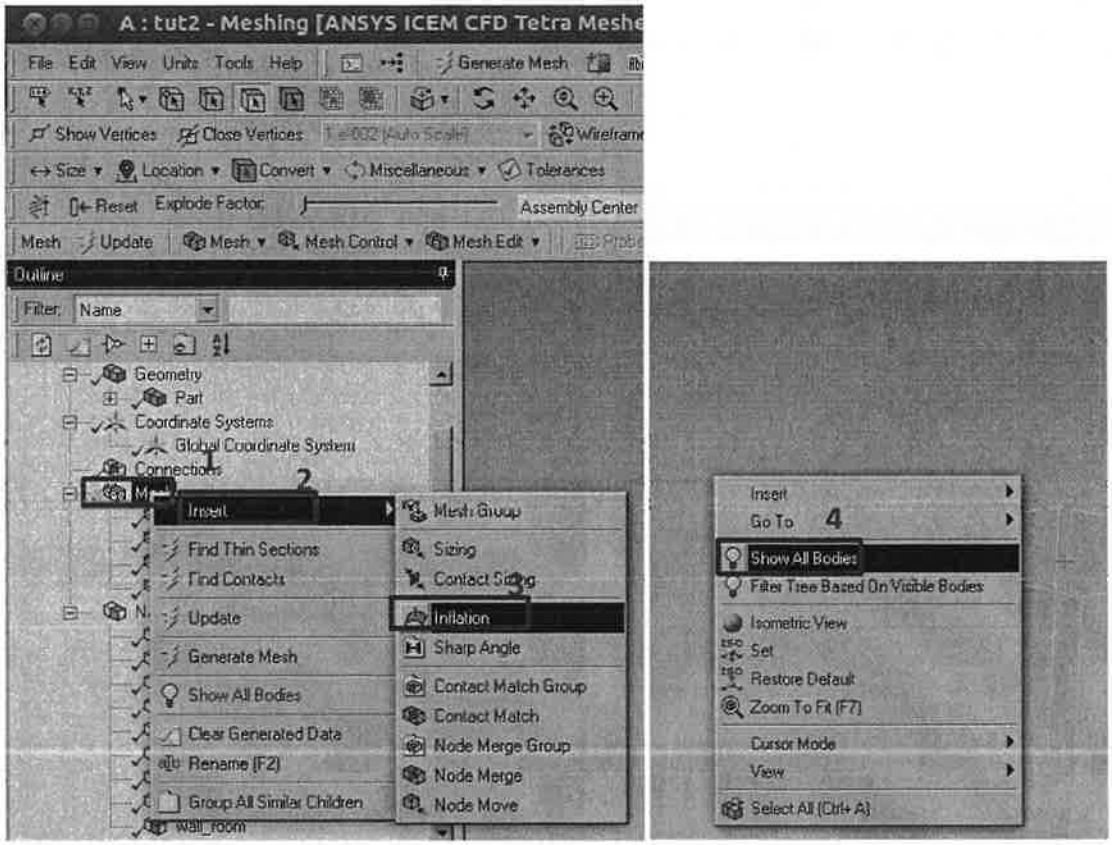
- "floor_room" (Element Size of **0.2**m);
- "wall_building" (Element Size of **1.0**m) and
- "ground" (Element Size of **5.0**m).

2.2.3 Inflation layers

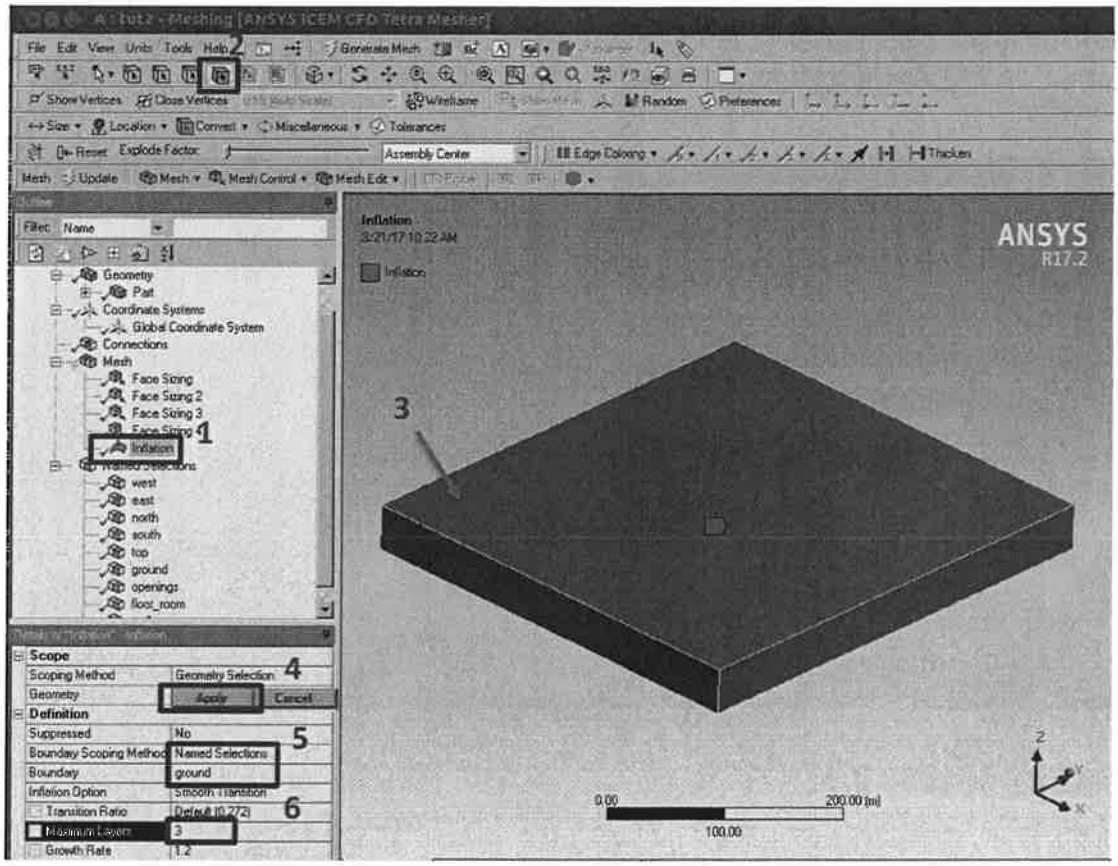
In order to well capture the atmospheric boundary flow, it is advisable to include finer grids to capture the velocity changes within the boundary layers near the ground. The cost-effective way is to add inflation layers during the meshing generation.

Under the Outline menu, right click **Mesh** to choose **Insert -> Inflation**.

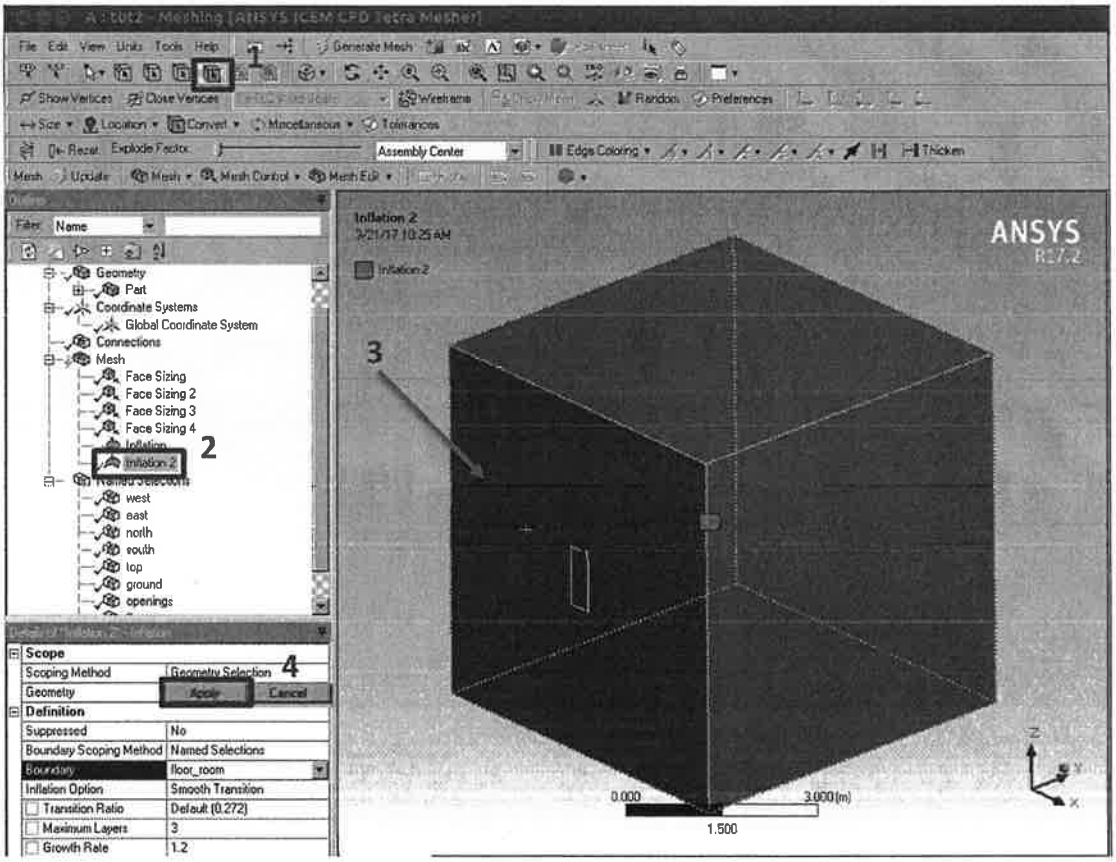
Make sure all the bodies are displayed on the screen, otherwise, please right click the mouse button anywhere within the Graphic view field to choose the option **Show All Bodies** to make the both bodies ("room" and "domain-ext") visible.



For the "Inflation" definition, left click anywhere of the body "domain-ext" in the graphic view field and then click the **Apply** button in the **Details of "Inflation"** side menu to set it as the geomtry where the inflation layer to be generated and then, and then select Named Selection "ground" as the source face from which three (**3**) inflation layers grow inwardly into Body "domain-ext".

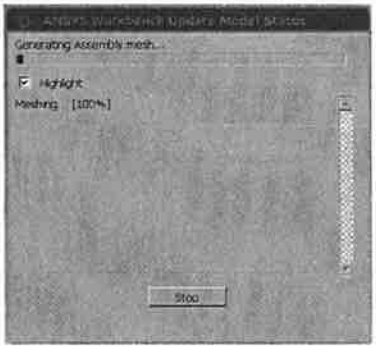


Again, Use **duplicate** operation for "Inflation" to create "Inflation 2", as what we done for "Face Sizing" previously.
In order to have access to Body "room", we need to hide the body "domain-ext". Make sure that the Body-selection filter is on first. Left click anywhere of Body "domain-ext" in the Graphic View Field and then right click the mouse button to choose **Hide Body** option to make it invisible. Fit the view to see Body "Room".
Follow the same procedure as for "inflation" definition to update the settings for "Inflation 2" by setting the 3 inflation layers to be generated from Name Selection "floor-room" inwardly into Body "room".



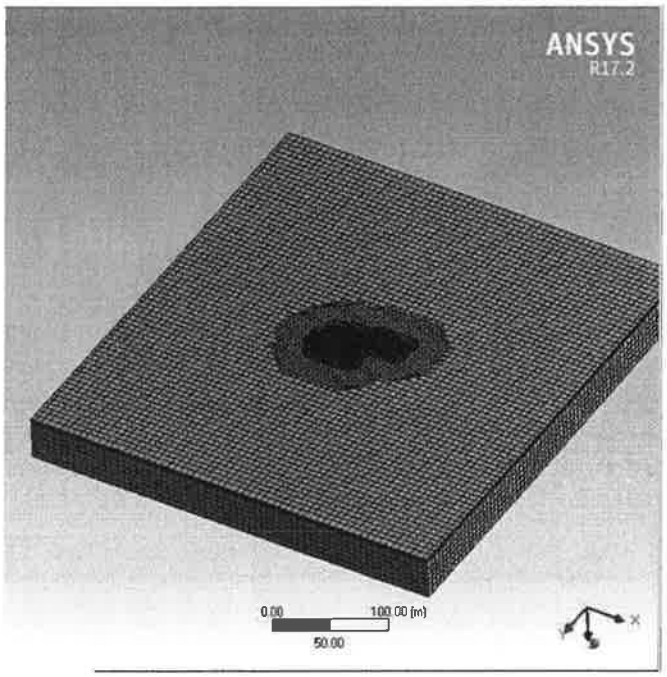
2.3 Grid generation

Save the project, and then choose **Mesh -> Update** to start the cutcell mesh generation.

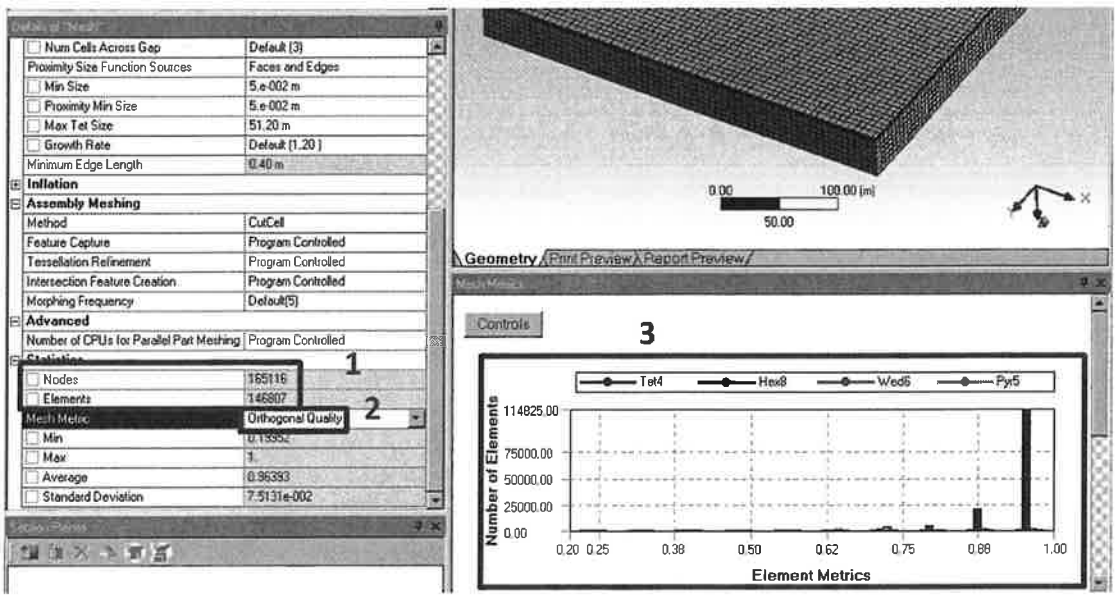


2.4 Mesh checking

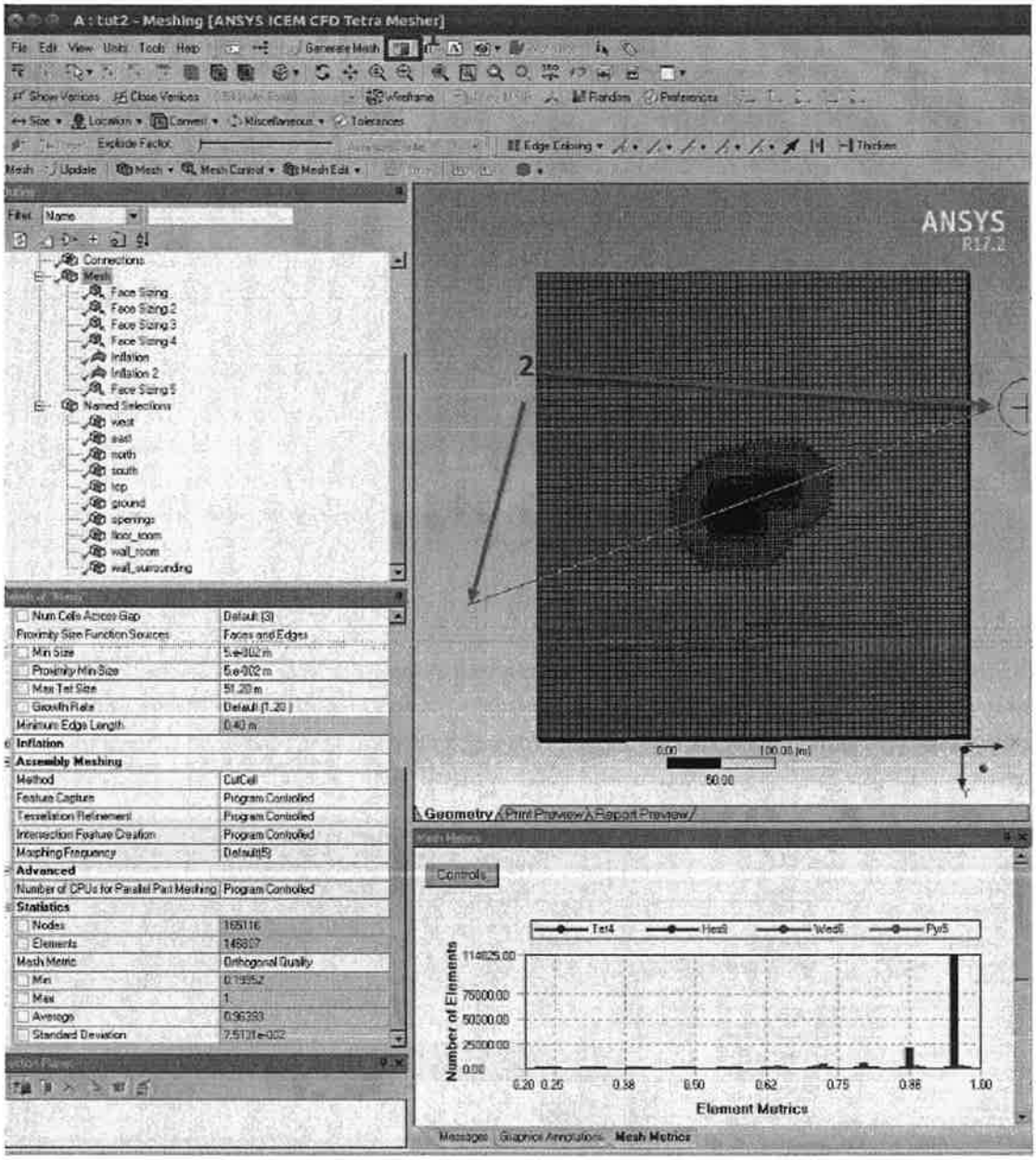
Finer meshes are expected to be created within and around the body "room". The mesh size gradually grows outwards. Three inflation layers should be generated from the plane of Z=0 upwards.

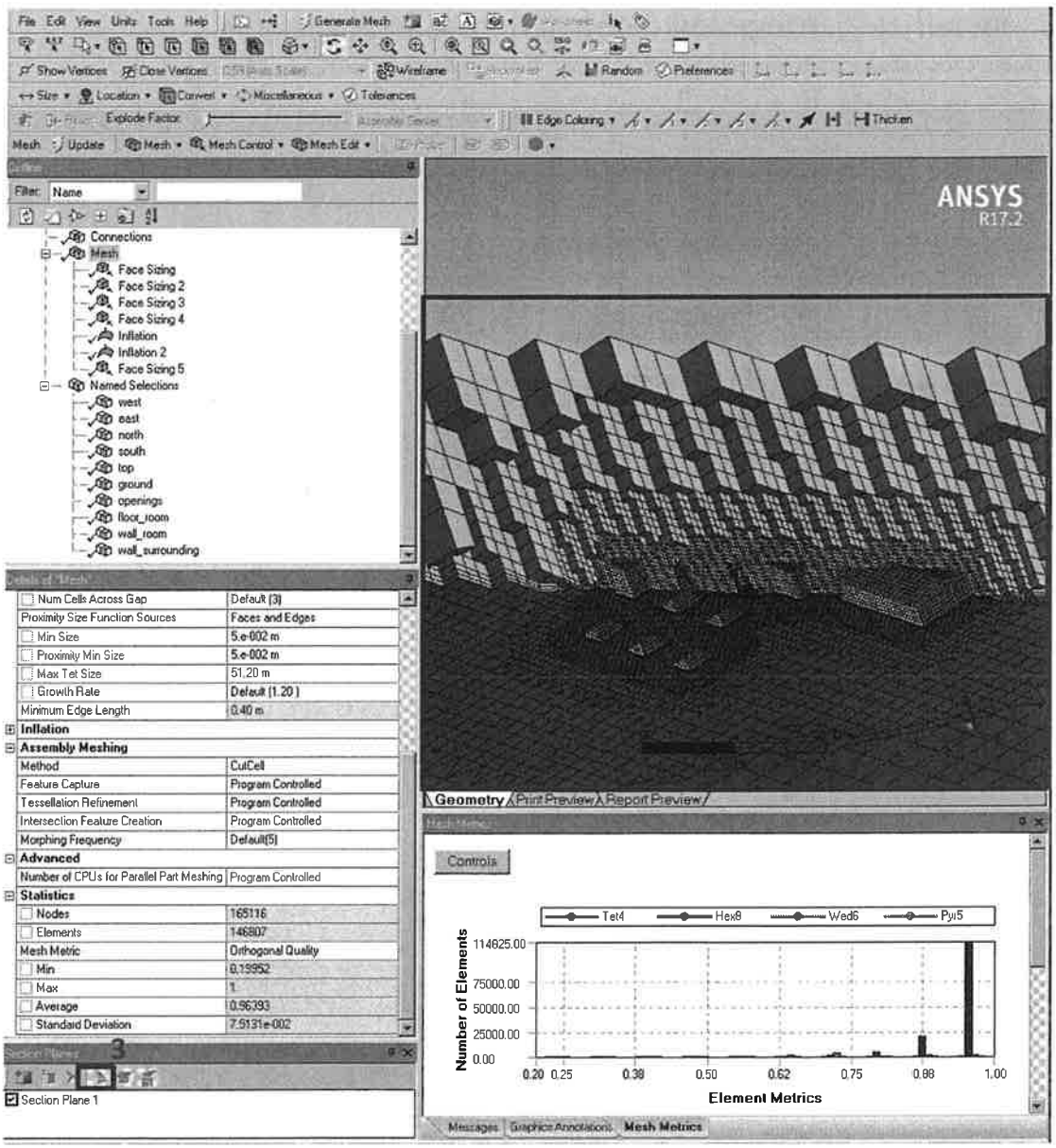


The total number of nodes and elements are summarized in **Statistics** Infos. You can check about the mesh quality. For instance, when the "Orthogonal Quality" is chosen as the Mesh Metric, you can examine the distribution of the number of elements under different range based on Orthogonal quality.



You can also visualize the spatial distribution of cutcell elements across the domains with the help of **Section Plane** tool. Choose the tool and then click two locations on the screen to form a cutting plane normal to the screen. Then you are able to visualize the element distribution cross the section plane.

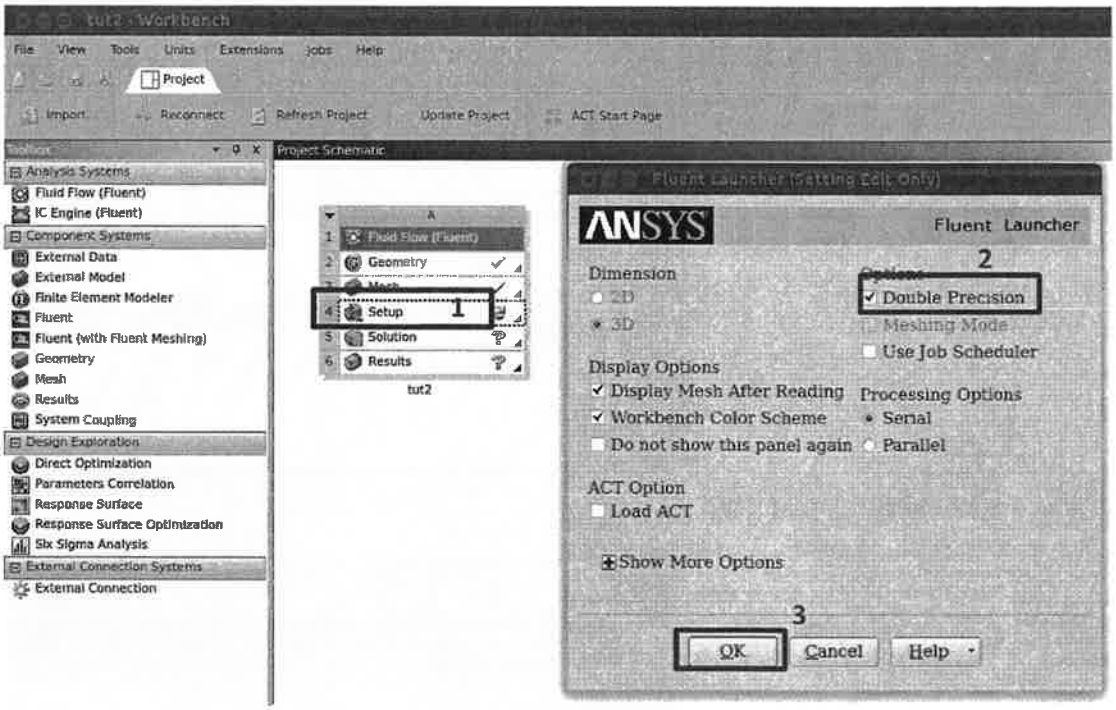




Save project and then exit from ANSYS Meshing.

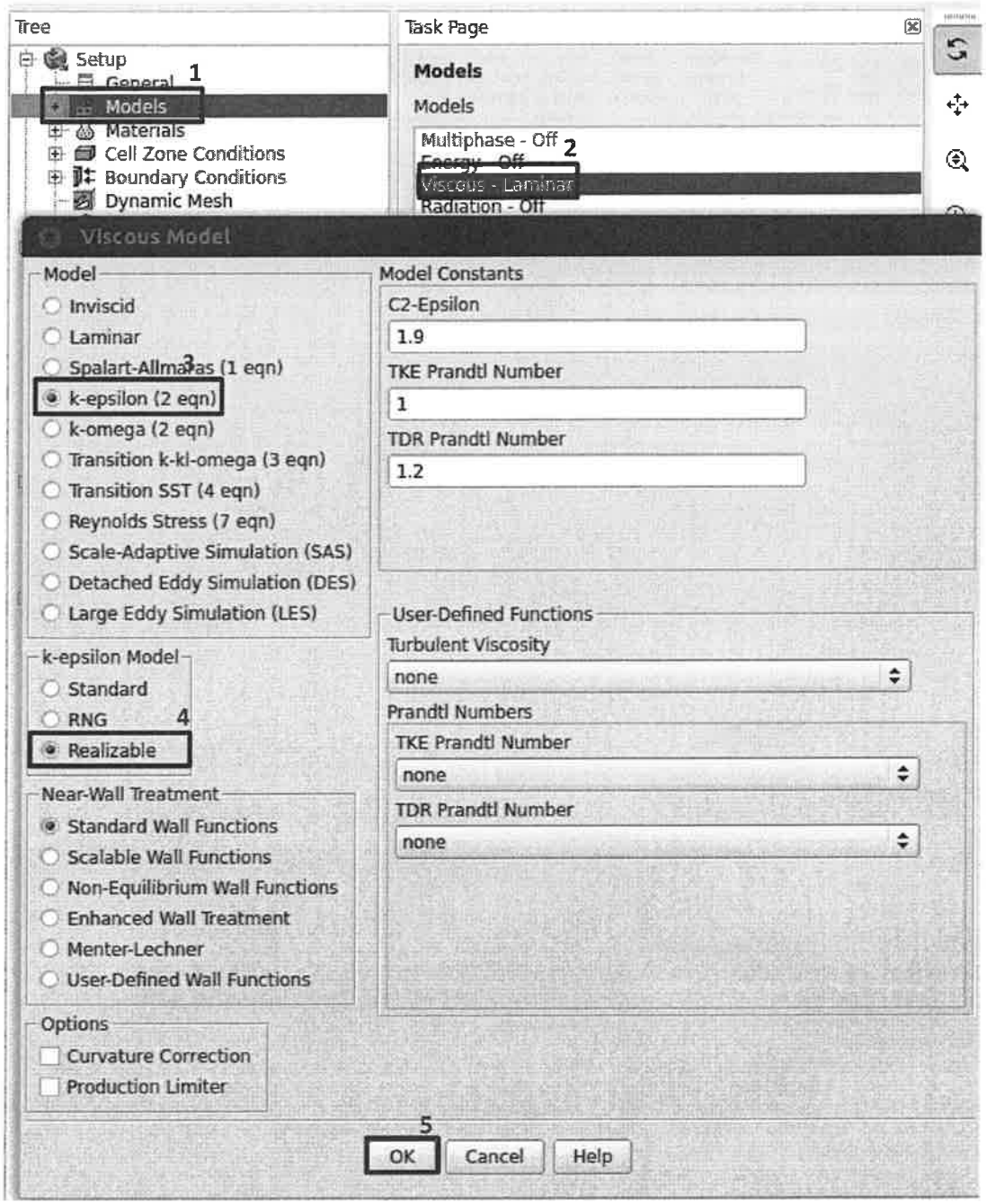
3 Numerical Settings and Computation in ANSYS Fluent

Double click **Setup** icon to launch ANSYS FLUENT. Turn on **Double Precision** option.



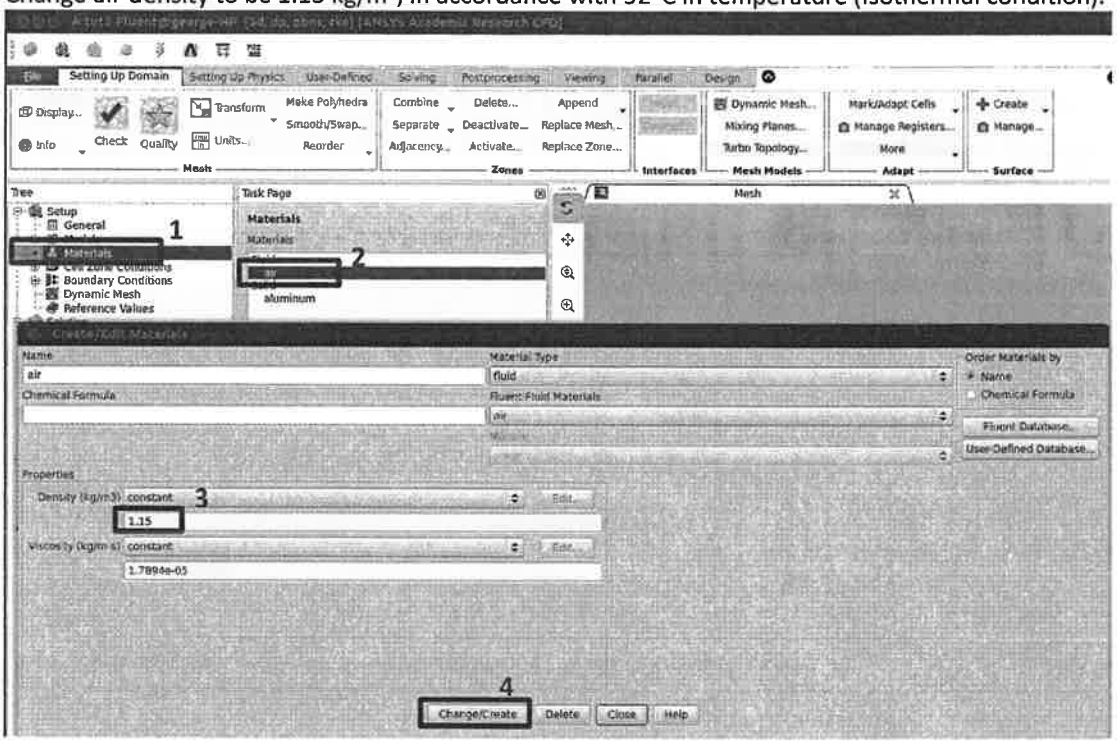
3.1 Models setting

Choose Realizable *k-e* turbulence model.



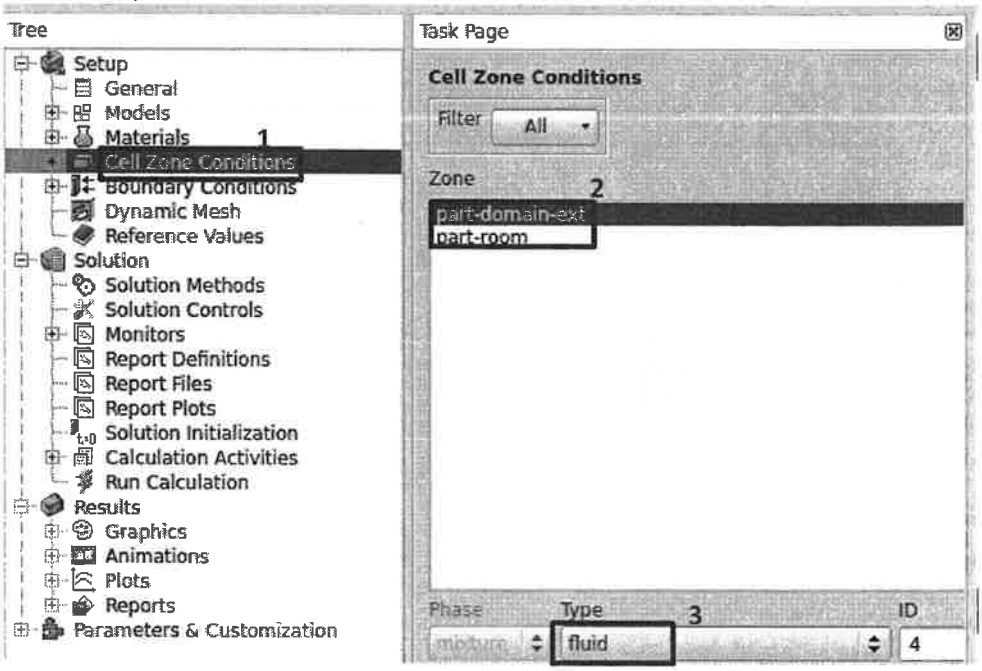
3.2 Materials setting

Change air density to be 1.15 kg/m^3, in accordance with 32°C in temperature (Isothermal condition).



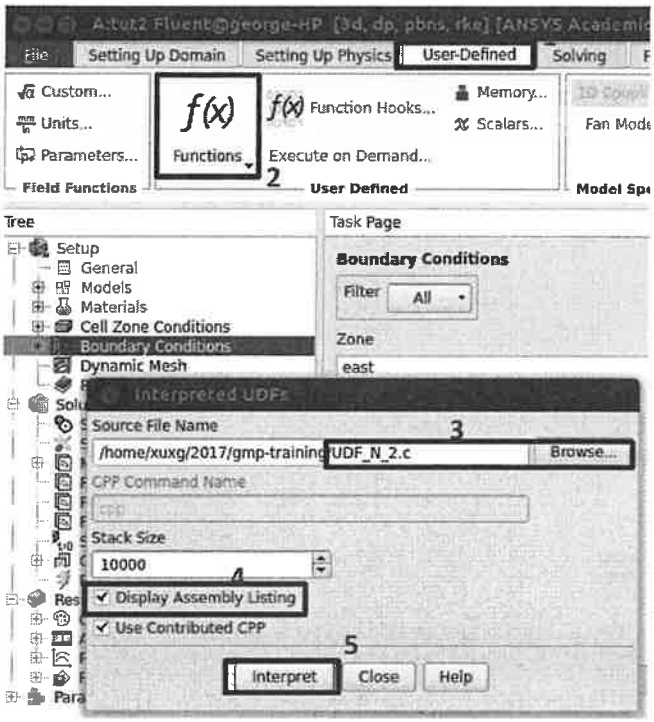
3.3 Cell zone setting

Make sure that the both cell zones (corresponding to the two bodies) are set as "fluid" type. Check them one by one.



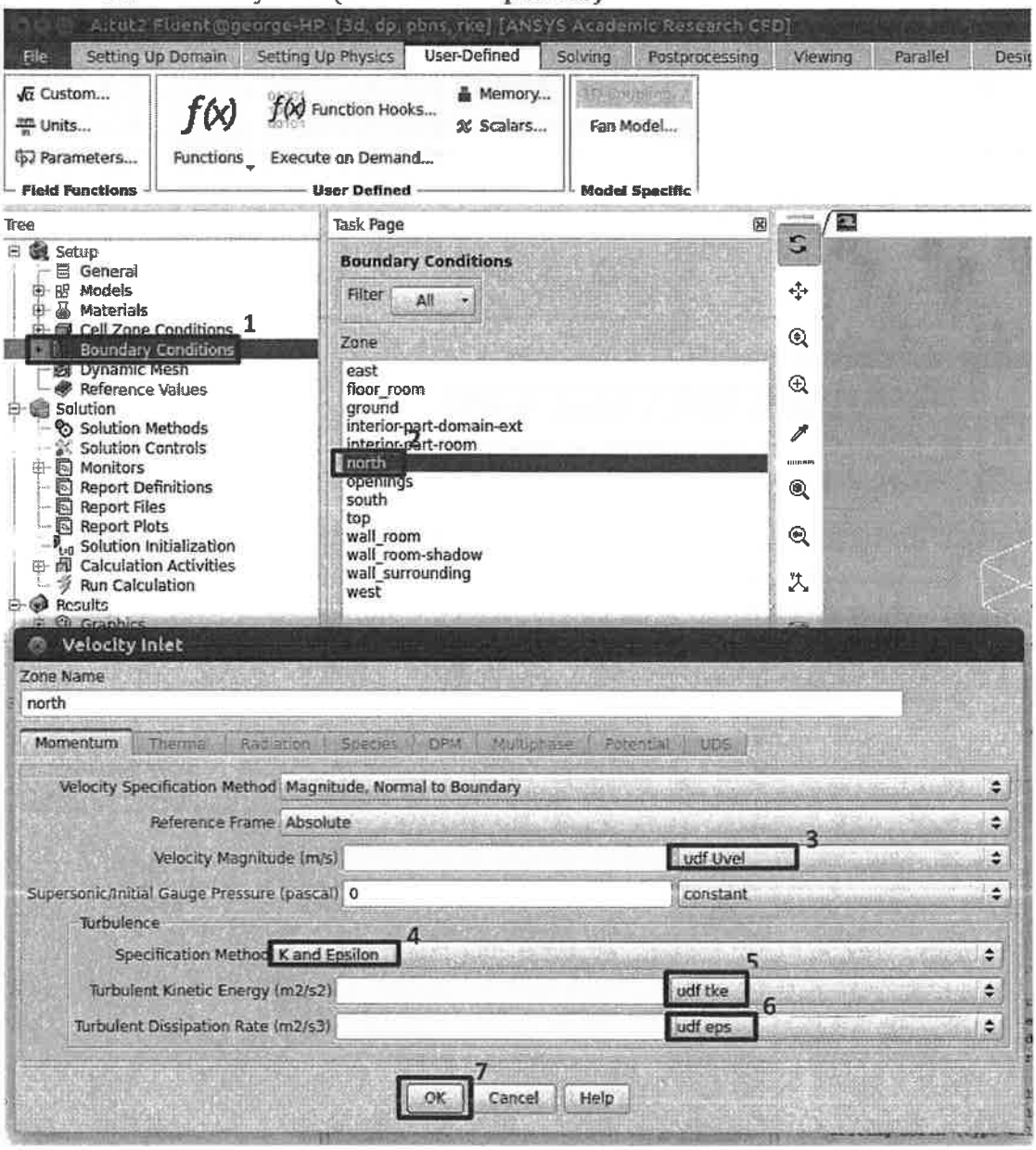
3.4 User define function (UDF)

From the main menu, choose **User-defined → *f(x)* Functions → Interpreted**
In the pop-up window for Interpreted UDFs, choose the udf file "UDF_N_2.c" to be compiled. The user defined functions will be applied at the velocity inlet to represent the atmospheric flow profiles.

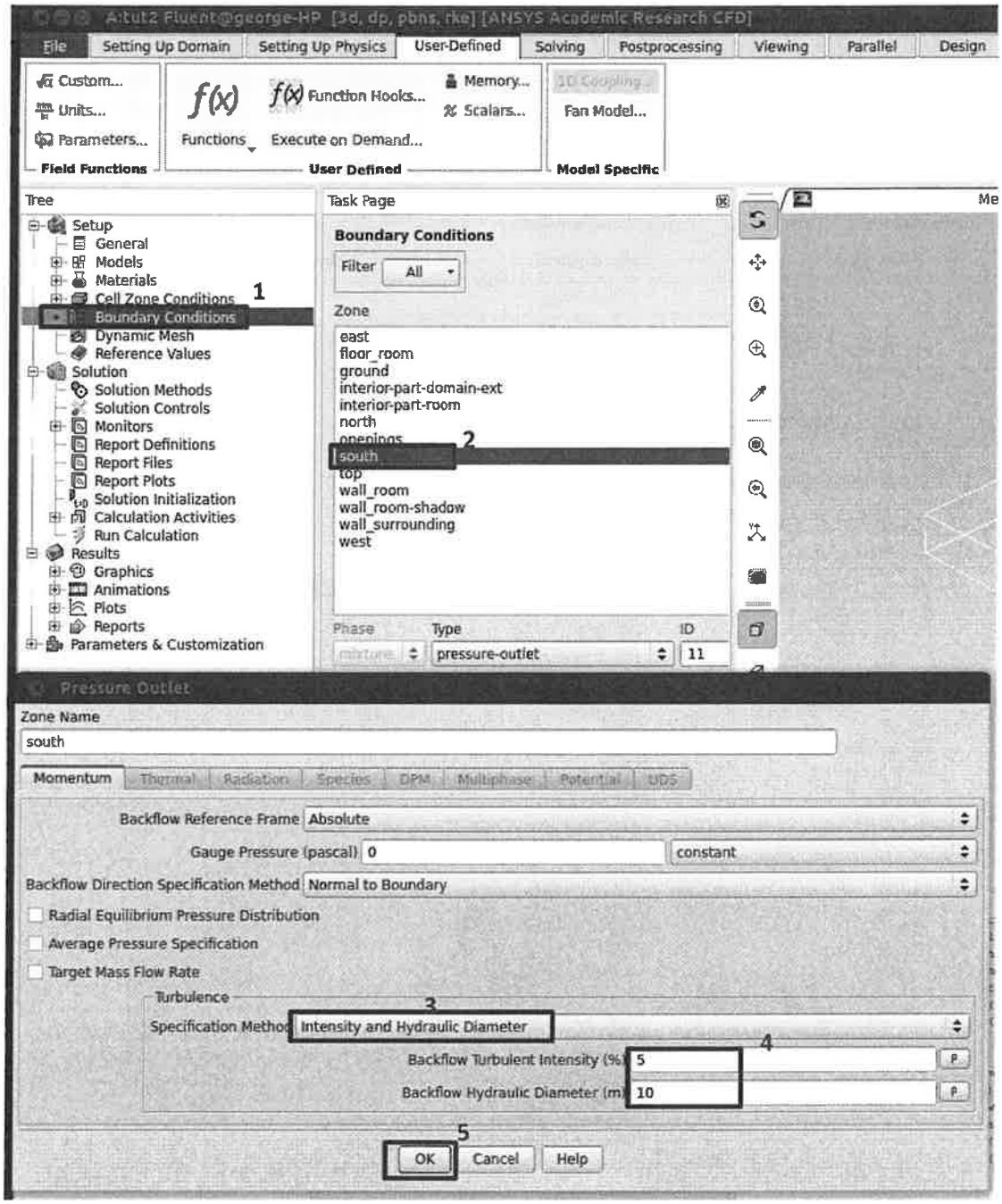


3.5 Boundary conditions

3.5.1 "north" – Velocity Inlet (based on UDF profiles)

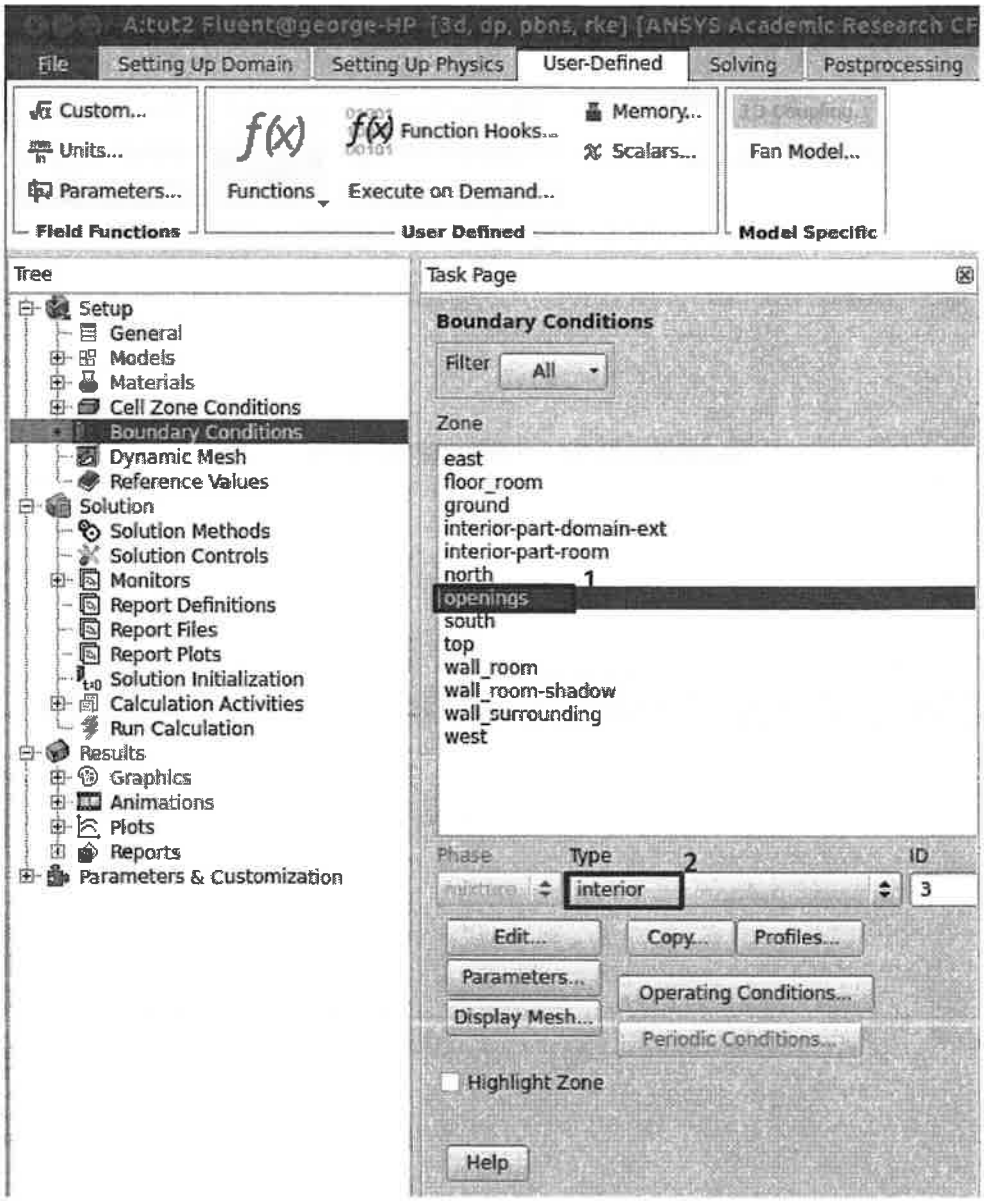


3.5.2 "south" – Pressure Outlet

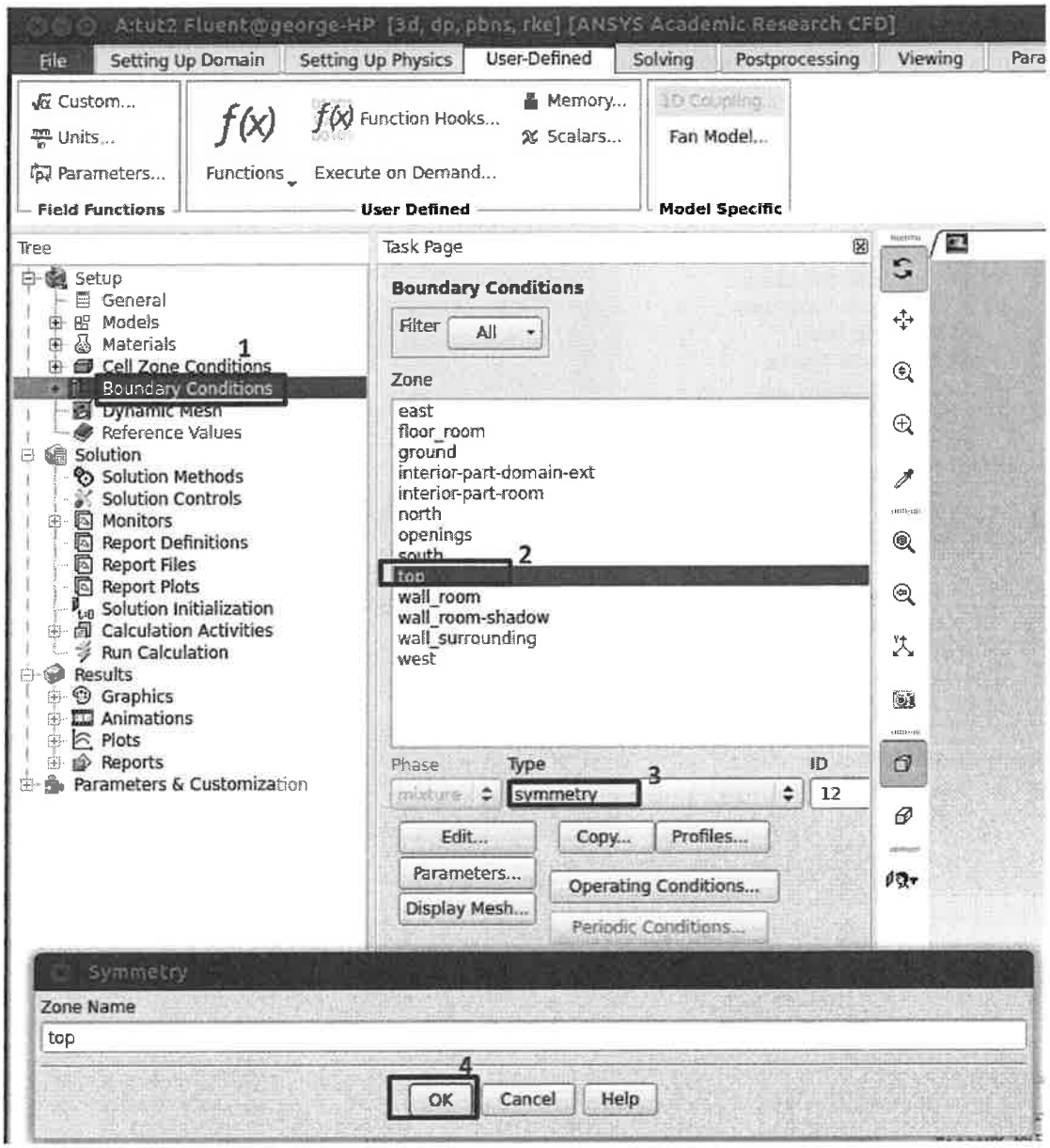


3.5.3 "openings" – interior

Make sure that interior type boundary conditions are applied for Named Selection "openings".



3.5.4 "top", "east", "west" – symmetry

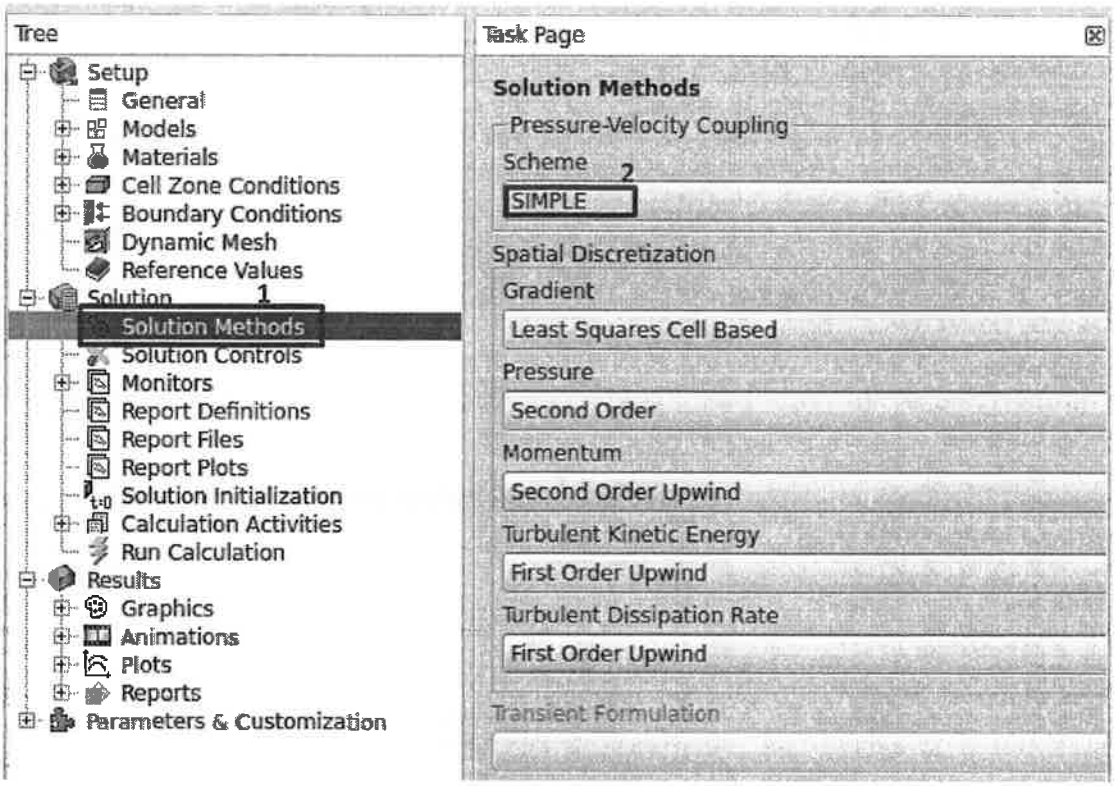


Follow the same procedure to change the boundary type for zonea "east" and "west" to be **Symmetry** too.

Zones including "ground", "wall_building", "wall_room" and "floor_room" are automatically defined as **wall**-type boundary conditions by default.

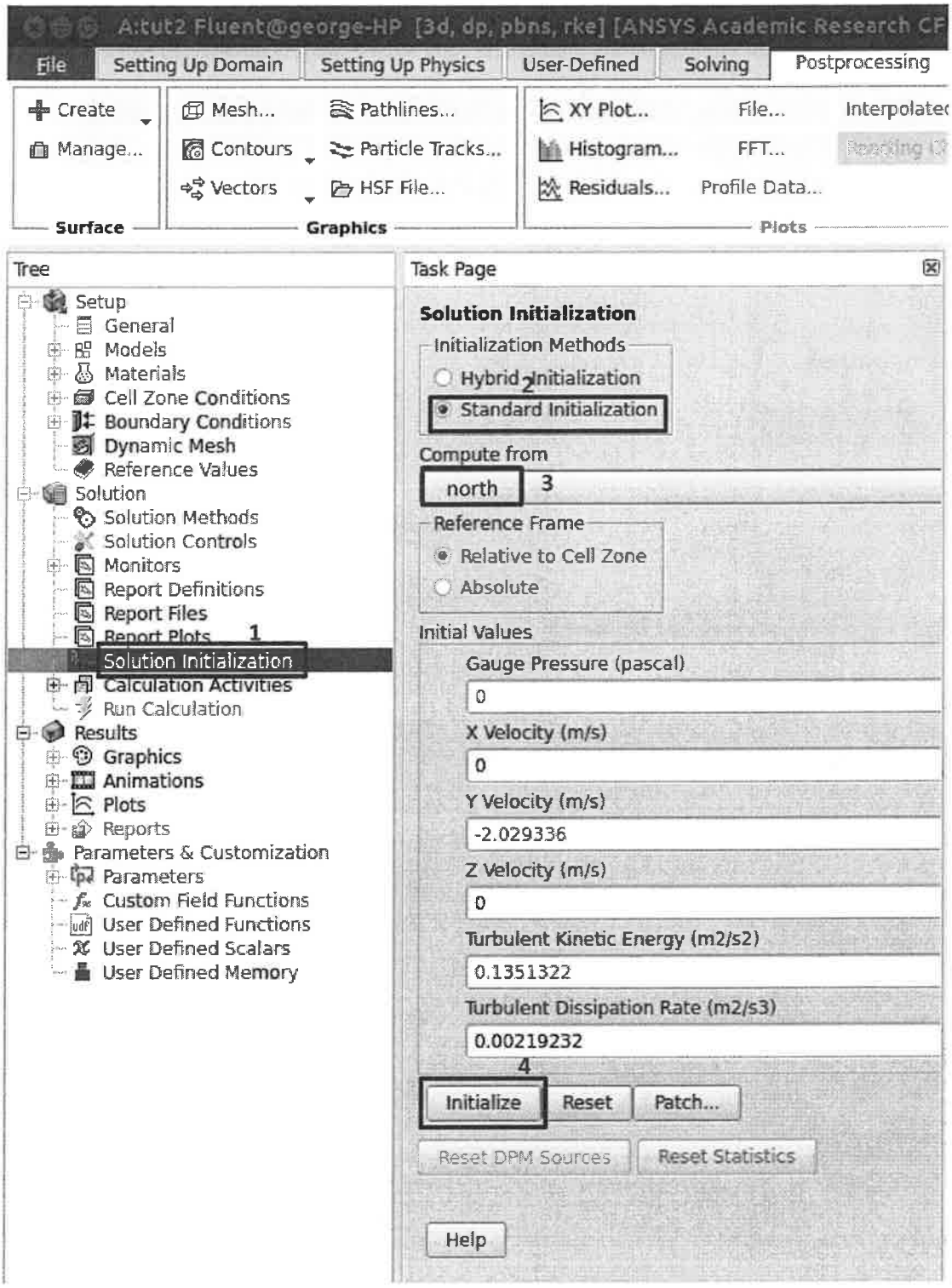
3.6 Solution methods

Choose **SIMPLE** pressure-velocity coupling scheme, and adopt the default spatial discretization schemes as the solution methods. Please take note that second-order discretization schemes are required in BCA GM CFD Methodology Guideline.



3.7 Solution initialization

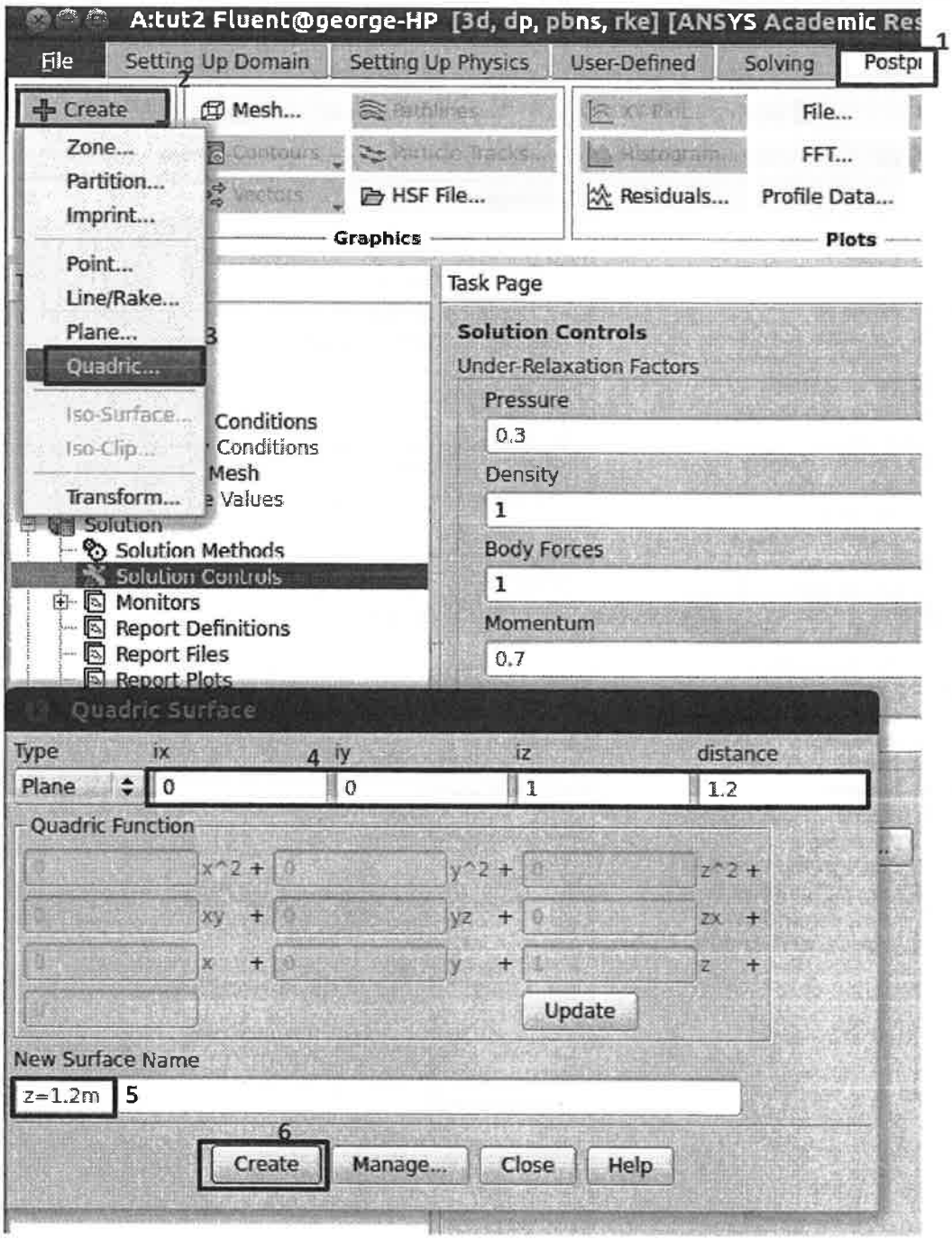
Use **standard initialization** with the initial values computed from the "north" boundary - velocity inlet.



3.8 Define cutting planes

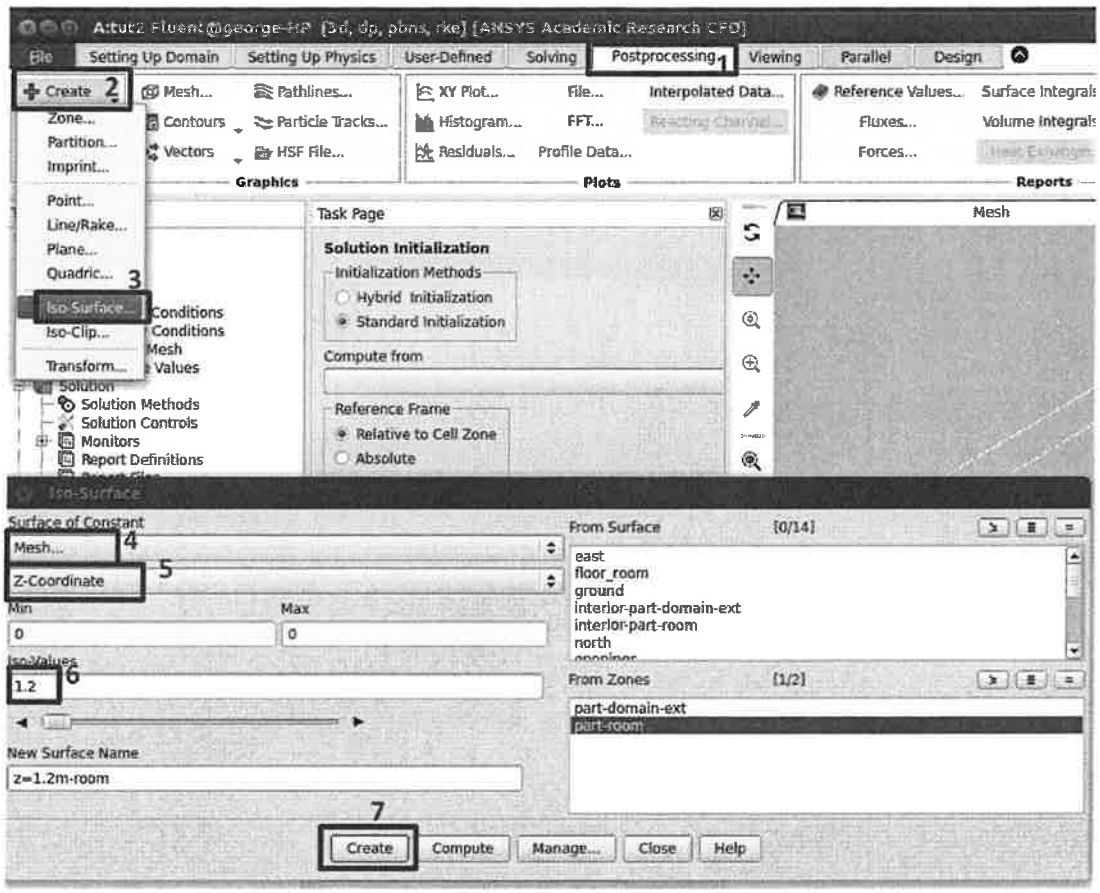
Choose **Postprocessing -> Create -> Quadric...**

Define the cutting plane at 1.2m above the ground face (namely z=1.2m), as well as the portion within the room (namely z=1.2m_room). The defined cutting planes can be used to trace the changes of monitored variables during the iteration, and to post-process simulation results later.



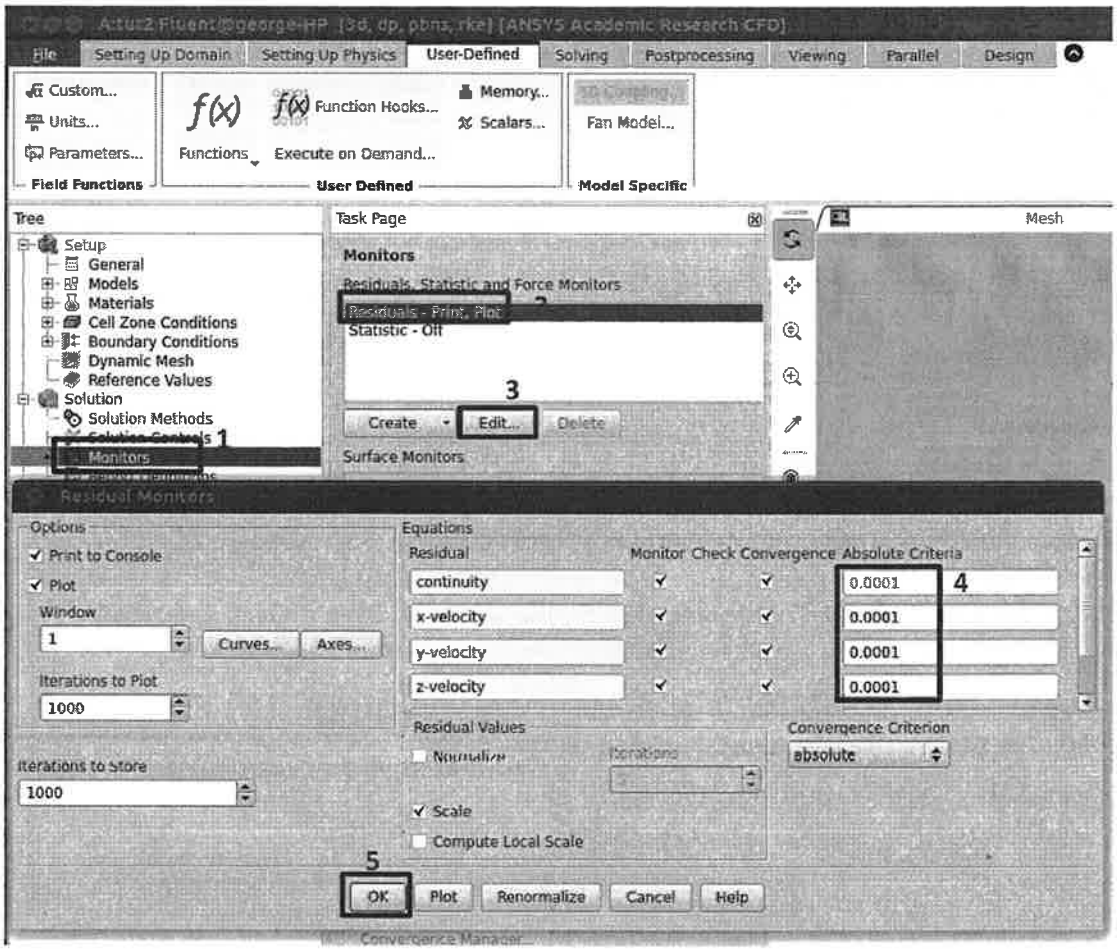
Choose **Postprocessing -> Create -> Iso-Surface...**

Define the cutting plane at 1.2m above the floor that is bounded within the room (namely z=1.2m_room).

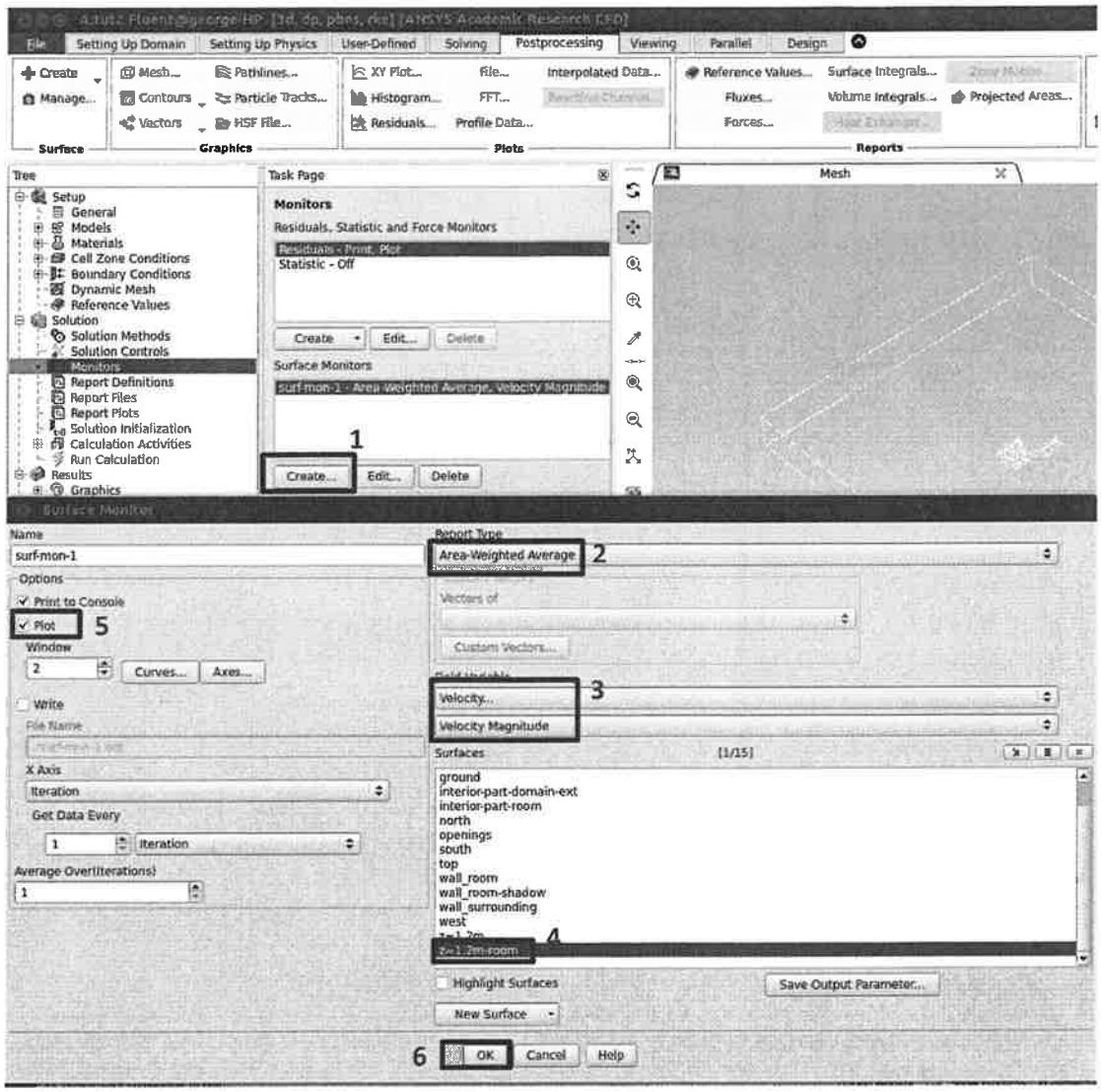


3.9 Monitors setting

Set target residuals for all quantities to be 0.0001.



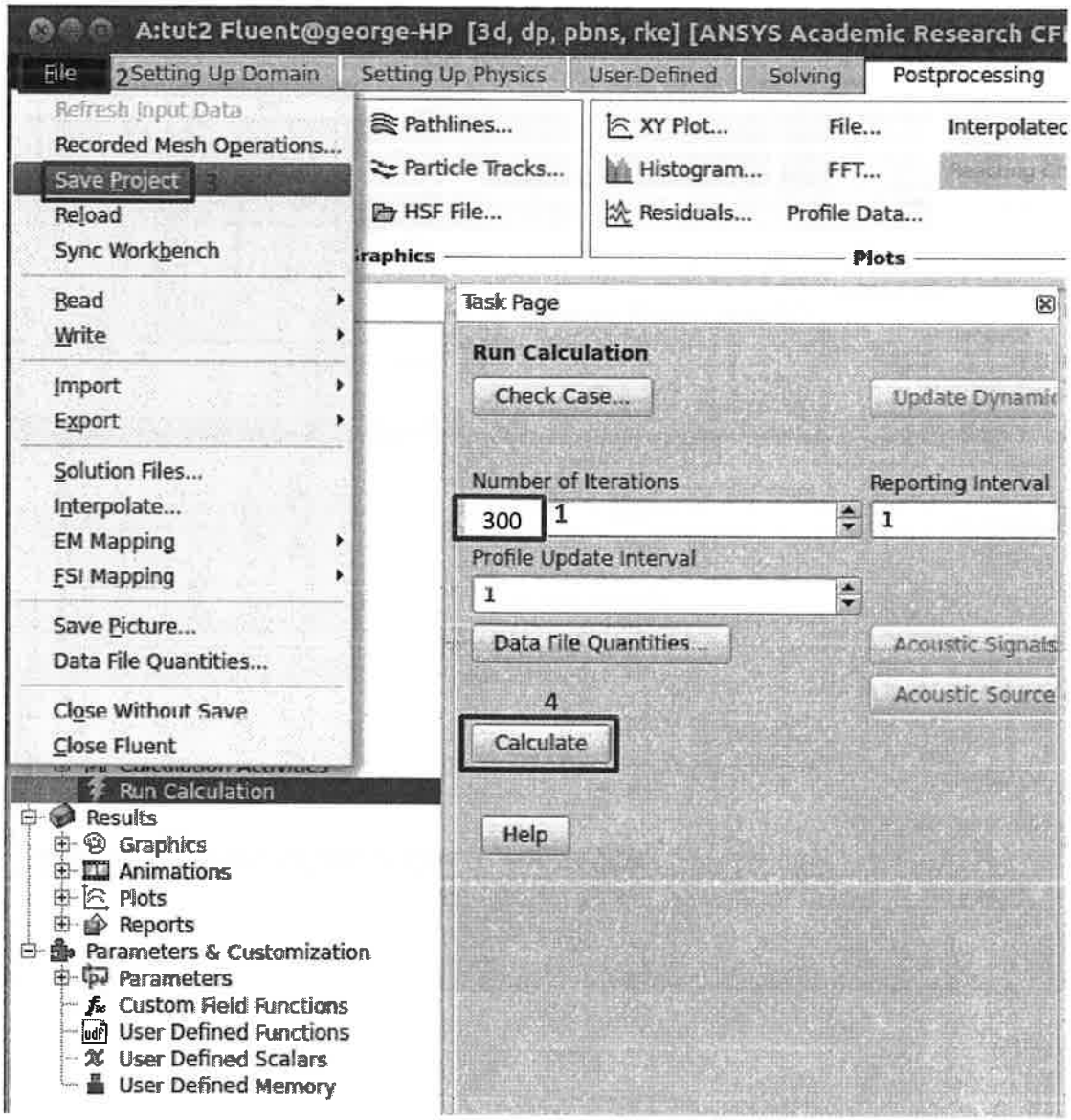
Create a surface monitor to track the area-averaged velocity changes on the cutting plane z=1.2m_room, as iteration advances.



3.10 Run Calculation

Set the maximum number of iterations to be 300 for the tutorial. Please note that 300 iterations may not be sufficient to reach solution convergence. Student can run more if necessary.

Save project and then click the **Calculate** button to start the simulation.

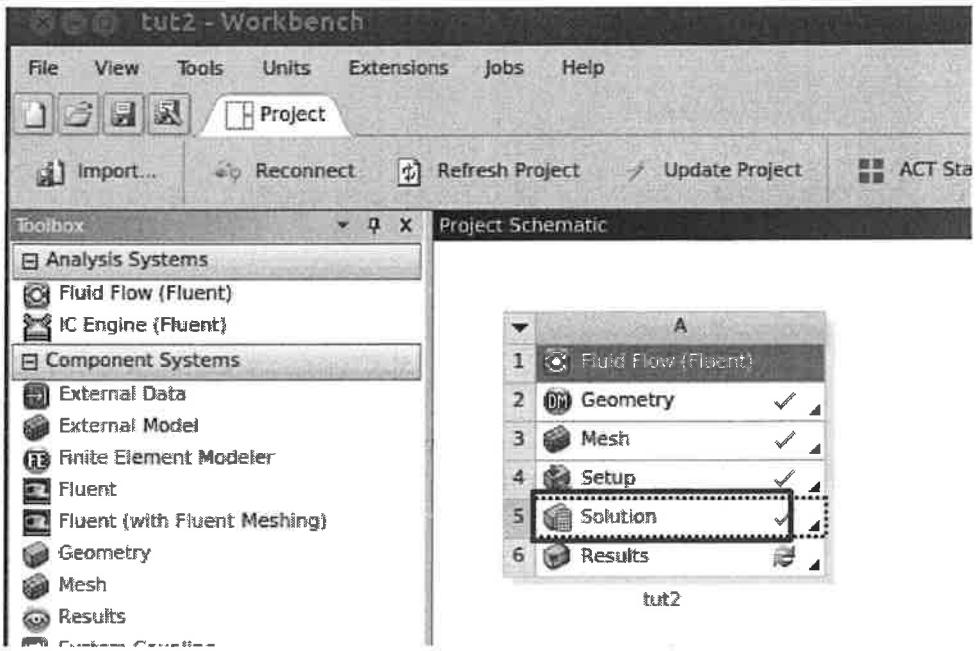


Switch between the two screens to check the changes of residuals and monitored velocity upon the iteration process.

Once the simulation is completed, **save project** again and then **Close Fluent.**

4 Post-Processing Simulation Results in ANSYS Fluent

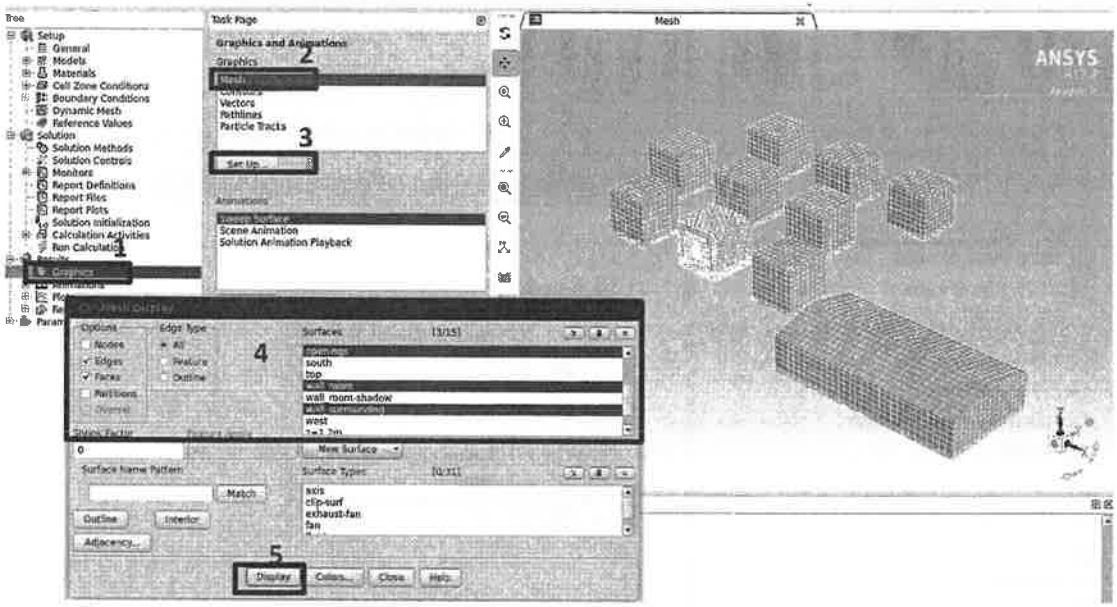
Within the Workbench, double click **Solution** to launch ANSYS FLUENT to visualize and report the simulation results.



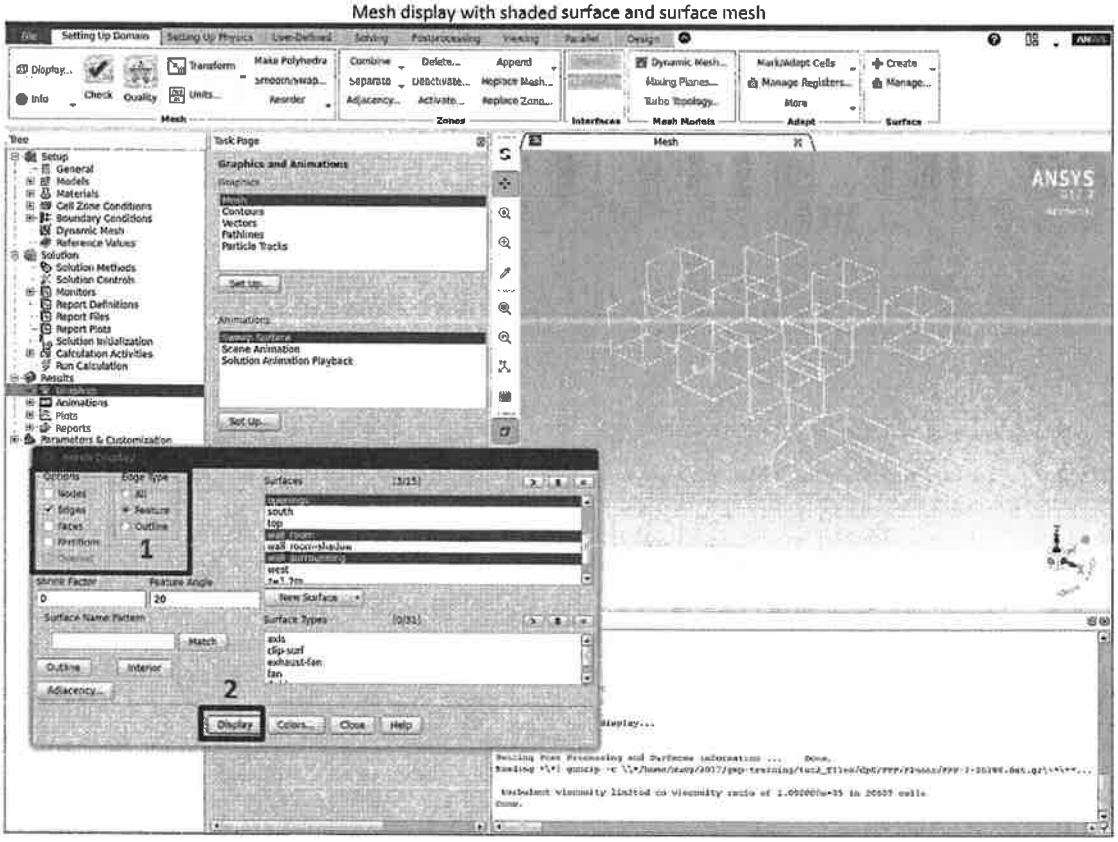
4.1 Mesh display

Results -> Graphics -> Mesh -> Set Up ...

You can choose to show the geometry with or without surface mesh displayed, thanks to the variable display options in ANSYS Fluent. The both presentation manners can be superimposed with the other visualization modes to be demonstrated later.



Mesh display with shaded surface and surface mesh

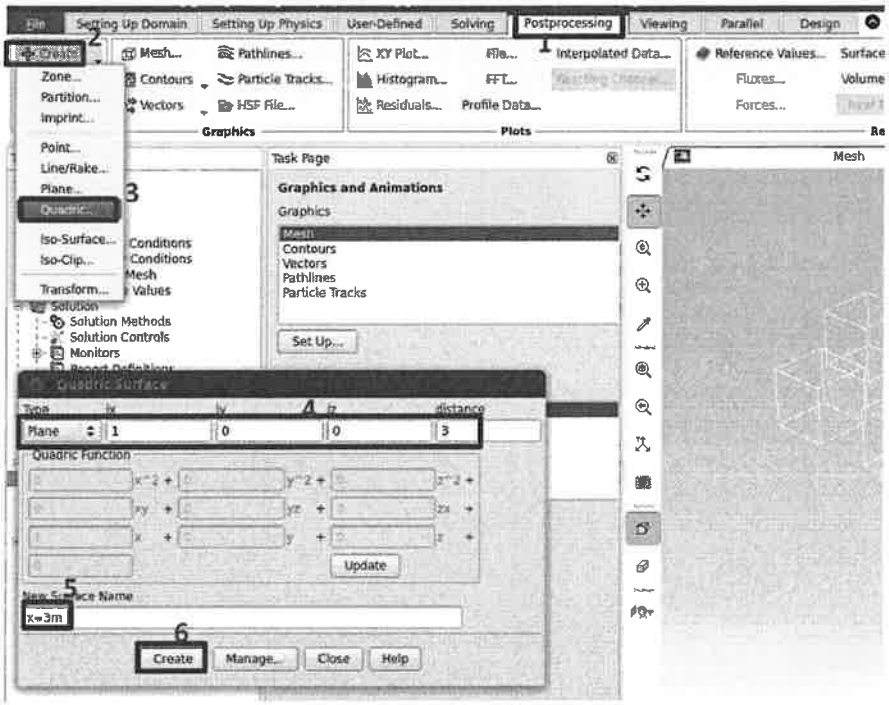


Mesh display with feature edges only

4.2 Creation of additional cutting plane

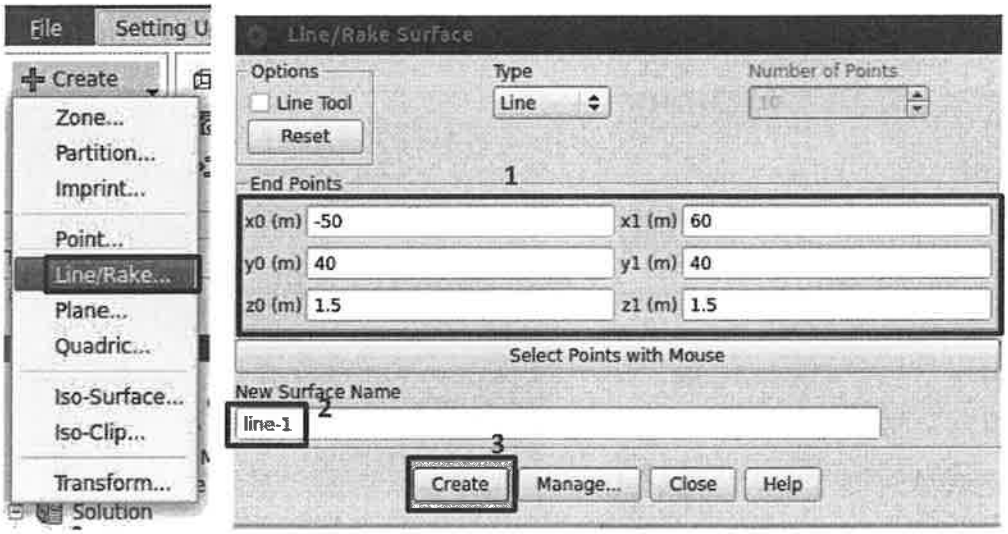
Create a vertical cutting plane (namely x=3.0m) sliced through the two openings of the Silsoe Cube of interest.

Postprocessing -> Create -> Quadric ...



Create a line (namely line-1) for the usage of pathlines plot later.

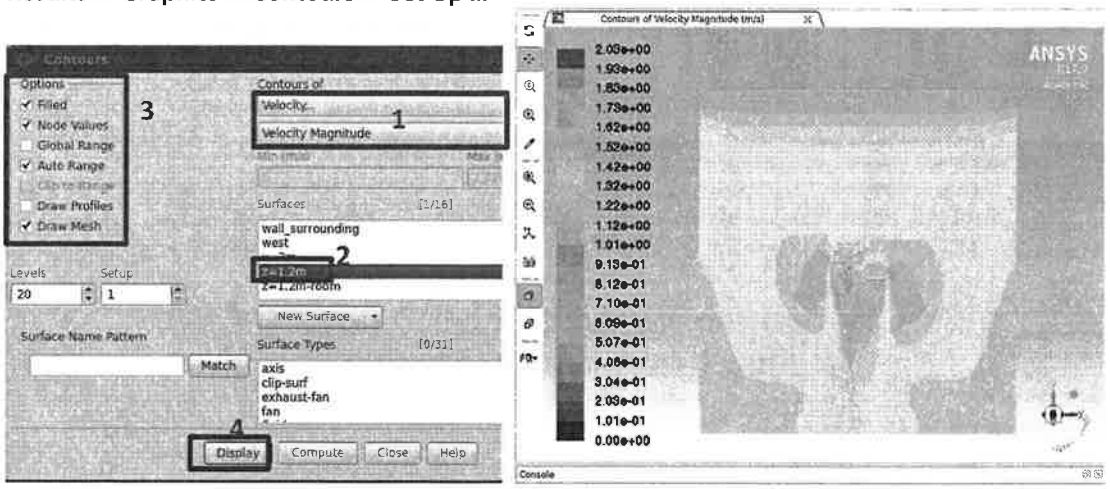
Postprocessing -> Create -> Line/Rake ...



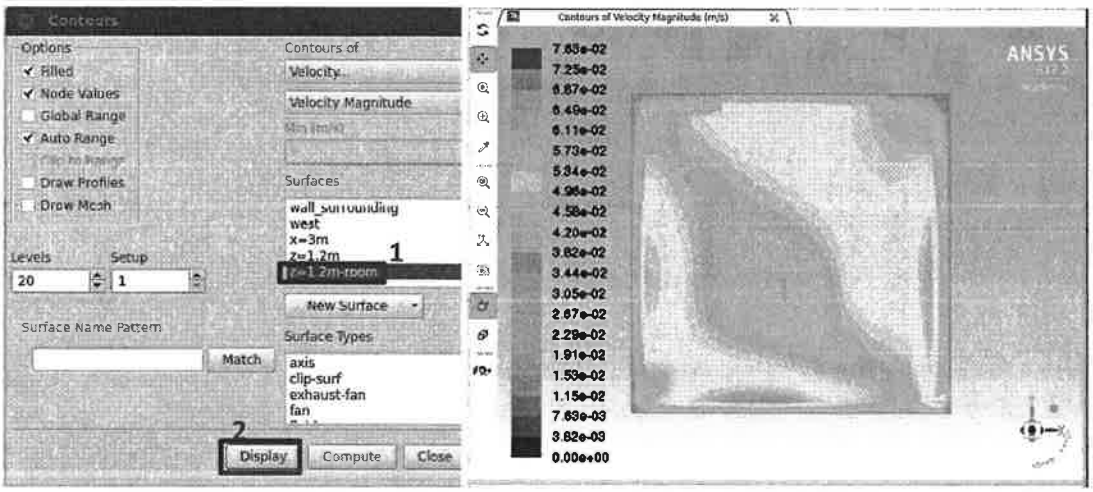
4.3 Contour plot

Spatial distribution of different variables can be readily presented in contour plots.

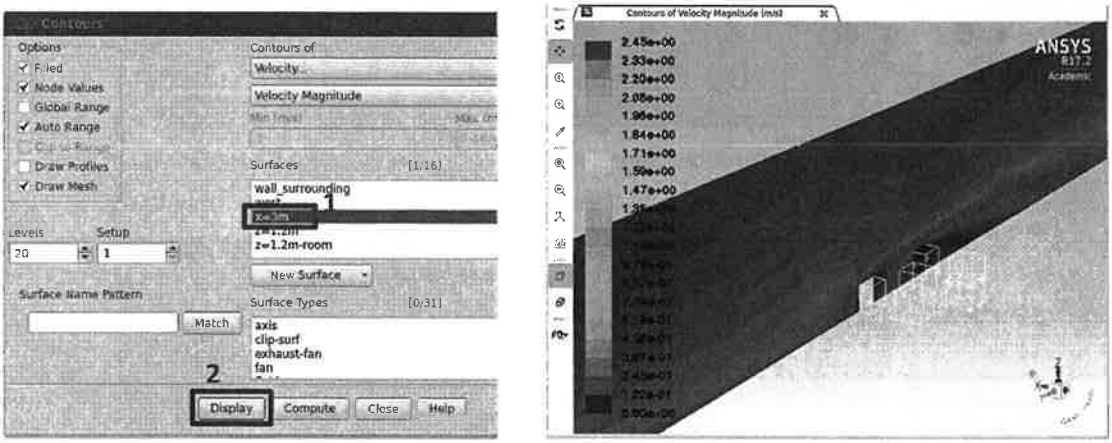
Results -> Graphics -> Contours -> Set Up ...



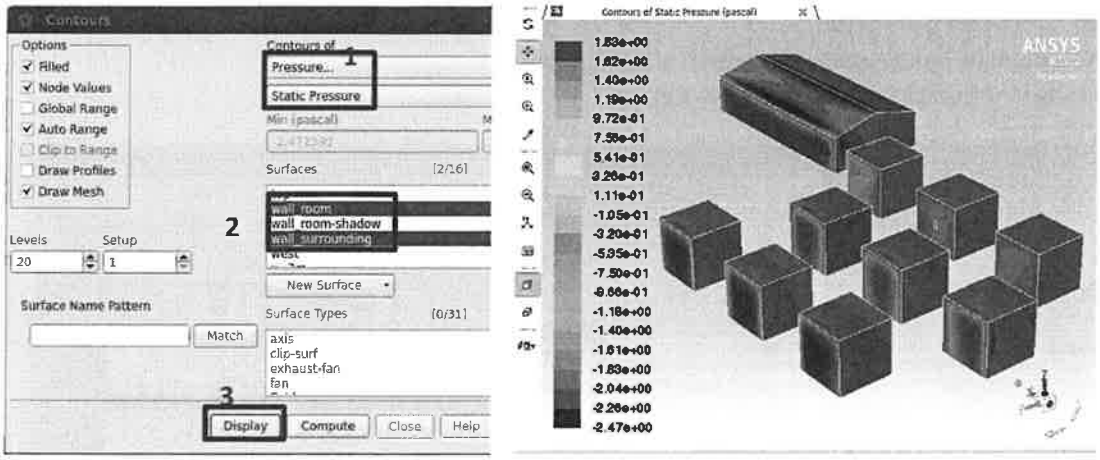
Contour plots of velocity magnitude on the horizontal cutting plane Z= 1.2m above the ground.



Contour plots of velocity magnitude on the horizontal cutting plane Z= 1.2m within the Silsoe Cube of concern.



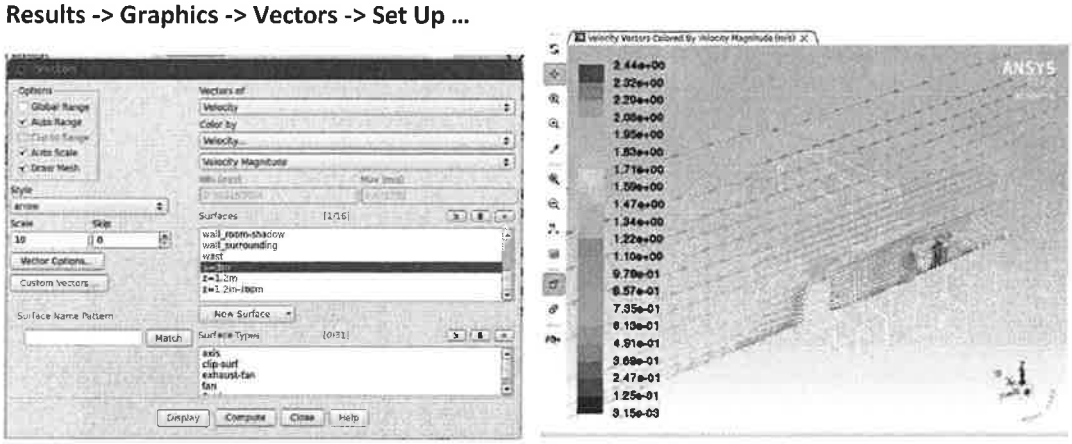
Contour plots of velocity magnitude on the vertical cutting plane X = 3m across the openings of the Silsoe Cube of concern.



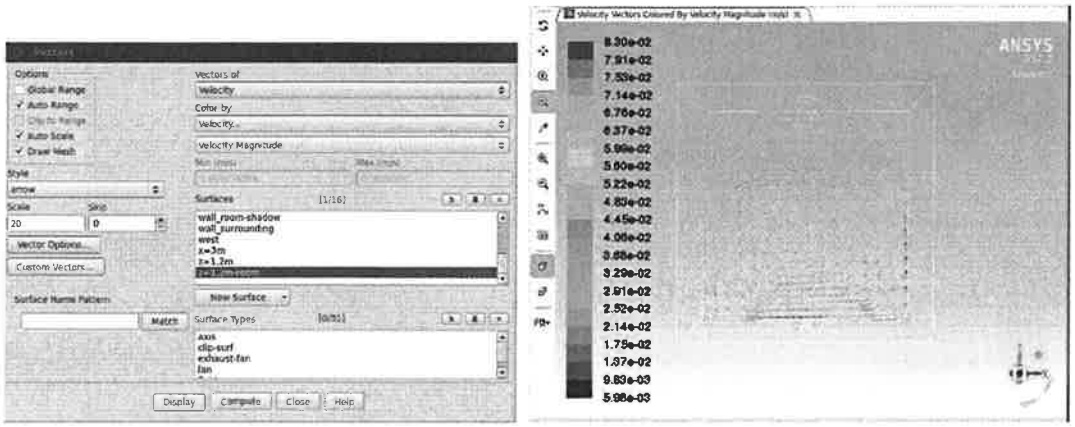
Contour plots of static pressure on the surfaces of building blocks.

4.4 Vector plot

Results -> Graphics -> Vectors -> Set Up ...



Vector plot of air movement around and across the Silsoe Cube on the vertical cutting plane of X=3m.

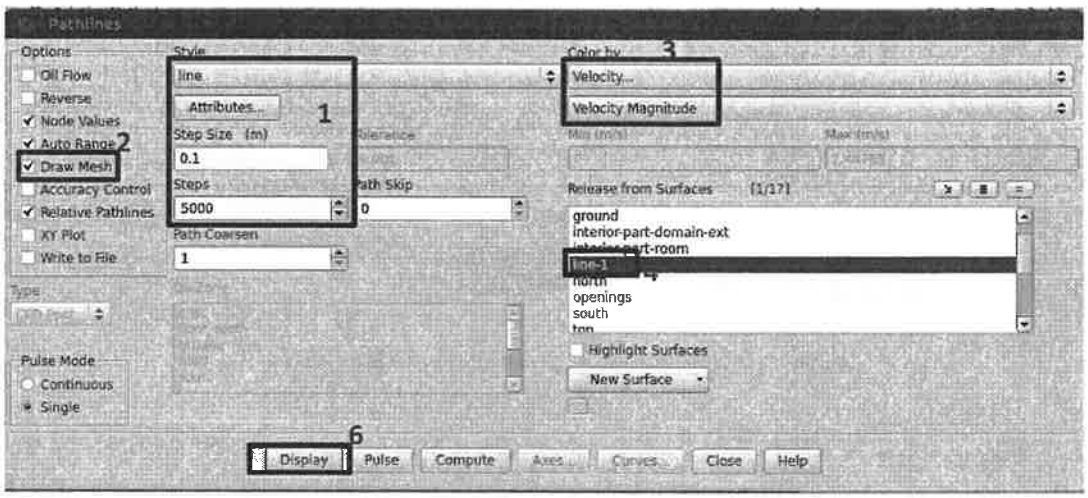


Vector plot of air movement within the Silsoe Cube on the horizontal cutting plane of Z=1.2m.

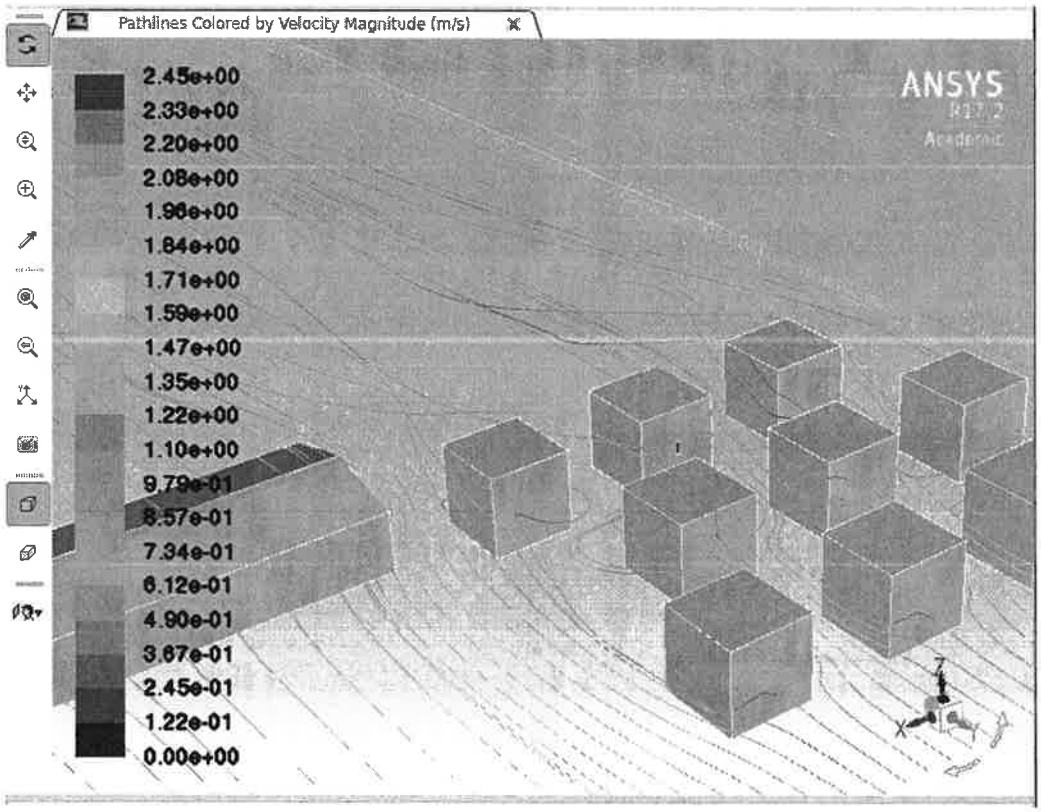
4.5 Pathlines plot

Air movement can be vividly presented by pathlines plot.

Results -> Graphics -> Pathlines -> Set Up ...



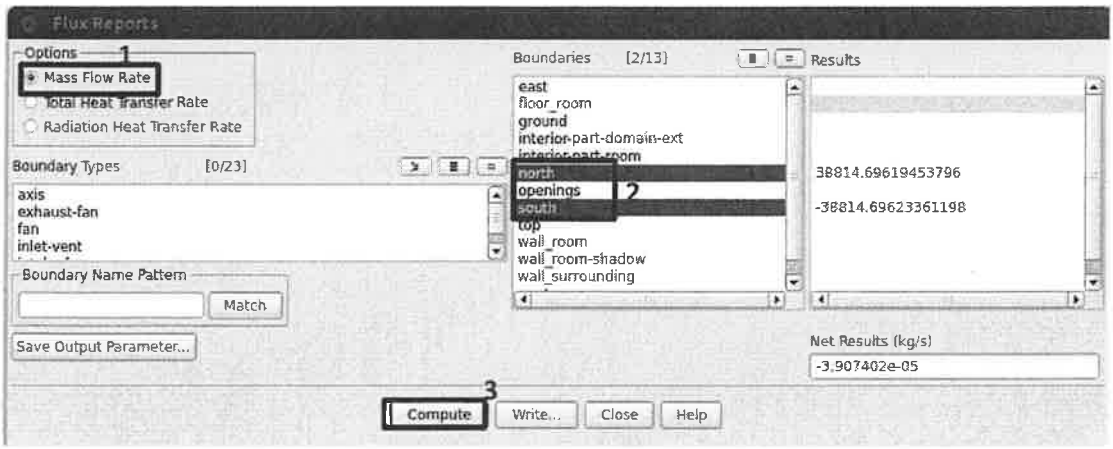
In the mesh display, please turn on the shaded face display option (referring to Section 4.1 Mesh display). Then the similar pathlines plot, as shown below, can be produced.



4.6 Mass balance and averaged velocity calculation

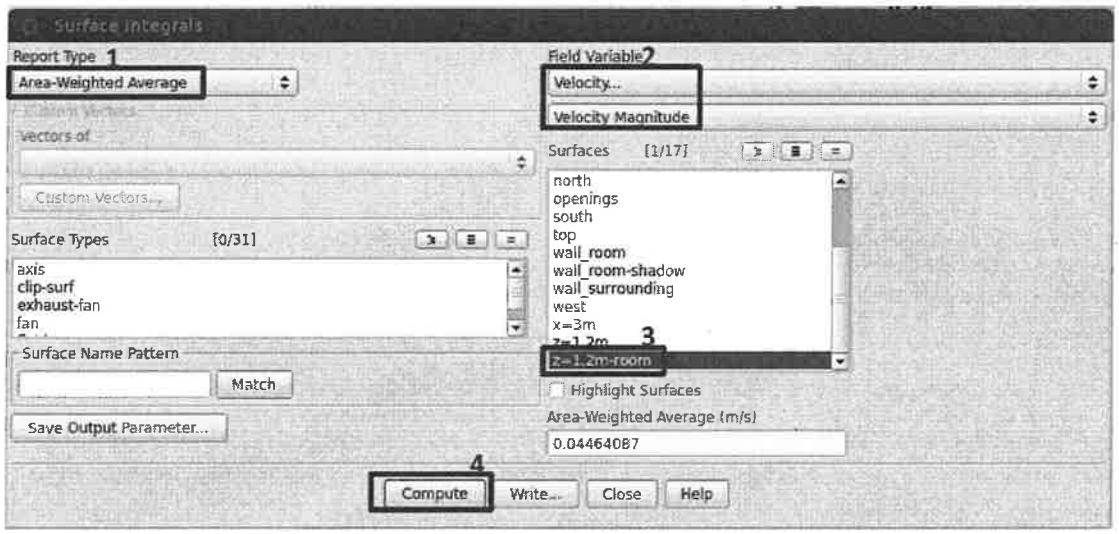
Select the boundaries "north" and "south", which quantify respectively the sole inflow and outflow conditions, should result in negligible difference in terms of mass flow rates for a converged and accurate simulation.

Results -> Reports -> Fluxes -> Set Up ...



As required by Green Mark code, the area-averaged velocity across the Silsoe Cube is expected to be close to 0.6m/s, in order to meet the good ventilation conditions as expected. You can easily calculate the area-averaged velocity using **Surface Integrals** tool in ANSYS Fluent.

Results -> Reports -> Surface Integrals -> Set Up ...



For this case, the averaged velocity is as low as 0.045m/s, which is far smaller than 0.6m/s referred in GM code.

It is your turn to figure out how to improve the natural ventilation conditions.

Hands-on Assignment 1 - Design and Simulation for Natural Ventilation

Project Title: Use CFD simulation tool (ANSYS-FLUENT) to perform ventilation simulation modelling and analysis in order to ensure good natural ventilation within the functional unit (100%)

Submission dateline:

Assignment Description:

Non-Residential Building type includes *hospitals, industrial facilities, hawker centres, sports facilities, commercial atriums and schools.* Do follow the Green Building certification criteria (e.g. Singapore BCA Green Mark Platinum GM Version 4.1 or equivalent standard) to assist the natural ventilation analysis for your project. The project work scopes should include firstly baseline design case (1 geometry); and followed by improved design cases (another geometry is required). Each simulation case is subjected to one (1) prevailing wind condition (either Northerly or Southerly wind).

Students can plan for computational domain with maximum site development area of 500m x 500m, subjected to compute resource availability. Plot ratio can be approximated necessarily. Existing building layout can be obtained from google map and on-site observation (for building height). Building geometry entities can be imported from SketchUp, AutoCAD 2D/3D, 3D Studio Max or BIM-REVIT. Alternatively, student can also plan for the new building with own drawing, or deriving the necessary info from your work experience or literature paper.

In this individual assignment, students are strongly encouraged to follow the CFD quality control checklist shown in Figure 1 below, in order to ensure the CFD Natural Ventilation results fulfill the modelling fidelity and accuracy standard.

Checklist Items	Description/Selection	Response & Criteria
Submission Details	Client, architect, ESD consultants	Provide the project details (especially info on NV design, building massing/orientation, GFA of NV spaces, % of opening & windows, credible source of site information with surrounding buildings, vegetation and terrain, future development etc)
Building Type	Hospitals, schools, hawker centres, sports facilities, commercial atriums or industrial facilities	Describe building functionality, targeted NV spaces and occupants, transit area

Checklist Items	Description/Selection	Response & Criteria
Problem Statement	Objective & Work Scope	Describe NV challenges, proposed solution, desired outcome and work scopes from the simulation model. Whenever necessary, use the architectural drawing for explanation. Describe design stages and fix simulation details. Subsequent design change has to be supported by simulation results.
Site Information	Within 500m distance stream wise and span wise from the edge of development of interest	Describe the site information (including surrounding buildings, terrain, greenery), and illustrate how the geometrical info is incorporated into the simulation model (conversion process). Surrounding buildings within 500m distance stream wise and span wise from the edge of development of interest should be modelled explicitly; while the greenery can be modelled implicitly with tree canopy approach. Terrain effect can be ignored if elevation is less than 10m.
CFD Approach	Simulation Methodology	The CFD solver shall have the minimum capability of solving the Navier-Stokes fluid flow equations for a three-dimensional incompressible flow at steady state. Turbulence modeling shall also be included with the minimum requirement of using the standard k-ε turbulence model, coupled with standard wall function
	Assumption & Simplification	Describe simulation model assumption, limitations and geometrical simplification. Whenever necessary, use the published literature data (including software manual) and comparison between architectural & CFD model for explanation
CFD Domain	Computational Domain	Describe the domain decomposition methodology; and relevant meshing type for each domain within the site. Describe the domain that be modelled implicitly with Davenport Roughness classification
CFD Meshing	Mesh independence study	To perform mesh independence study at the functional unit NV space
	Mesh size, distribution and quality	To carry out proper domain decomposition. To use hexahedral cells in the rectangular domain of NV space. Tetrahedral cells can be used to model the surrounding site features of the NV space. Hybrid pyramid or cut-cell mesh can be adopted at the interface. For implicit modeling of terrain roughness effect,

Checklist Items	Description/Selection	Response & Criteria
		prismatic or hexahedral cells will be used. As a guide, the dimension of the computational mesh should be set at 0.1 to 0.5 m within the functional space of interest, 0.5 to 1.0 m for building of interest and 1.0 - 5.0 m for surrounding buildings. The computational element size in vertical direction should be set at 0.5 - 1 m from ground surface to 10m height; followed by 1 - 5m to H_{max} height. Reporting on skewness and aspect ratio of the mesh is required.
CFD Model	Atmospheric Boundary Layer	1. To ensure horizontal ABL homogeneity in upstream and downstream 2. To ensure sufficiently high mesh resolution in vertical direction near ground (e.g. height of first cell < 1m) 3. To know the relationship between equivalent sand-grain roughness height (k_s) and corresponding aerodynamic roughness height, z_o 4. To ensure first cell center point (y_p) to be larger than physical roughness height (k_s)
	Buoyancy	To use Boussinesq or variable density and check gravity direction, if thermal simulation is performed.
	Turbulence model	To use the steady Reynolds-Average Navier Stokes (RANS), with minimum requirement k-ε turbulence model equation for NV flow.
	Wall treatment	To ensure wall functions producing y+ values that is satisfying the log-law fully turbulent zone: 40 < y+
CFD Boundary Conditions	Inlet wind profile	To ensure the vertical profile for wind velocity and turbulence in the ABL should be modelled by assuming constant shear stress with height.
	Top & lateral sides of domain	To use zero velocity gradients and zero normal gradients of all variables.
	Outlet plane	To use zero static pressure.

Checklist Items	Description/Selection	Response & Criteria
	Window modeling	To use actual window opening size. Attach window schedule and drawing for verification.
	Mechanical fan modeling	To use fan input with appropriate swirl radial and swirl flow components.
	Louver modeling	To use simplified porous zone with appropriate pressure drop components and directional effect.
	Heat source modeling	To use aggregated heat load from heat dissipating devices, such as cooking stalls for hawker center facilities, heat generators for industrial facilities and etc.
CFD Numerical	Discretization scheme	To use 2nd order for momentum equations
	Convergence criteria	To ensure solution is converged and monitored points at functional space reach steady values
User		Training, experience, consult expert
Documentation		Full documentation of parameters, readable scale
Design Iteration	Baseline case Modified case Optimal case	Baseline case – fixed building massing and layout Modified case – to highlight improvement on NV design Optimal case – final design with incorporation of NV features or passive innovative ideas

Figure 1: CFD Quality Control Checklist

In addition, students can also incorporate the following comments and improvement in the individual assignment

- For residential unit type
    - Justify whether the 0.6m/s criteria is fulfilled in the simulation study
    - Describe the unit selection for micro level simulation base on the estate level pressure differential; and provide your comment whenever possible
- For hawker center ventilation study, do justify the following scenario description
    - Mechanically-assisted ventilation

    - Heat source consideration in the Hawker center
    - Effective temperature (Dry Bulb + Mean Radiant temperature)
    - Provision for heat dissipation from the roof for the hawker center design
- For industrial or sport facilities ventilation study, do provide the following scenario description
    - Various functional area of the space
    - Justify the use of high volume low speed fan, if any
- For hospital facilities ventilation study
    - lowest level of the NV wards should be modelled
    - component/device modeling of the louver is necessary
    - describe whether the NV design of thermal comfort provision is targeting the working nurses or warding patients
- For school ventilation study
    - Vegetation/greenery effect on the wind aerodynamic drag – how to account for it?
    - Louver opening feature plays crucial part of the NV design
- For open atrium ventilation study
    - Deep podium ventilation cross flow vs. wind driven rain problem
- To include executive summary in the assignment report
- Shows the residual plot for simulation cases
- Whenever possible, use Two layer all y+ wall treatment & Realizable k-epsilon turbulence model

Submission details: Group assignment (team of 3, maximum). Put all the files into the group folder name and copy it to portable hard disk (to be handed out by course instructor) by 5pm, 1st March 2016. Your files should include the following details

1. Power point presentation file to describe the project nature, objective, work scopes, deliverable, results of baseline and improved design cases, and conclusions. Executive summary on the front page is required
2. ANSYS-FLUENT simulation files with proper naming (e.g. project d and case study details, e.g. Marina_Bay_Nat_Vent_Baseline.cas)

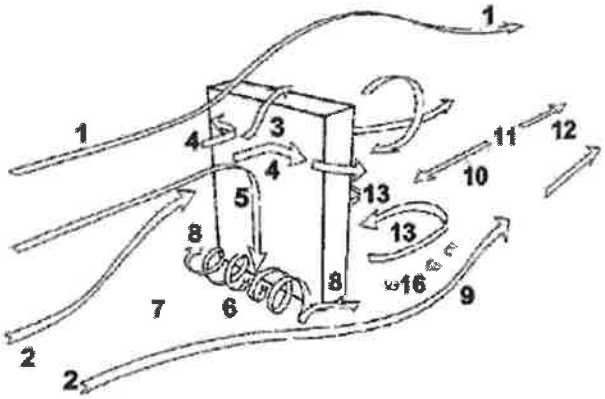
Hands-on Assignment 1 - Design and Simulation for Natural Ventilation

Benchmarking Test Cases for Building Aerodynamics & Urban Street Canyon

Apr 2017

Building Aerodynamics

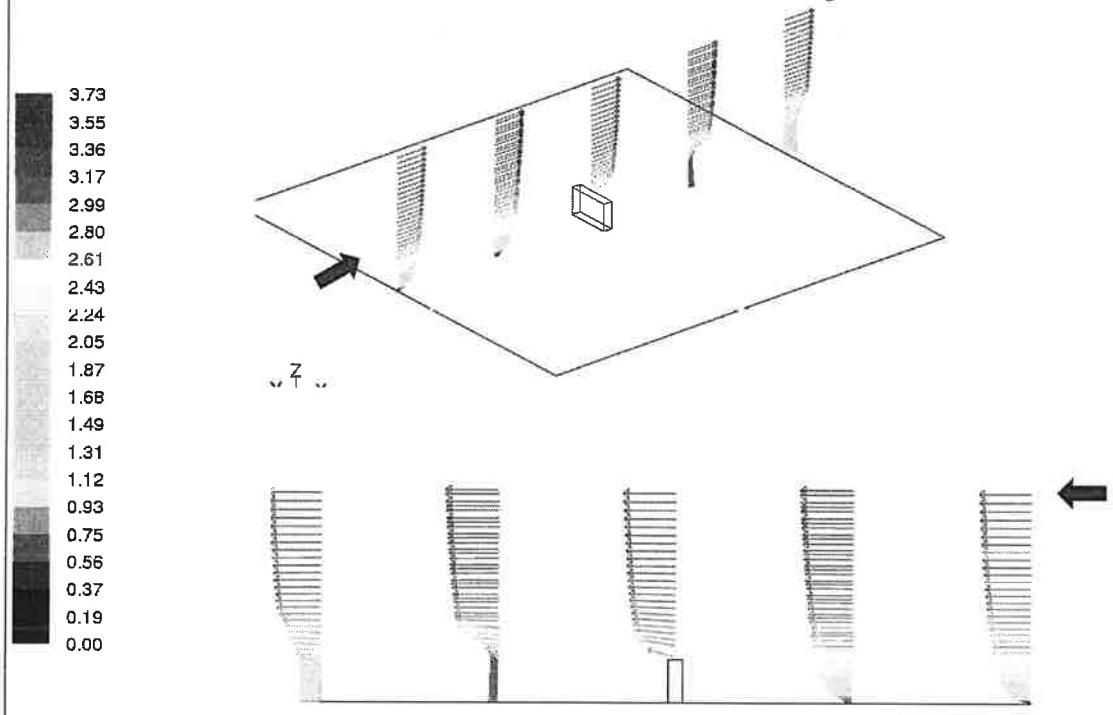
2.1. Wind flow around a single building



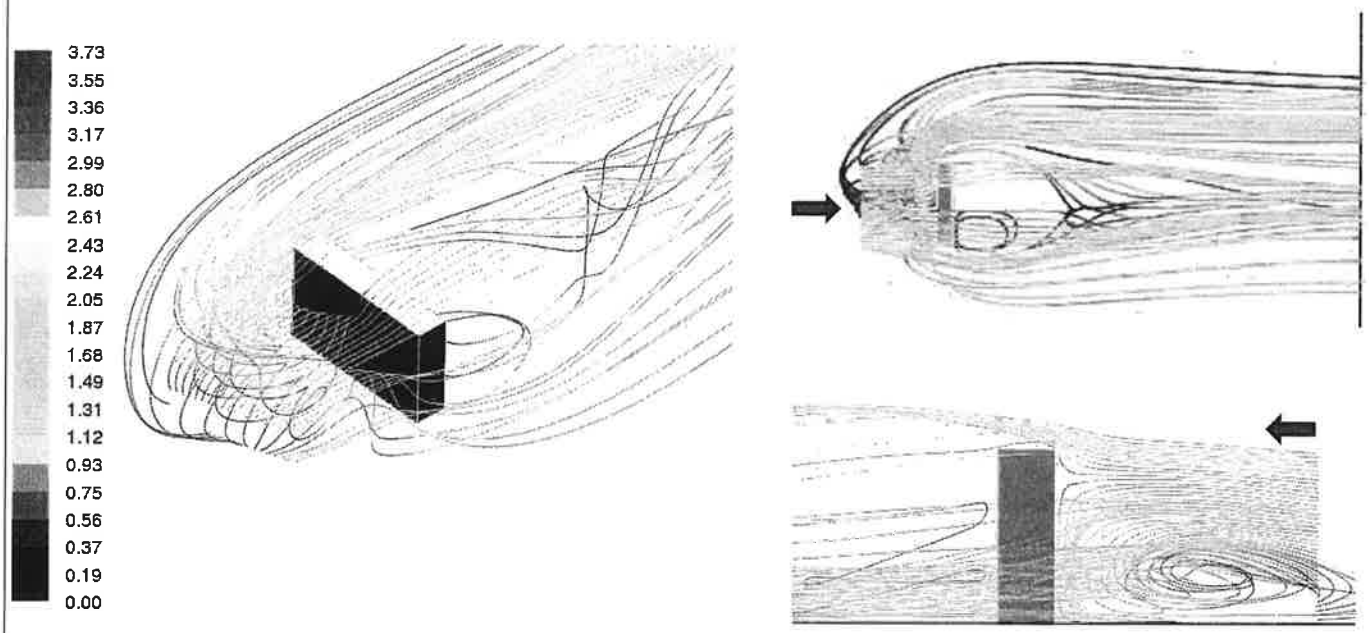
1. Flow over building
2. Oncoming flow
3. Flow from stagnation point over building
4. Flow from stagnation point around vertical building edges
5. Downflow from stagnation point
6. Standing vortex, base vortex or horseshoe vortex
7. Stagnation flow in front of building near ground level
8. Corner streams (vortex wrapping around corners)
9. Flow around building sides at ground level (adding to corner streams)
10. Recirculation flow
11. Stagnation region behind building at ground level.
12. Restored flow direction
13. Large vortices behind building
16. Small vortices in shear layer

I. P. CASTRO AND A. G. ROBINS, The flow around a surface-mounted cube in uniform and turbulent streams, *J . Pluid Mech.* (1977), *vol.* 79, *part 2, pp. 307-335*

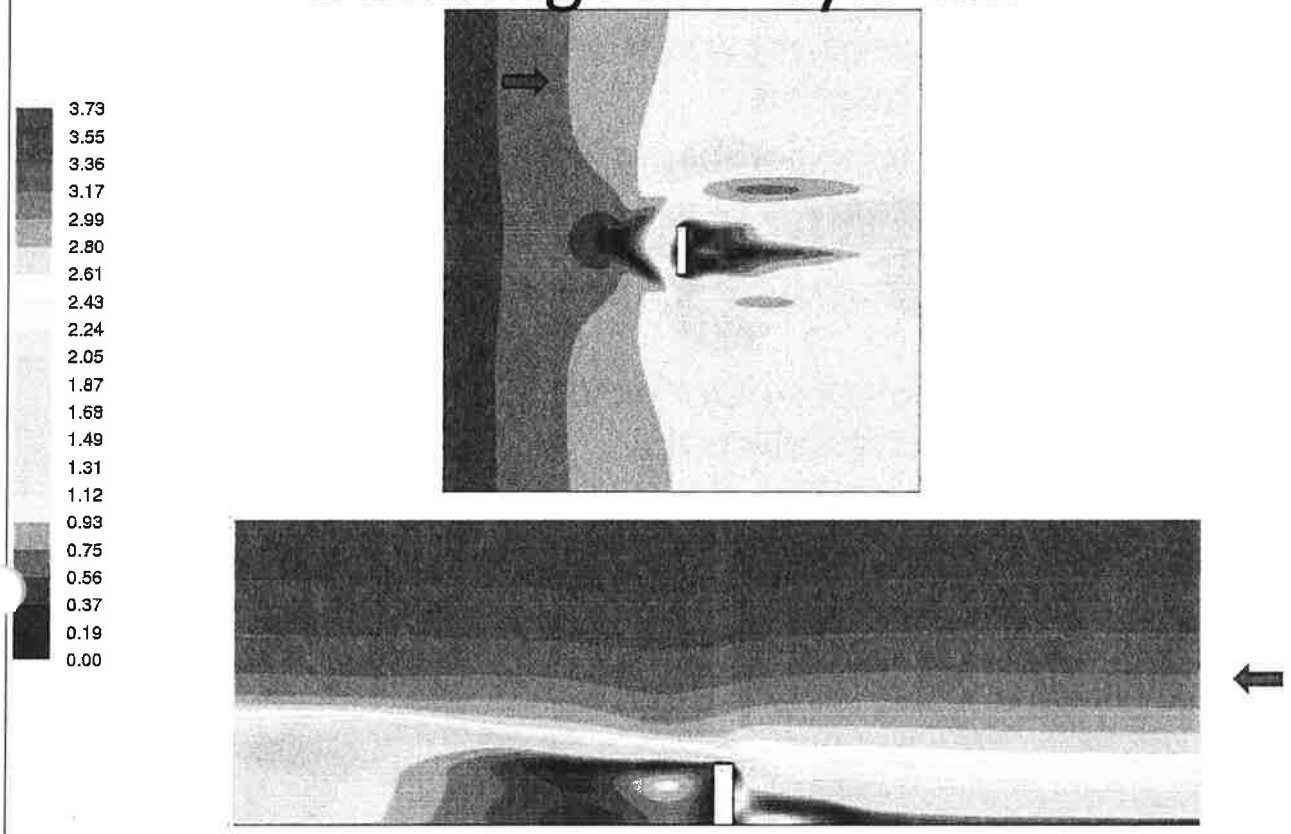
Building Aerodynamics



Building Aerodynamics

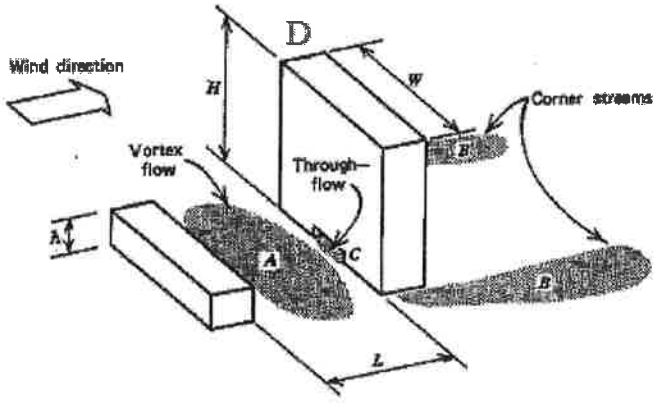


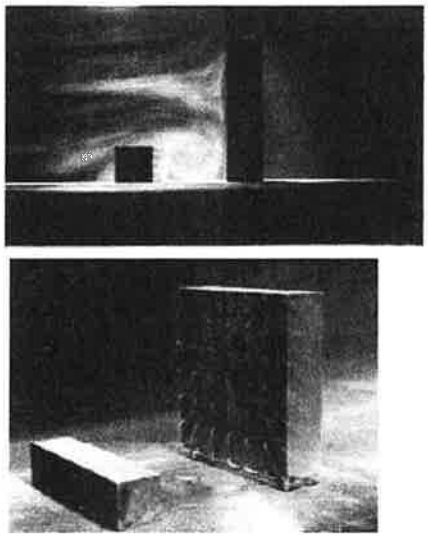
Building Aerodynamics



Wind Effects on Building Structure

- Wind Flow over 2 Buildings, with one smaller building in front of the big building
- Three main types of surface winds:
  - Vortex flows – arise from wind deflected downwards by building
  - Corner streams – arise from wind accelerated around corners
  - Through flows – arise from wind sucked through openings





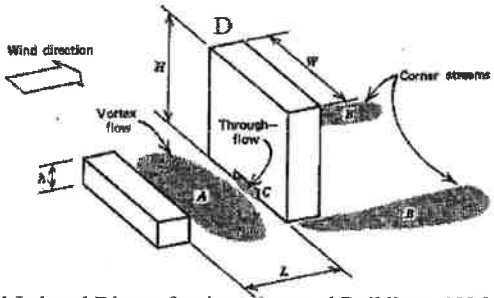
E. Simiu and R. Scanlan, Wind Effects on Structures, 1996

Wind Effects on Building Structure

- Surface winds expressed in terms of speed ratio, $R_H=V/V_H$, where V is wind speed at pedestrian height and V_H is wind speed at building height H
- The wind speed was experimentally measured and related at 10m in open country

H (m)	20	30	40	50	60	70	80	90	100
$\frac{V_H}{V_0}$	0.73	0.82	0.89	0.94	0.99	1.04	1.08	1.11	1.14

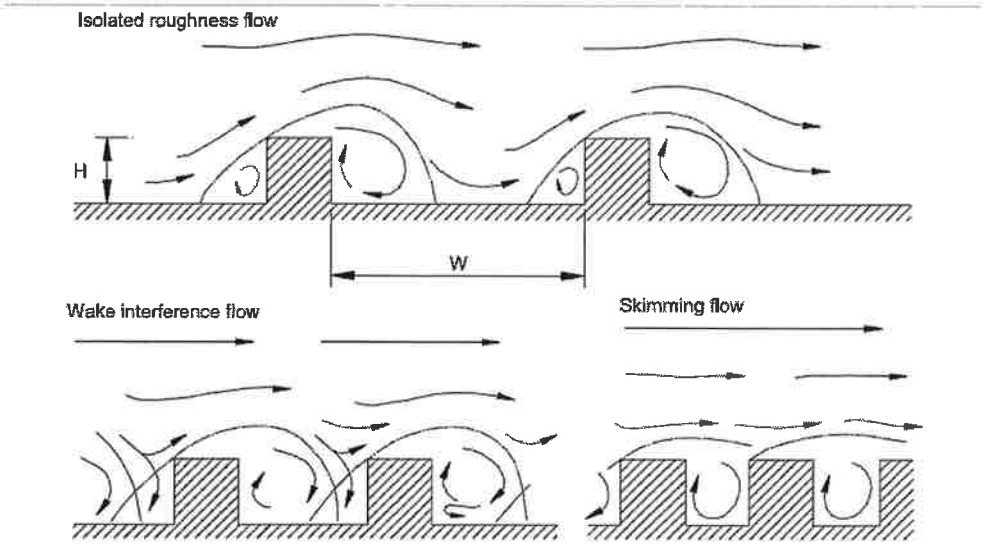
- Wind tunnel studies of surface winds around a building complex have been conducted
- Dimensional analysis gives R_H= f(L/H, W/H, D/H, H/h, $\rho VH/\mu$)



E. Simiu and R. Scanlan, Wind Effects on Structures, Chp. 15 Wind-Induced Discomfort in and around Buildings, 1996

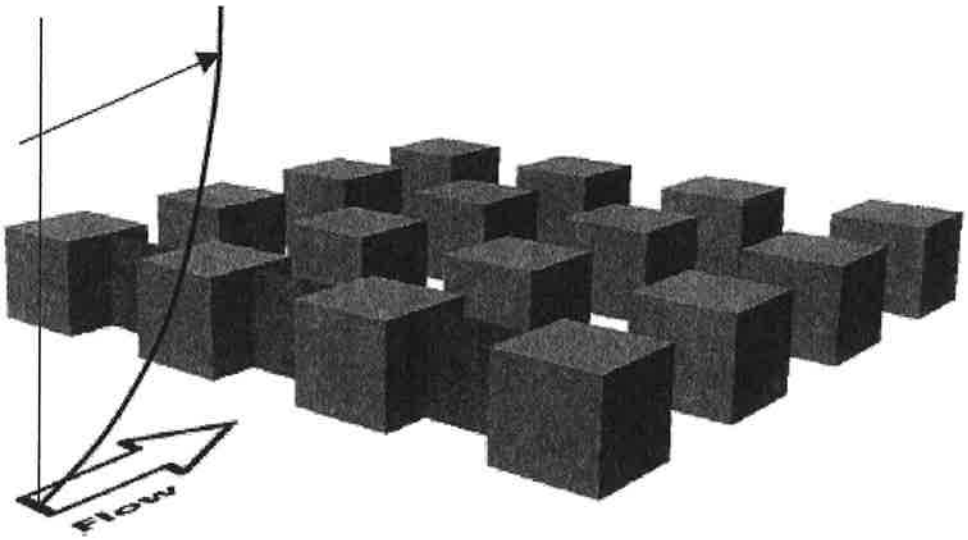
Urban Street Canyon Flow

- Wind in the urban canopy – wind flow regimes
  1. Isolated Roughness Flow (IRF) → H/W<0.3
  2. Wake Interference Flow (IRF) → 0.3<H/W<0.7
  3. Skimming Flow (SF) → H/W>0.7



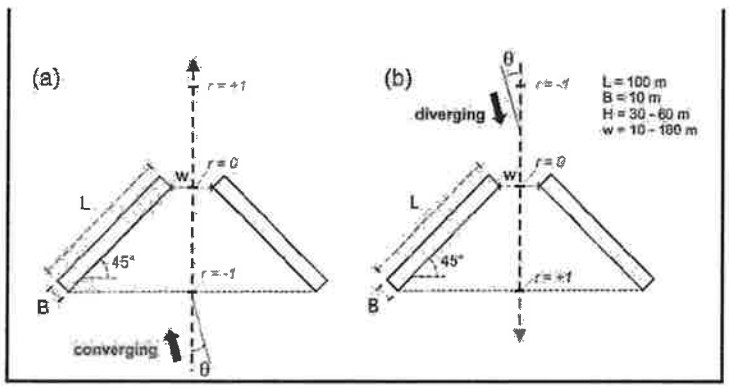
Oke, T. R.: 1988, **Street Design and Urban Canopy Layer Climate,** *Energy and Buildings, 11* (1988) 103 - 113

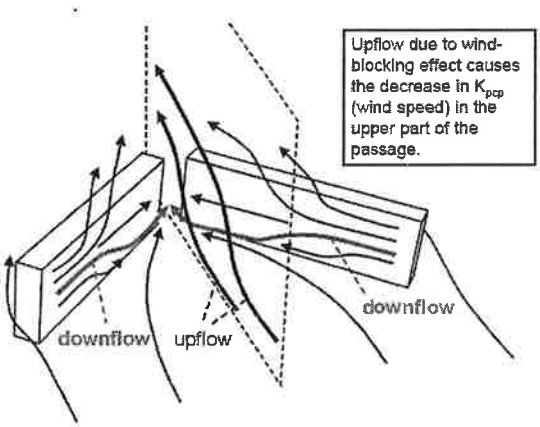
Urban Like Flow Around Building Blocks



1. Jonas Allegrini a,*, Viktor Dorer a, Jan Carmeliet, Influence of morphologies on the microclimate in urban neighbourhoods, Journal of Wind Engineering & Industrial Aerodynamics, 2014

Venturi Effect In Between Buildings – Wind Flow Over V Shape channel between two buildings

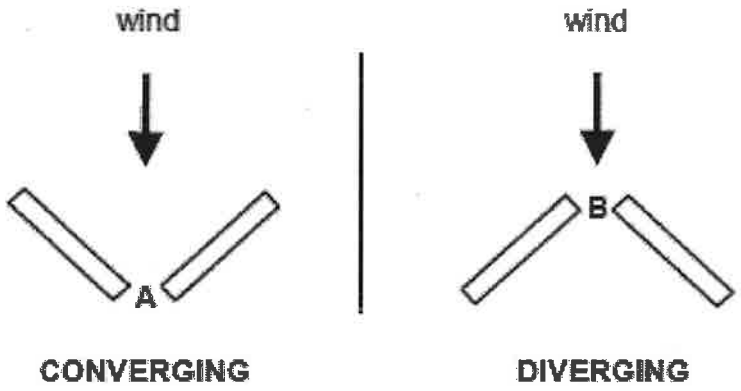




1. Blocken B, Moonen P, Stathopoulos T, Carmeliet J. 2008. A numerical study on the existence of the Venturi-effect in passages between perpendicular buildings. ***Journal of Engineering Mechanics – ASCE*** 134(12): 1021-1028.
2. Blocken B, Stathopoulos T, Carmeliet J. 2008. Wind environmental conditions in passages between two long narrow perpendicular buildings. ***Journal of Aerospace Engineering – ASCE*** 21(4): 280-287

QUESTION 1: WIND FLOW BETWEEN BUILDINGS

Top view of two high-rise buildings in "V" arrangement. Wind direction from bottom to top. Which configuration yields the highest wind speed in the passage?



CONVERGING **DIVERGING**